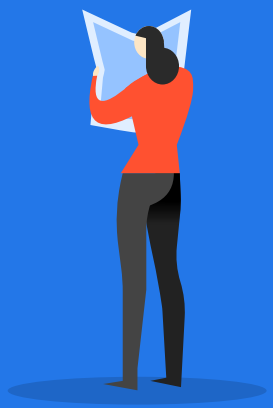
BricsCAD®

Offline Help Center





BRICSYS

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Chapter 1

BricsCAD

1.1 Installation & Licensing

1.1.1 Run your BricsCAD trial as Classic, Pro or Platinum

https://www.youtube-nocookie.com/embed/x2XPFpbC-1w

This video tells you how you can run BricsCAD in a different license level.

BricsCAD is available in three license levels – the entry level – Classic version, the mid-range called Pro and the premium version is called Platinum. The license that you have purchased determines the highest level at which you can run your BricsCAD. Back to Tutorial Overview

1.1.2 BricsCAD Migration – The BricsCAD User Interface

The BricsCAD User Interface

What's different versus AutoCAD®? You may have heard phrases like "**muscle-memory compatible**" to describe the BricsCAD user interface (UI), with regard to AutoCAD's. The command syntax is almost identical. Your scripts and menus generally work in BricsCAD without modification, and your AutoLISP routines just load and run. And of course, if you're a LISPer, you'll be excited to meet BLADE, the BricsCAD LISP Advanced Development Environment. There are some advances in BricsCAD®that might confusea long-time AutoCAD user. Let's talk about them up front, tosimplify your migration journey. First: did you download the free eBook,BricsCAD for AutoCAD Users? If not, please dothis right now. It's written for Bricsys by Ralph Grabowski, who is very fluent in both products.

The BricsCAD Start Screen

The simplified startup screen in BricsCAD makes it easy for new users to get going.The BricsCAD "Get Started" dialog sports a simple, single-pane layout. It lets you select an existing drawing, or create a new one from a template. You can also select your drawing units from the start screen.Wondering what's new in the latest release? It's easy to get to our (comedy) award-winning Release Notes and our on-line tutorials from the startup screen, too. Don't like it? Turn it off or on with the GETSTARTED setting.

Inside the Main Window

Are you one of those "**left hand on the keyboard, right hand on the mouse**" CAD users? The BricsCAD command prompt uses a simple ":" character to tell you that it's ready for your input. You can change the BricsCAD setting "CMDLNTEXT" to any text string you want.BricsCAD's prompt menu is a floating menu box that pops up whenever a command supports additional options. You can use it to pick command modifiers (or simply key-in the modifier, as you would in AutoCAD). You can control the display of the prompt menu – go to SETTINGS, search for "Prompt menu", or just type in PROMPTMENU. I set it to "4", so that the prompt menu appears near my command window. The biggest UI enhancement inBricsCAD has to be the Quad cursor. It's a fully customizable, context-sensitive "head-up"tool palette. The Quadbrings relevant commands directlyto your cursor. For more information on the Quad, go to Chapter 2, Page57 of the Grabowski book (you**DIDdownload**it, right?). – or check out the extensive HELP on "The Quad Cursor Menu" in Chapter 1 of the BricsCAD User Guide.

The BricsCAD SETTINGS Dialog Box

All 954 BricsCAD Settings (options + "sysvars") live here!

All BricsCAD User Interface options and settings are controlled from a single dialog box, named… SETTINGS. The search function in the SETTINGS dialogis like magic.Type in the first few letters of asetting's name, title or description and your cursor will jump to that setting automatically. More than one? Move to the next using the arrows next to the search text box. The search box changes color toshow when you've typed an invalid string (or run to the end of the options). In BricsCAD, you can export all of your current settings to a CSV file by clicking a button in the SETTINGS dialog. A comprehensive overview of the SETTINGS dialog is in "BricsCAD for AutoCAD Users", Chapter 2, starting on page 60. I recommend Ralph's guided tour so that you aresavvy with this important BricsCAD feature.Once you use this powerful tool, you'll be smitten!

"Panels"versus "Palettes" The menu elements called "palettes" in AutoCAD are called "panels" in the BricsCAD user interface.Some of your favorite AutoCAD palettes appear to be missing in BricsCAD, until youdiscover that they have been centralized into the BricsCAD Drawing Explorer. This tool gives you all the necessary information about your open drawing(s), and the ability to copy data between drawings via drag and drop. Just type "EXPLORER" at the command prompt or invoke a related command like "LAYER" or "XREF". The Properties panel in BricsCAD lets you discover data about entities, or edit entity properties. You can turn on the Properties Bar by entering "PROPERTIES" at the command prompt, or right-clicking in any toolbar (or the ribbon) and selecting "Properties".

Rollover Tips

It's important to know that, in the BricsCAD user interface, the Properties panel isn't the only way to get information on the entities in your drawings. Rollover Tips in BricsCAD appear on your cursor when activated (via the "RT" switch in the Status Bar). You can use the CUSTOMIZE (CUI) tool to set up the properties displayed in the rollover tip for each entity type. Tool palettes in AutoCAD and tool panelsin BricsCAD operate very similarly, but customization works a bit differently. For more info, see Chapter 4 of Ralph's book, starting on page 121. While AutoCAD tool palettes use a file extension of "ATP", BricsCAD palette bars use "BTP". They are both stored as XML data, and you can import ATP files into BricsCAD.

More on the BricsCAD User Interface Next installment, we'll pick up with the differences in Sheet Sets between AutoCAD and BricsCAD. In the meantime, please feel free to "read ahead" in Ralph's book. The more you know, the more you'll like BricsCAD. You can never know too much about the world's best CAD software.Try BricsCAD V18 Today!

Tutorial: Exploring the BricsCAD User Interface Click here (1.2.1) to watch Back to Tutorial Overview

1.1.3 BricsCAD License Server Setup

https://www.youtube-nocookie.com/embed/WvCh0tjDEGg

Overview on setting up BricsCAD's network license server

The video explains how to set up a BricsCAD network license server and what advantages comes withit.

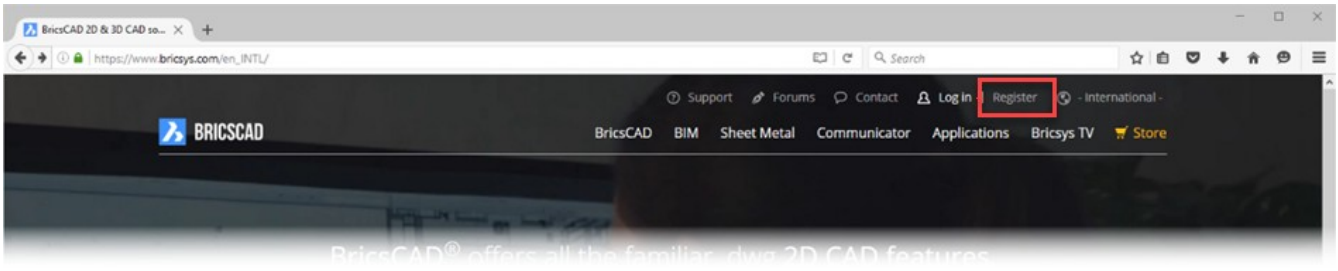
Share BricsCAD with multiple users You can run a network license server allows BricsCAD to be shared amongst multiple users. This is a great option for people that need to utilise CAD as part of their work process. It is also extremely cost effective.

A Bricsys account provides youtwo key advantages If you're not the License owner, then it's recommended that you first create a user account with Bricsys. This will provide you with two key advantages:

1. Firstly, It will allow the license holder to assign you as the licence manager, meaning you can manage their BricsCAD license on their behalf;
2. Secondly, should you strike any issues with administering the network license, you will be able to directly access next business day support from Bricsys.

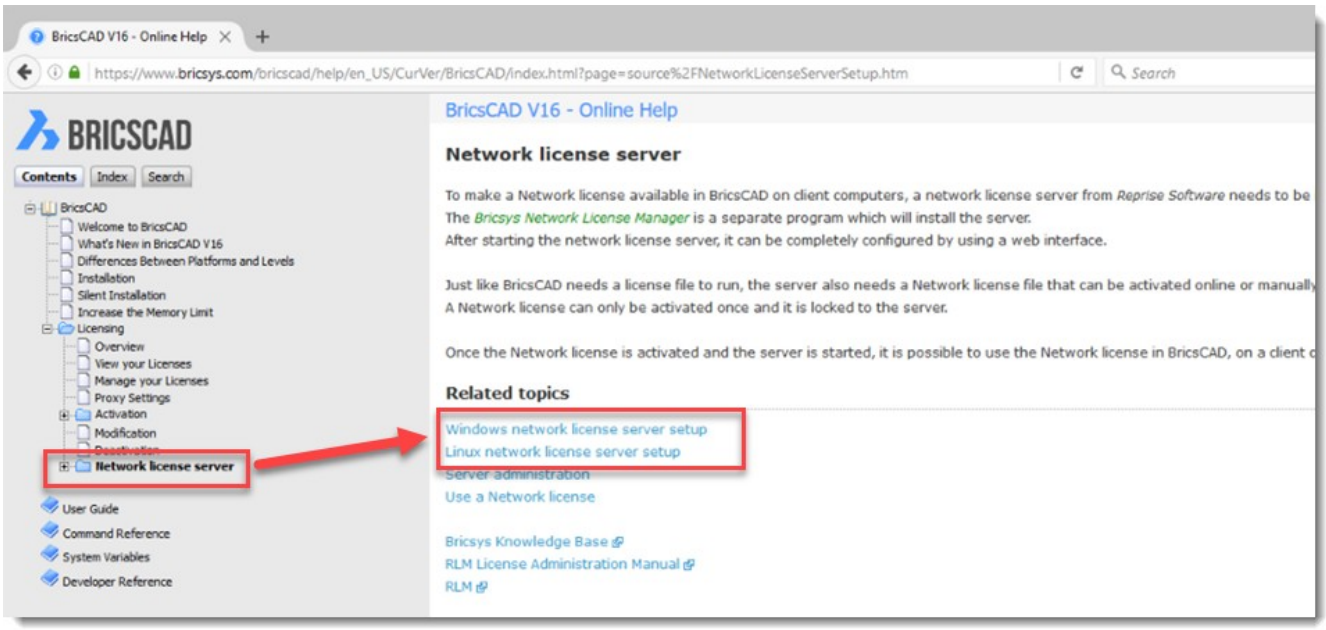
(\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2016/12/imageContent3-1-2.jpg})

Figure 1.1



License servers for Windows and Linux There are license servers available for Windows and Linux. Details on how to set the server up can be found in the online help (1.10.1). This also includes links to download the server installer. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2016/12/imageContent5-1.jpg})
Back to Tutorial Overview

Figure 1.2


BricsCAD V16 - Online Help
https://www.bricsys.com/bricscad/help/en_US/CurVer/BricsCAD/index.html?page=source%2FNetworkLicenseServerSetup.htm
Search
BRICSCAD
Contents
Index
Search
BricsCAD
Welcome to BricsCAD
What's New in BricsCAD V16
Differences Between Platforms and Levels
Installation
Silent Installation
Increase the Memory Limit
Licensing
Overview
View your Licenses
Manage your Licenses
Proxy Settings
Activation
Modification
Network license server
User Guide
Command Reference
System Variables
Developer Reference
BricsCAD V16 - Online Help
Network license server
To make a Network license available in BricsCAD on client computers, a network license server from Reprise Software needs to be
The Bricsys Network License Manager is a separate program which will install the server.
After starting the network license server, it can be completely configured by using a web interface.
Just like BricsCAD needs a license file to run, the server also needs a Network license file that can be activated online or manually
A Network license can only be activated once and it is locked to the server.
Once the Network license is activated and the server is started, it is possible to use the Network license in BricsCAD, on a client c
Related topics
Windows network license server setup
Linux network license server setup
Server administration
Use a Network license
Bricsys Knowledge Base
RLM License Administration Manual
RLM

1.1.4 How to roam with a network license

https://www.youtube-nocookie.com/embed/98zt4duxRlI

Book your BricsCAD license out

If you're running BricsCAD with a network license, then you have the option of roaming.Roaming allows you to book your BricsCAD license out, providing you a way to continue using BricsCAD without a network connection.This video will show you how to do this. Back to Tutorial Overview

1.2 Interface

1.2.1 Exploring the BricsCAD User Interface

https://www.youtube-nocookie.com/embed/YebY9ibxuo0

This tour briefly shows how BricsCAD is the same as — and different from — AutoCAD.

> Once you get a closer look, you begin to see ways that BricsCAD is different — and better — than AutoCAD

In many ways, BricsCAD is similar to AutoCAD in looks and operation. But once you get a closer look, you begin to see ways that BricsCAD is different — and better — than AutoCAD. This video gives you a quick look at the similarities and differences, and highlights the areas where BrisCAD has features that are lacking in AutoCAD. Back to Tutorial Overview

1.2.2 How to navigate quickly around your drawing

Youtube This video shows you how to use Viewswith 2D drawings, how to plot with named views, and the extra functions available for 3D models, such as setting individual visual styles, perspective mode, and backgrounds.

Views let you switch instantly between zoomed in areas of the drawing and back to the full drawing.

Back to Tutorial Overview

1.2.3 Maximizing the Drawing Area with Clean Screen

https://www.youtube-nocookie.com/embed/Xmn5F140KSw BricsCAD has a lot of user interface elements for you to access information and carry out commands -- menu bar, ribbon, toolbars, panels, command bar, status bar, scroll bars, and more. Fortunately, it provides a way to dismiss them all to maximize the drawing area. In this tutorial, you learn to press Ctrl+0 (or Cmd+0 of MacOS devices) to maximize the drawing area and then return to the normal interface. You also learn where in the Settings dialog box you control which user interface elements are hidden -- and which are shown -- in clean screen mode.

1.2.4 On-screen Tips Widget in BricsCAD

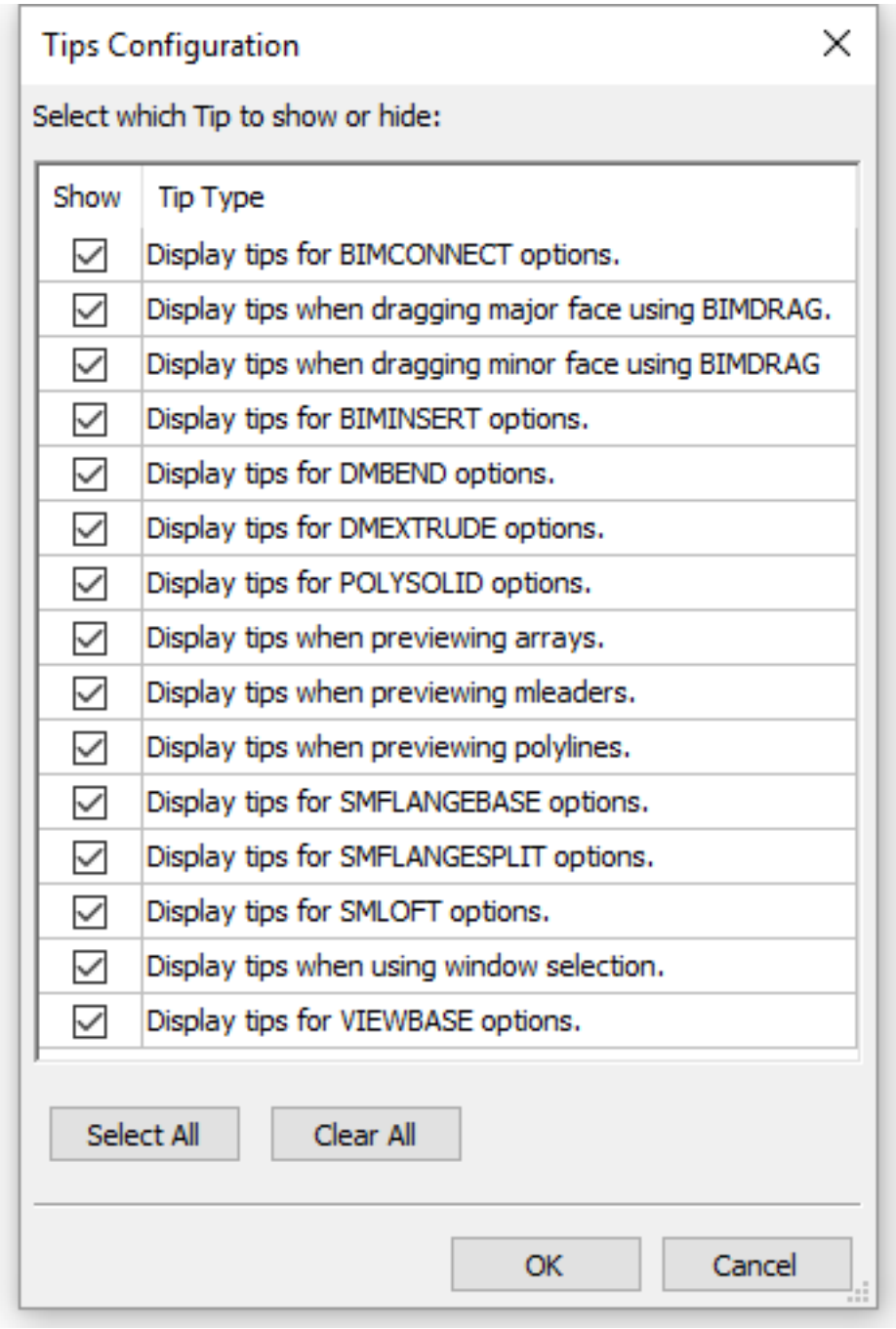
https://www.youtube-nocookie.com/embed/vGn81ExUU2o

The on-screen Tips widget

Many CAD systems – including BricsCAD – use the Ctrl key to change how a drawing or editing command operates. But how doyou know when more options are available?BricsCAD provides a on-screen Tips widgetthat shows your options. For example, pressing Ctrl while selecting a solid changes the entities selected from the entire solid to specific faces or specific edges.

Where is the Tips widget? The Tips widget pops up during multipleactions in BricsCAD. It is availablewhen dragging major/minor faces using BIMDRAG, for POLYSOLID options orfor DMEXTRUDE options. In the video,Ralph Grabowskishows you howTipscan aid you duringwindow selection.You can view all the possible actions that display Tips inside the Tips Configuration dialog box. To see this dialog, simply right-click on “TIPS” on the extreme right-hand side of the Status bar, and click “Configure”. As your skill with BricsCAD increases, you may find that you don’tneed to see a specificTip. Click the ‘x’ while the Tip isactive and you’ll dismiss that specific Tip for good. Of course, you can always activate it again in the dialog box. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/08/00TipsConfigurationDialog.png}) BricsCAD V17 introducednew Tips for polylines, multileaders and associative arrays. In the video, Ralph shows howthe Ctrl keyworks when you are selecting any of these entities. While hovering over one of these entities, the Ctrl key gives you the option selectthe entire entity, or subentities within the larger entity. This functionality is consistent with the way that Windows works, and it gives you a simple way to get your work done more easily and… faster. Need more information about how Tips work in BricsCAD? You can find more about the Tips widget**at the bottom of the** User Interface page (1.12.2)in the BricsCAD On-line Help. As long as you’re on the User Interface page, explore some of the other key features of BricsCAD – there’s a lot there that you may not know about! Back to Tutorial Overview

Figure 1.3


Tips Configuration
Select which Tip to show or hide:
Show
Tip Type
Display tips for BIMCONNECT options.
Display tips when dragging major face using BIMDRAG.
Display tips when dragging minor face using BIMDRAG
Display tips for BIMINSERT options.
Display tips for DMBEND options.
Display tips for DMEXTRUDE options.
Display tips for POLYSOLID options.
Display tips when previewing arrays.
Display tips when previewing mleaders.
Display tips when previewing polylines.
Display tips for SMFLANGEBASE options.
Display tips for SMFLANGESPLIT options.
Display tips for SMLOFT options.
Display tips when using window selection.
Display tips for VIEWBASE options.
Select All
Clear All
OK
Cancel

1.2.5 Experience the Quad Cursor in BricsCAD

https://www.youtube-nocookie.com/embed/j0dQ63Bd6Ng

The best UI enhancement ever – the BricsCAD Quad cursor!

Imagine an entire command system located directly on your mousecursor. No need to dream it – we're talking about the BricsCAD Quad cursor! There's no easier way to select commands in any CAD system.Move your cursor across the screen, and the Quad detectsthe entity underneath the cursor. Based on this, the Quad presents commands that work on the entities you select, saving you time and effort. You can execute an entire range of commandswith a single click from the**Quad-based command palette**. Ralph Grabowski'sshort video tutorial shows you how to use the Quad to place hatch patterns with a single click, and how to edit that hatch pattern with a double-click. He also shows you how to place dimensions and dimensional constraints on entities with a single click of a Quad icon. Finally, the single-click command concept applies to offset entities; and just about any other command that you can imagine. With the Quad, you can execute complex commands that normally require several clicks, options, and dialog boxes – simply and easily.

A lot more to the Quad cursor...

There's a lot more to the Quad cursor in BricsCAD, but the first step is just to turn it on – in BricsCAD, go down to the status bar in the lower-right corner of your screen and click "QUAD". Then go up and hover over some on-screen entities and, Bingo! If you right-click the "QUAD" box on the status bar, you'll get direct access to the Quad's Settings and Properties. You can find out more about the Quad cursorhere in the BricsCAD on-line documentation (1.12.5). Once you have basic operations under control, you might want to customize your Quad cursor. Just move your cursor over any toolbar, right-click and select "Customize". Back to Tutorial Overview

1.2.6 Tips and Tricks for Drawing Explorer

https://www.youtube-nocookie.com/embed/iyTZJJmoViE

Here are a half-dozen tips for making better use of Drawing Explorer in BricsCAD.

Tip #1. Directly access sections through command names, like Layer Tip #2. Use the blue dot specific the current style or property. Tip #3. Click the Regen button to see the effects of changes. Tip #4. Layers node shows previews of each layer. Tip #5 Drag elements between drawings Tip #6. Right-click the names of blocks, layers, layer states, xrefs, and attachments to see bonus options

Bonus Trick: How to place blocks that are aligned with angled walls.

Back to Tutorial Overview

1.2.7 Those Five Other Selection Modes in BricsCAD

https://www.youtube-nocookie.com/embed/0RgInrXDIVU Back to Tutorial Overview

1.2.8 Use Layer States instead of Clicking Bulbs

https://www.youtube-nocookie.com/embed/GV-z6vdLPZs

Layer States allow to switch quickly between your layers

If you are tired of clicking a lot of bulbs to switch between your drawing layers, then you will love this feature. Back to Tutorial Overview

1.2.9 BricsCAD Spreadsheet Import Techniques

https://www.youtube-nocookie.com/embed/mfP54UP_2Bo

You have a spreadsheet. Here's how to bring it into BricsCAD!

This four-minute video is narrated by industry expert Ralph Grabowski ofWorldCADAccess.com. Got spreadsheets that you want to bring into BricsCAD? This short tutorialgives you two options for importing them directly. The first method isto use the Table command's new File option. You can use this feature to import**any**spreadsheet file. The video shows how to save yourexisting spreadsheet as a CSV (comma-separated values) file. Then, you can import the CSVinto yourdrawing by leveragingthe TABLE command. This method is good for static data that you don't plan to change on a regular basis. If you use Microsoft Excel, the second method gives you a powerful benefit: any formulae in the spreadsheet are retained as FIELD text. To do this, you'll use your computer'scopy and paste functionality. First, in the spreadsheet application, you simply copy the cells you need to the system Clipboard. Next, use the PasteSpec (Paste Special) command to insert the spreadsheet into BricsCAD as a linked object. The key benefit of the second method? You can edit the spreadsheet's contents and have thoseedits updated automatically in your drawing.You can test it out for yourself usingthissample spreadsheetfile,available viaour Chapoo cloud collaboration service. Back to Tutorial Overview

1.3 Customization

1.3.1 Customizing the Ribbon

https://www.youtube-nocookie.com/embed/HSTdIIdFeqY The ribbon is the most prominent element of BricsCAD's user interface, but customizing it is not as straight-forward as, say, a toolbar. After completing this video, you will be able to:

- Identify the parts of the ribbon.
- Add a button to a panel.
- Add a panel to a tab.

1.3.2 User profiles

https://www.youtube-nocookie.com/embed/mEPm7aelSs8 The user profiles customizes BricsCAD for the way you like to have it; such as what toolbars you want to see, the color of the screen etc.. After you customize the way BricsCAD works and looks, you can copy the settings to other computers so that BricsCAD works and looks the same way there for you. After completing this video, you will be able to:

- Understand the role of user profiles in BricsCAD.
- Create your own user profile(s).
- Know how and when to employ user profiles.
- Export and import .arg profile files.

1.3.3 Customizing settings

https://www.youtube-nocookie.com/embed/jvGu8CmjrQs Many commands have default settings that you can change in BricsCAD. After completing this video, you will be able to:

- Use Settings dialog to change BricsCAD's workings.
- Access settings quickly through the status bar.
- Modify settings of editing commands.
- Change the way dynamic input works.

1.3.4 Customizing display settings

https://www.youtube-nocookie.com/embed/wzkGnJ90REI BricsCAD lets you change many aspects of its look, from the color of the drawing area to the sizes of UI elements. After completing this video, you will be able to:

- Use Settings dialog to change BricsCAD's look.
- Change the background color.
- Modify the look of the Quad.
- Adjust the color and linetype of the rubberband.

1.3.5 Controlling BricsCAD Through Settings

https://www.youtube-nocookie.com/embed/fKWpnys7Q8k BricsCAD allows you to change the way it works and looks, and most of these changes are made through the Settings dialog box. After completing this video, you will be able to:

- Use the Settings dialog box effectively.
- Change the pickbox size in Settings.
- Set variables at the command line.

1.3.6 Settings Dialog in BricsCAD

Command: Settings (6.27.23) Youtube

Our Settings Dialog is better than Options. Here is why.

AutoCAD's Options dialog box provides access to the values of many system variables, but not all. BricsCAD does it better. In BricsCAD, the equivalent is the Settings dialog box and its design lets you access all 770 variables easily through an interactive search box: just type the first few characters of a name, title, or description of a variable, and BricsCAD jumps to the first instance in realtime. This video also explains the other functions of the Settings dialog box. Back to Tutorial Overview

1.3.7 Working with Tiled Viewports in Model Space

https://www.youtube-nocookie.com/embed/bVxlrON1ffo

Working with Viewports in BricsCAD

The drawing window in model space can split into two or more viewports. This lets you see the same drawing in several different ways, for example, zoomed-in in one viewport, and with different layer settings applied in another.We willshow you how to create and delete viewports, draw entities that start in one viewport and end in another, switch between multiple, create a new viewport inside an existing viewport, and merge adjoining viewports together. Viewports are created with the VPORTS command. Default options let you create 2, 3, or 4, and you can orient them vertically or horizontally. The dominant (active) viewport is surrounded by a heavy dark line. That's where the cursor appears and where things happen when you enter a command. Want to switch between them? Simply click in another viewport or press Ctrl+R to cycle between them.

2D Drawings Get an overview of your drawing in one viewport and a detailed view of your plan in another. Also, you can draw from one viewport to another. Start a line in the zoomed-in detail, and end it in another viewport.

Tips for Viewports and 3D Drawings When you are modeling in 3D, you can easily create 4 views – top, front, side and isometric – with the VPORTS command. Control your view angles using the "Look from" widget. Eachviewport can display its owndifferent visual style.

Want more than 4? Select the dominant viewport and enter the VPORTS command. You can savethe viewport layout if you like the set up. Enter the VPORTS command, and then enter the 's' sub-command and type in a name for your viewport setup. You can**join multiple viewports**. Enter VPORTS command, enter 'j', select the dominantone and select the viewport you wish to join it with.

Cancel the viewports Enter the VPORTS command, enter 'SI' and you're back to one, single view of your model.
Back to Tutorial Overview

1.3.8 Saving Your Customizations with Partial CUI Files

https://www.youtube-nocookie.com/embed/ecmKyfjtCos

Load UI elements independently of BricsCAD's default settings

This video shows you how to customize with partial CUI files instead of the main one, as well as to share your customizations with other BricsCAD users. When you customize BrisCAD through the default.cui file, you can lose all of your work! Unless you use partial CUI files instead; these files are not overwritten when you need to reset the system or install an upgrade. Back to Tutorial Overview

1.3.9 Customizing the Command Prompt

https://www.youtube-nocookie.com/embed/e_C5C-zMEM8

For its command prompt, BricsCAD uses a simple ':' to indicate it is ready for you to enter a command. You can change what BricsCAD displays through the Settings dialog box. Older releases of AutoCAD used 'Command:', and newer releases use 'Type a command'. You can change what BricsCAD displays through the Settings dialog box. Command prompt customization is not available in AutoCAD. Back to Tutorial Overview

1.3.10 Easy Menu Customization in BricsCAD

https://www.youtube-nocookie.com/embed/-gtsGPlpMRc

How to make simple changes to the BricsCAD's menus

The menu bar and menus can be completely customized in BrisCAD. In this video, you learn how to make a new menu to the bar, add commands to the menu, and then to rename and remove menu items. Back to Tutorial Overview

1.3.11 Customizing shortcut menus

https://www.youtube-nocookie.com/embed/-6ihKCNPjHc

Choose the commands you want to have within reach

BricsCAD has the ability to customize your shortcut menu when you right click on an entity. You can edit them by right clicking on your toolbar and choose Customize. Then you'll find the Context menuin the Menu tab, where you can change your shortcut menu per entity. Back to Tutorial Overview

1.3.12 Advanced Menu Customization

Youtube This video explains the parameters and metacharacters that control each menu item: Title, Diesel, Tool ID, Help, Command, and Image. It also gives some tips on what to do, and what to not do. Back to Tutorial Overview

1.3.13 Customizing Tool Palettes in BricsCAD

https://www.youtube-nocookie.com/embed/YvG_wcm9xSQ

It's easy to customize BricsCAD Tool Palettes!

What are tool palettes? They aredockable, graphical panels that contain commands, hatch patterns, or blocks for insertion – or all three! To use a palette object, click the tool and move your cursor to the graphic screen. Does this sound simple enough? It getsbetter – it's almost as easy to customize your own tool palettes, and add your own commands or custom insert-able blocks. In this short video lesson,Ralph Grabowski givesa brief overview of Tool Palettes and how they work. The content of each button (icon) in the palettes can be fully customized to meet your needs. Ralph willgive you the basics on how to create your own custom commands in the video, also.

What are the commands to access tool palettes?

- Type "**ToolPalettes**"at the command line to open the Tool Palettes bar
- Type"**ToolPalettesClose**"to close the Tool Palettes bar
- The "**TPNavigate**"command displays a specific named tool palette or a tool palette group

You can open a tool paletteby right-clicking anyuser interface element. From the the context menu, select "**Tool Palettes**".

How do youcustomize commands in your tool palettes? First, right click on a command icon in atool palette. Then click "Properties". You can change the "command string" (a.k.a. "macro") of an existing icon, or create a new one. In this lesson, Ralph builds a macroby typing this text inthe command string text box: "**^C^C_line;1,1;2;2**". When Ralph clicks on the modified command icon, BricsCAD draws a line from (1,1) to (2,2). In macros, the semi-colon (**;**) acts like you are pressing Enter, or hitting the space-bar. In the video's first example, a single trailing semi-colon is implied. You'll see that this causes the line command to continue. You can end the line command macro by adding "**;;**" – two semi-colons – at the end of the command string.

Customizing entity properties with tool palettes You can customize the style and properties of various BricsCAD entities easily, also. In the video, Ralph edits the Revision Cloud command to change its properties. First, he makes a copy of the Revision Cloud iconin the Draw palette, and pastes it into the Command Tools palette. Then he goes back to the icon's properties to customize the way that the revision cloud appears in the drawing upon insertion. Need more? Click this link toaccess the BricsCAD on-line help forcreating and modifying Tool Palettes (6.29.19)!
Back to Tutorial Overview

1.3.14 Customize mouse buttons to add tools to BricsCAD

https://www.youtube-nocookie.com/embed/FWk79sNscmk

Customizemouse buttons in BricsCAD easily

Like almost everything else in BricsCAD, you can customize mouse buttons to increase your productivity. The left mouse button, of course, performs the most important CAD function – selecting entities. Because of this, you can't customize your left mouse button. Never fear though – the middle and right mouse buttons are fully customizable. Also, using combinations of the CTRL, Shift and CTRL+Shift keys while clicking, you can get a total of eight commands from the middle and right mouse buttons.

How does BricsCAD identify the mouse buttons? (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads
Shot-2017-12-27-at-103241-300x190.png}) **Leftbutton = 3** (**not customizable – as it's always used to pick things**)
Middle button/roller wheel= 2 Right button = 1

Figure 1.4



Customize mouse buttons with the CUI Right-click any User Interface (UI) element (menu, toolbar or panel), or just type in “CUI” at the command prompt to access the UI customization dialog box. Select customize and go to the Mouse tab. There you can program the middle and right mouse buttons for click, CTRL-click, Shift-click and CTRL-Shift-click options. The customization of BricsCAD doesn’t end here, this is just the start! Take a few minutes to watch this video that willshow you how to customize BricsCAD’s tool palettes.

Want the full scoop? We have an eBook for you The author of this video tutorial on how to customize mouse buttons is Ralph Grabowski, industry expert and long-time CAD author. Ralph has written a comprehensive eBook that tells the complete story for current AutoCAD® users moving to BricsCAD®. You can get a free copy of this incredible eBook for no charge on our website. Read through it at your leisure and discover what makes BricsCAD the best value in .dwg-based CAD today. Back to Tutorial Overview

1.3.15 Creating Custom Template Drawings

https://www.youtube-nocookie.com/embed/nc0vXt-COqY

How to start new drawings faster by pre-loading them with content

When you start new drawings in BricsCAD, they are quite a barebones effort: just one layer, no special line types, no nothing. When you start a new drawing project, you typically need to add in the office layer standard, the line types, dimension styles, and other material. You can start new drawings with all that in place through template drawings. This video shows you how to access template drawings, how to add content, and then how to save a drawing as a .dwt template file with the SaveAs command. Back to Tutorial Overview

1.3.16 Clipping Attached Images and PDF Files

Youtube Sometimes photographs, scans, and PDF files show more data than is needed for the drafting task at hand. This video shows you how to cut out parts of these attachments, both using a rectangular and a polygonal (irregular) clipping border. You learn how to turn clipping on and off, and find out why PDF files sometimes do not display. Back to Tutorial Overview

1.3.17 Clipping x-refs and blocks

https://www.youtube-nocookie.com/embed/5a1CXEVX_-M

XCLip Command

This video shows you how to use the XCLip command to cut away a portion of a block and an xref. It demonstrates doing this with a rectangular shape, an irregular polygon, and with a curved shape.

> **Clipping is like cutting out part of a photograph, but in the case of BricsCAD the process can be reversed.**

When blocks and/or xrefs (externally-referenced drawing files) are placed in drawings, you can have BricsCAD show a portion of them — by hiding parts through "clipping." Back to Tutorial Overview

1.3.18 Employing the Display Power of Layer States

Youtube Layers have always been used to segregate information, and now layer states lets you instantly switch the drawing display between various layer combinations. A layer state freezes the layers you don't want to see; it can also change the color and other layer properties in an instant. This video shows you how to name and create Layer States with the Drawing Explorer, and then how to create a toolbar that lets you quickly switch the drawing between layer states.

Use Layer states to switch between the display of different groups of layers

Back to Tutorial Overview

1.3.19 Joining Multiple Entities in BricsCAD

https://www.youtube-nocookie.com/embed/m_JroUT4two

Joining multiple entities is simple with JOIN

In this shortvideo, BricsCAD expert Ralph Grabowski shows you the power ofthe JOIN command. He discusses joining multiple entities, like co-linear lines and concentric arcs, into single objects. Ralph goes into detail with regards to multiple entities with different properties. Which one will "win" the properties "battle" (**beware – spoiler!** It's based on how you pick)? He also explores the conditions under which entities of different types can be joined together into one, and the resultant entity types. This is powerful, especially in drawings that have been imported, or drawings that have been throughmultiple editing sessions. Depending on technique, there are scenarios wherepoly-lines will come in as lines and arcs. You can use the PEDIT command to turns them back into polylines. However, there is a better method when joining multiple entities. The JOIN command can maketwo or more lines into a single line, a connected line and arc into a polyline, two or more arcs into a single arc, and even convert aline and a helix into a spline. JOIN is another great tool in your CAD toolbox, and Ralph shows you how to make it work for you – on your command. Back to Tutorial Overview

1.3.20 The Prompt Menu and Its Many Uses

https://www.youtube-nocookie.com/embed/It9t2eAylu0

Using the Prompt Menu for accessing command options

The Prompt Menu is found in a few CAD systems like BricsCAD but not AutoCAD. This video shows when the Prompt Menu shows up automatically, why it is useful, and how to change its settings through the Settings dialog box. Back to Tutorial Overview

1.3.21 Abbreviating Command Input with Aliases

Youtube BricsCAD lets you enter abbreviations for command names, such as L for drawing lines and E for erasing them. This video tours you through using a variety of aliases in BricsCAD, and then shows you how to create your own. Back to Tutorial Overview

1.4 Drawing

1.4.1 Drawing Accurately: Snap and Grid

https://www.youtube-nocookie.com/embed/JRjBDLXAAks

Using snap allows to specify points at fixed intervals. The Grid command displays a grid pattern in the current viewport. Synchronize snap and grid spacing The 'Snap' option of the Grid command forces the grid spacing to be in sync with the snap spacing. When you then change the snap spacing, the grid follows. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent2-1.jpg}) **Control the appearance of the grid** The appearance of the reference grid is controlled by a series of system variables. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent4.jpg}) Back to Tutorial Overview

Figure 1.5

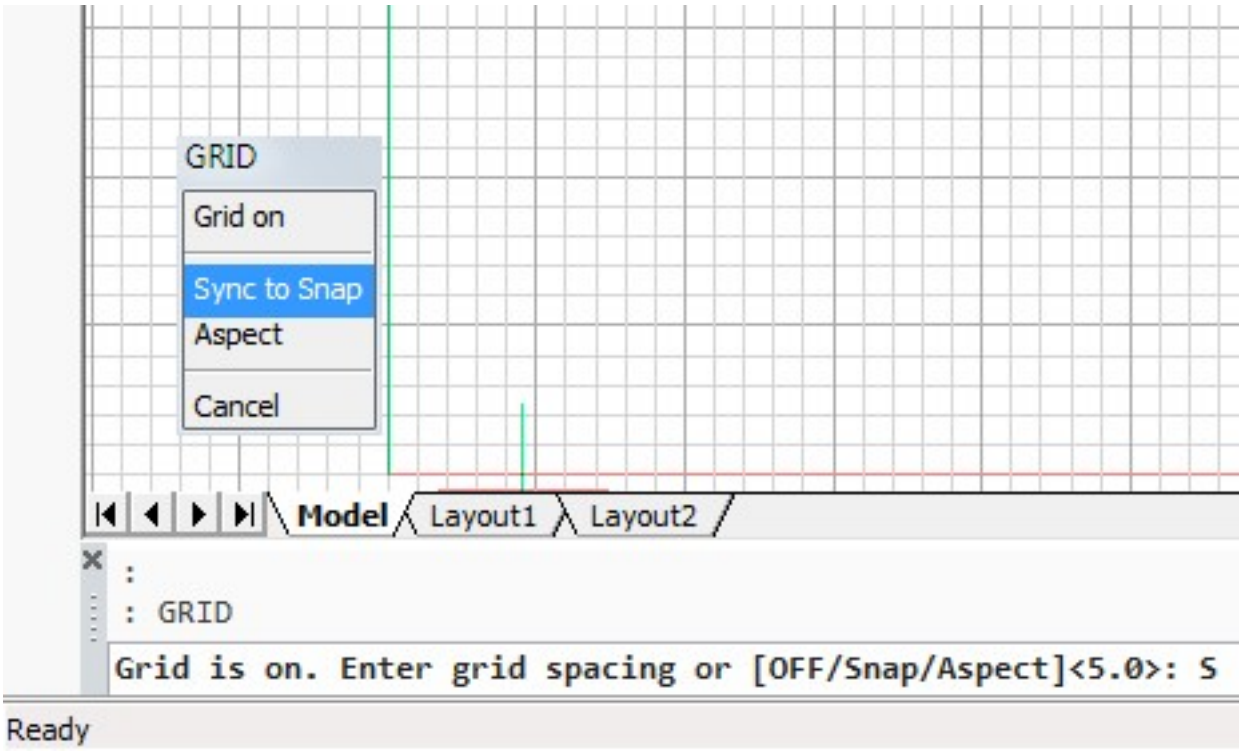
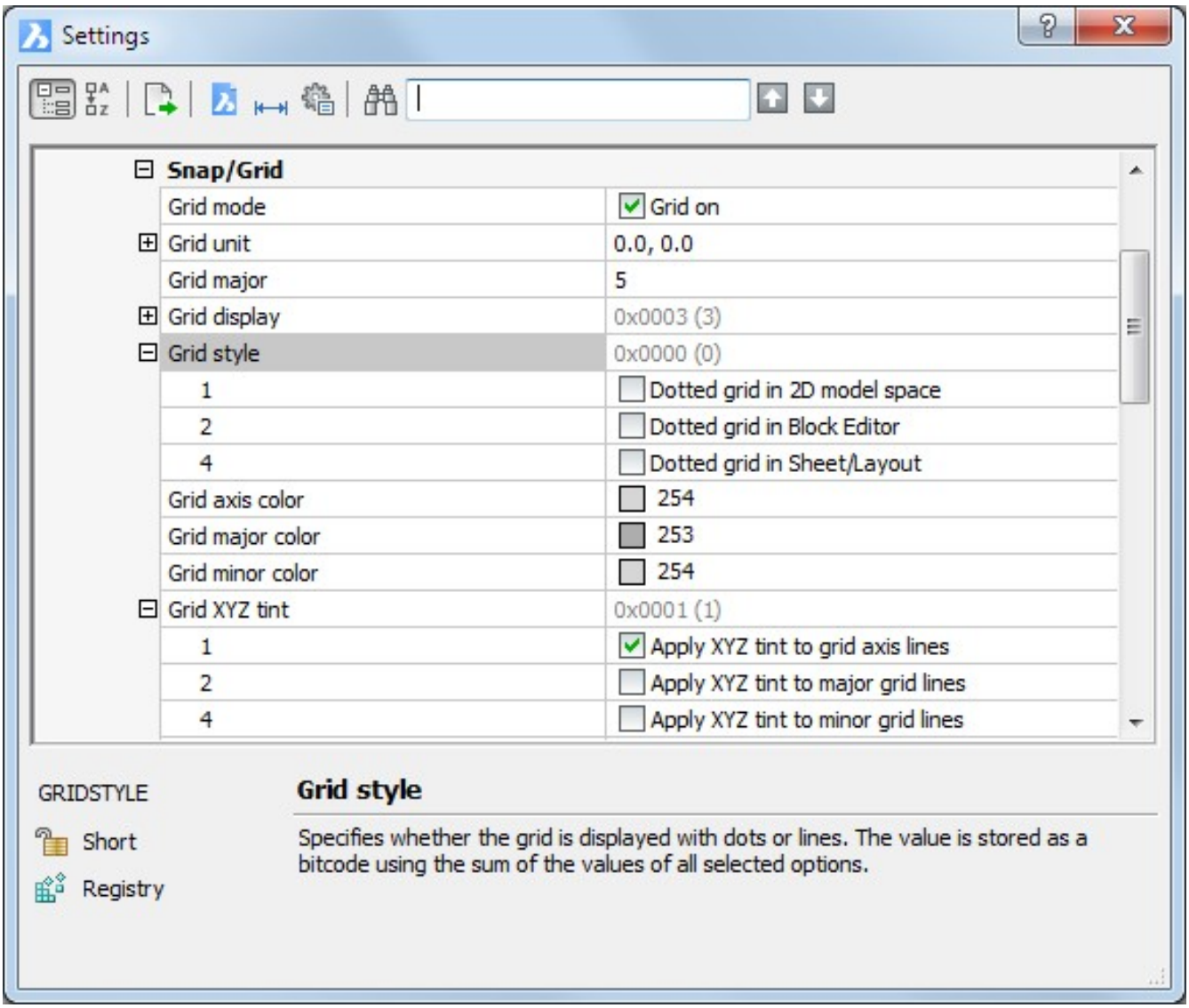

GRID
Grid on
Sync to Snap
Aspect
Cancel
Model
Layout1
Layout2
:
: GRID
Grid is on. Enter grid spacing or [OFF/Snap/Aspect]<5.0>: S
Ready

Figure 1.6

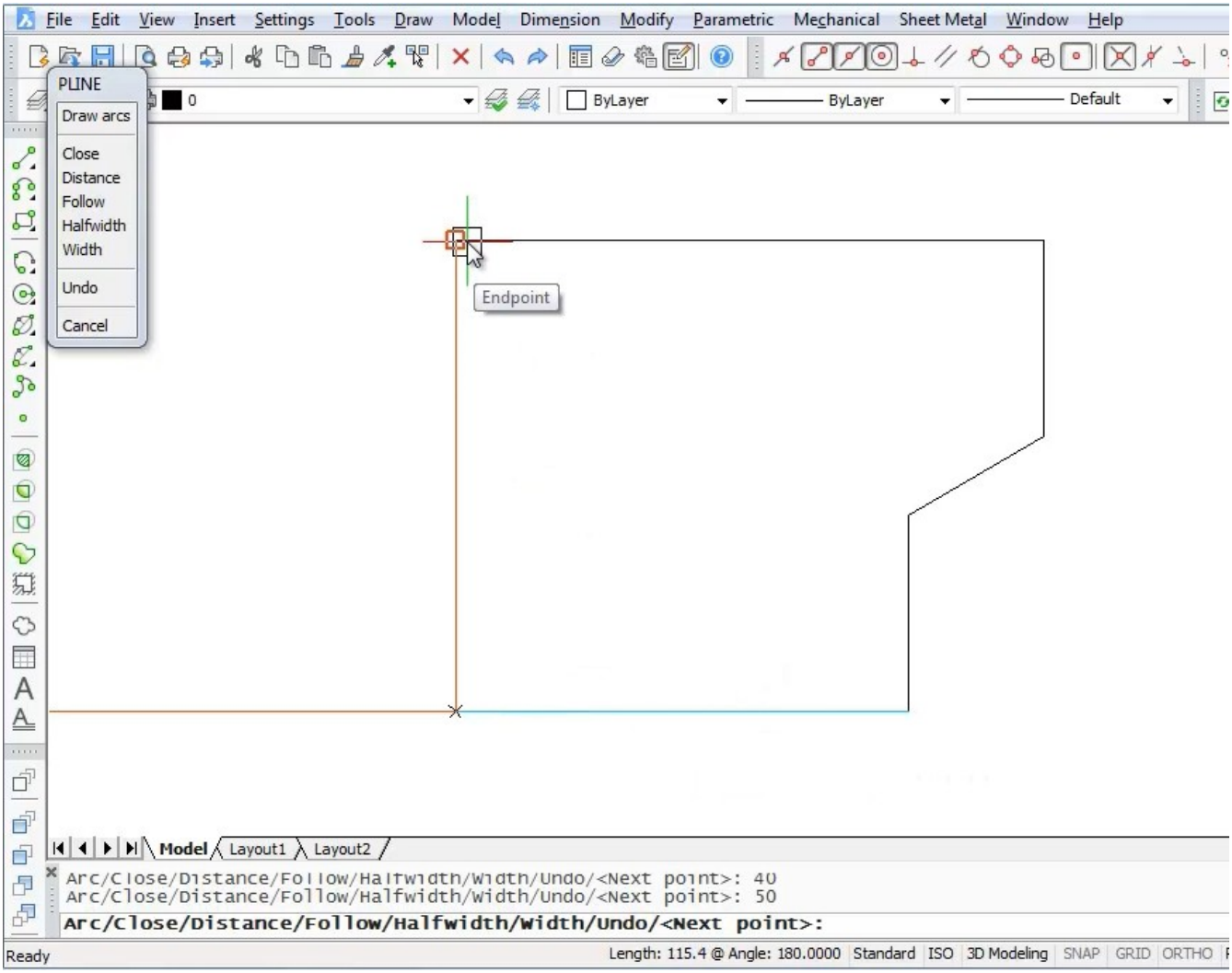

Settings
Snap/Grid
Grid mode
Grid on
Grid unit
0.0, 0.0
Grid major
5
Grid display
0x0003 (3)
Grid style
0x0000 (0)
1
Dotted grid in 2D model space
2
Dotted grid in Block Editor
4
Dotted grid in Sheet/Layout
Grid axis color
254
Grid major color
253
Grid minor color
254
Grid XYZ tint
0x0001 (1)
1
Apply XYZ tint to grid axis lines
2
Apply XYZ tint to major grid lines
4
Apply XYZ tint to minor grid lines
GRIDSTYLE
Short
Registry
Grid style
Specifies whether the grid is displayed with dots or lines. The value is stored as a bitcode using the sum of the values of all selected options.

1.4.2 Drawing Accurately: Polar and Snap Tracking

https://www.youtube-nocookie.com/embed/C3o278SVEQI Polar tracking assists you to draw at exact angles at fixed intervals and optionally at specific additional angles. Entity Snap Tracking assists you to draw or position entities with respect to other entities.

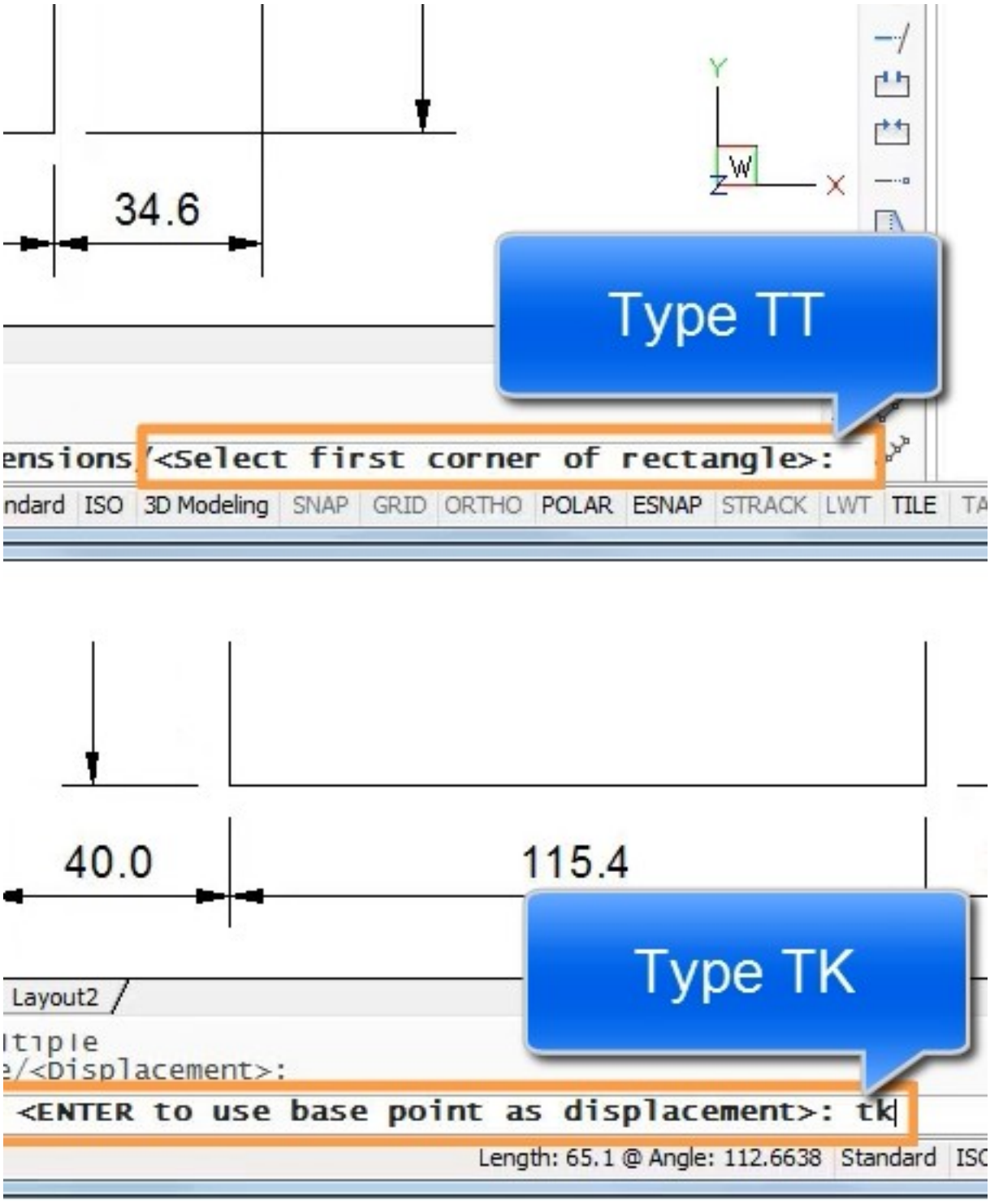
Lock a Tracking Path Hit the Shift key to lock a tracking path . Hit the shift key again to unlock. It works as a toggle.
A locked tracking line turns red. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/Locktrac

Figure 1.7


File Edit View Insert Settings Tools Draw Model Dimension Modify Parametric Mechanical Sheet Metal Window Help
PLINE
Draw arcs
Close
Distance
Follow
Halfwidth
Width
Undo
Cancel
ByLayer
ByLayer
Default
Endpoint
Model
Layout1
Layout2
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 40
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 50
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>:
Ready
Length: 115.4 @ Angle: 180.0000
Standard
ISO
3D Modeling
SNAP
GRID
ORTHO

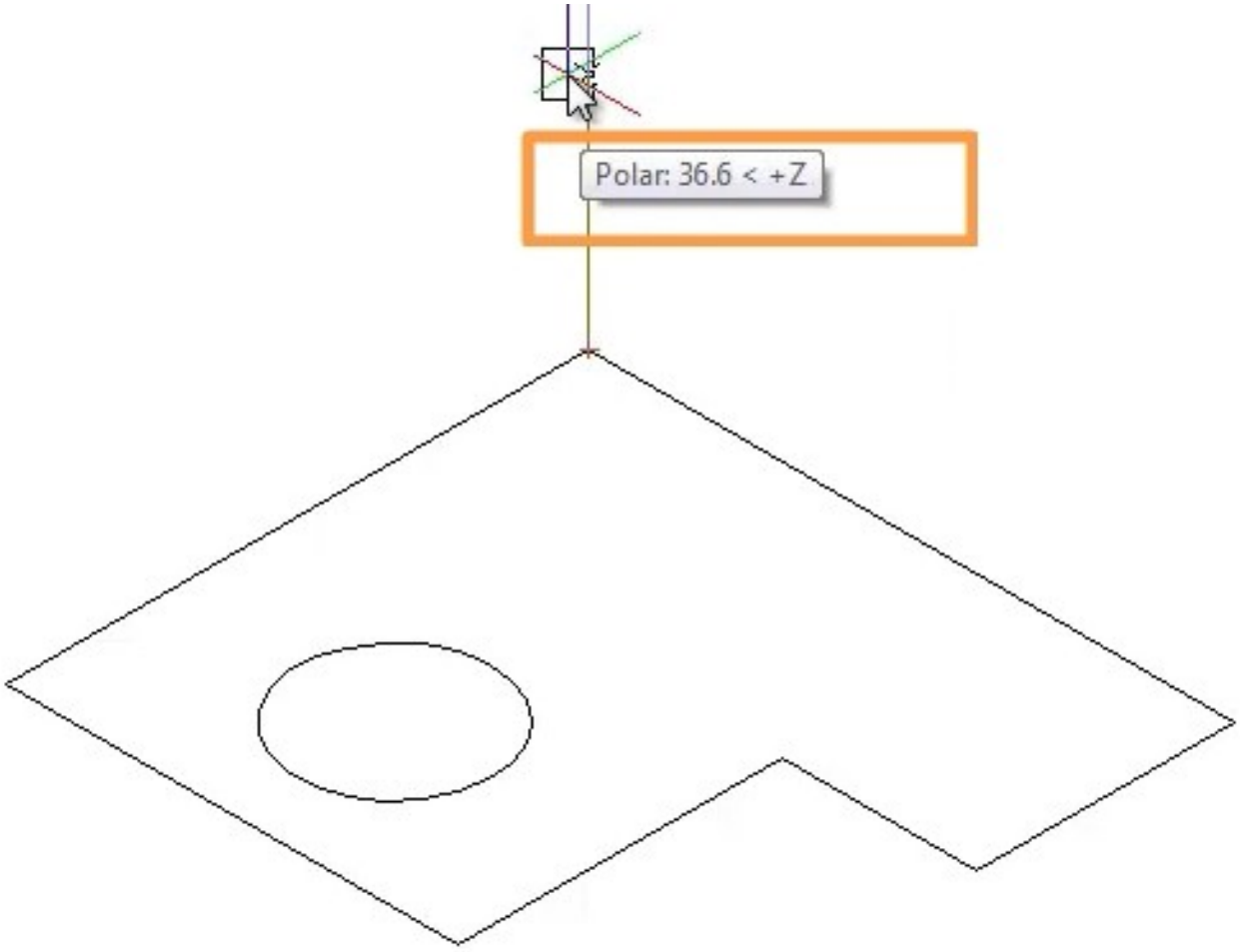
Temporary Tracking Points Whenever BricsCAD prompts to specify a point, type “TT” at the command prompt and BricsCAD asks to specify a Temporary Tracking point. Type “TK” at the command prompt to specify multiple temporary tracking points. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/TemporaryTr

Figure 1.8


34.6
Type TT
ensions/<Select first corner of rectangle>:
ndard ISO 3D Modeling SNAP GRID ORTHO POLAR ESNAP STRACK LWT TILE
40.0
115.4
Type TK
Layout2
ltiple
e/<Displacement>:
<ENTER to use base point as displacement>: tk
Length: 65.1 @ Angle: 112.6638 Standard

Z-direction Tracking Path In 3D views an additional tracking line displays parallel to the Z-axis of the current UCS. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/Z-directionTrackingPath.jpg})
Back to Tutorial Overview

Figure 1.9



1.4.3 Drawing Accurately: Dynamic dimensions

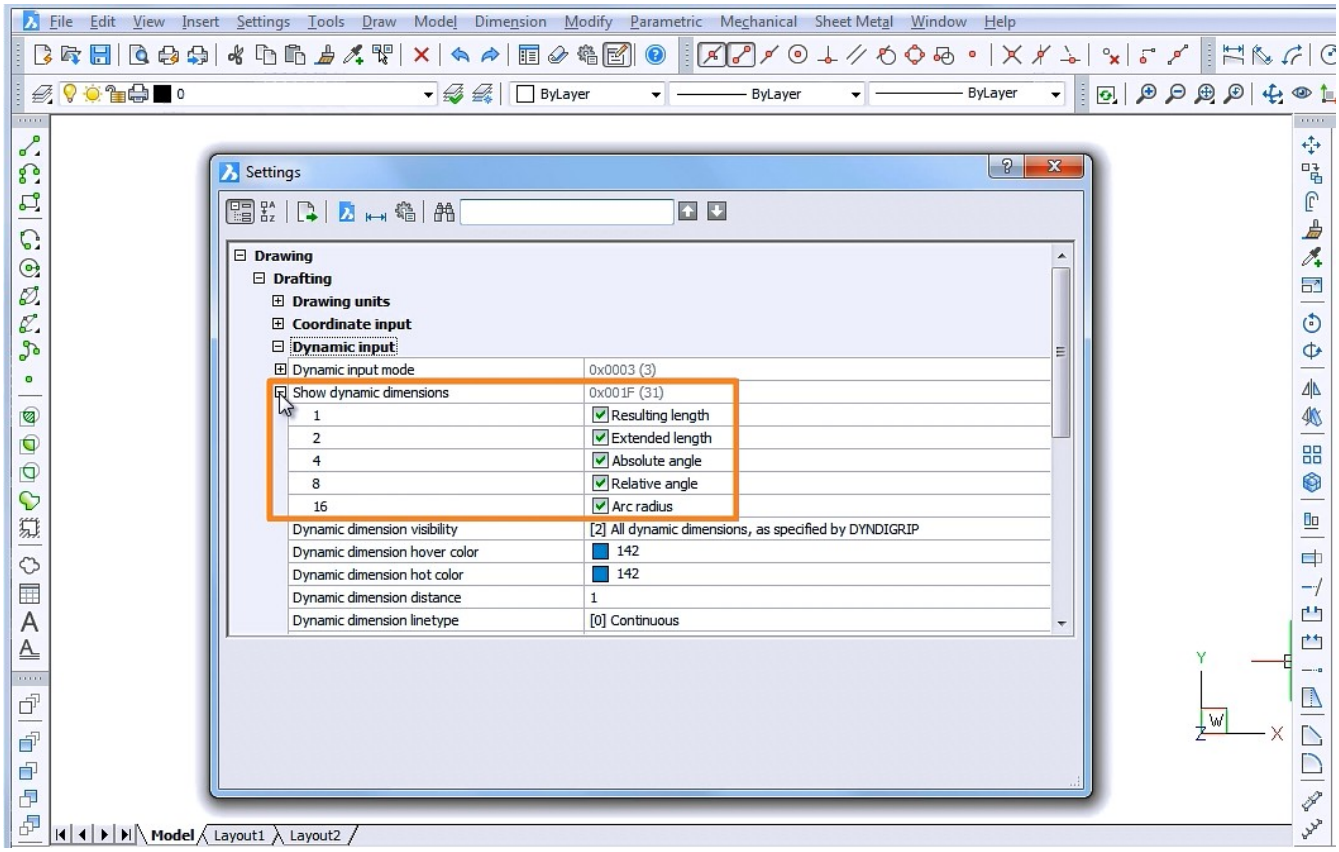
Youtube

Dynamic dimensions help you focus on the drawing area

Dynamic dimensions provide an interface at the cursor position to specify the length and angle of the linear entity being created or grip-edited.

System Variables A series of system variables and user preferences controls the behavior and appearance of dynamic dimensions. Right click the DYN field to open the Settings dialog on the Dynamic Input settings. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/SettingDynamicDimensionsBricsCAD.jpg})

Figure 1.10



Drawing Lines and Polylines Dynamic dimensions are especially useful when drawing lines or polylines. Hit the TAB key to go back and forth between the entry fields. Press Enter when the dimensions are correct.

Measuring Entities When you select an entity and pause the cursor over one of its grips, dynamic dimensions display. You can use this feature to measure entities in your drawing.

Grip Editing When grip editing a line, the following fields are available: total length, angle from the X-axis, length increment and relative angle. Back to Tutorial Overview

1.4.4 Tip on how to create blocks quickly on the fly

Youtube

Ever find yourself in the middle of a project with deadlines to meet, need to create a block, but hate how can disrupt your workflow? Or would like to collect some objects together so you can manipulate them easily? Then this video tip on how to create blocks quickly, on the fly is for you! Forms, don't you hate them? Wouldn't you rather just get on and get the drawing done? I've always found block creation really disruptive to my workflow. By the time I've mucked around and filled in the form to create the block, I've often forgotten what I needed it for in the first place.

> "There is a solution for real time, not a form in sight, block creation, and best of all it's built right into BricsCAD."

Back to Tutorial Overview

1.4.5 Dynamic Grips and Tips

https://www.youtube-nocookie.com/embed/2941QC44G-4

How to manipulate the special grips of dynamic blocks

As of BricsCAD V15, we can manipulate the special grips of dynamic blocks directly.Previously, this was possible only through the Properties palette. This video explains what the different grip icons mean, how they operate to change the block, and then provides a few tips on using dynamic blocks. Attached to this post is dyngrips.dwg, the drawing used in the video. Back to Tutorial Overview

1.4.6 Grips Editing in BricsCAD

https://www.youtube-nocookie.com/embed/RyMKZeWP_pE

Drag grips to stretch, move, mirror, rotate or scale entities.

Most people know about grip stretching, but there is much more. When you stretch a grip point and then repeatedly press the space bar, the selected entities are moved … rotated … scaled … or mirrored.

> **Each grips editing command has a Copy option which keeps the original entities.**

Back to Tutorial Overview

1.4.7 Use Quick Select

https://www.youtube-nocookie.com/embed/TqSIV9tKN4s

Use smart filters to compose a selection set

Quick Select allows you to specify filters, such as entity type, color, line type or any other entity property to compose a selection set. **Start Quick Select** In BricsCAD the Quick Select tool is integrated in the Properties panel. Click the Quick Select button to switch between Properties mode and Quick Select mode. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent2-2.jpg}) **Quick Select Modes** After applying the filter criteria you can choose to add the entities to the current selection set, remove the matching entities from the current selectionor create a new selection set. Optionally, Quick Select applies either to the entire drawing or to the current selection set. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageCon 1.jpg}) Back to Tutorial Overview

Figure 1.11

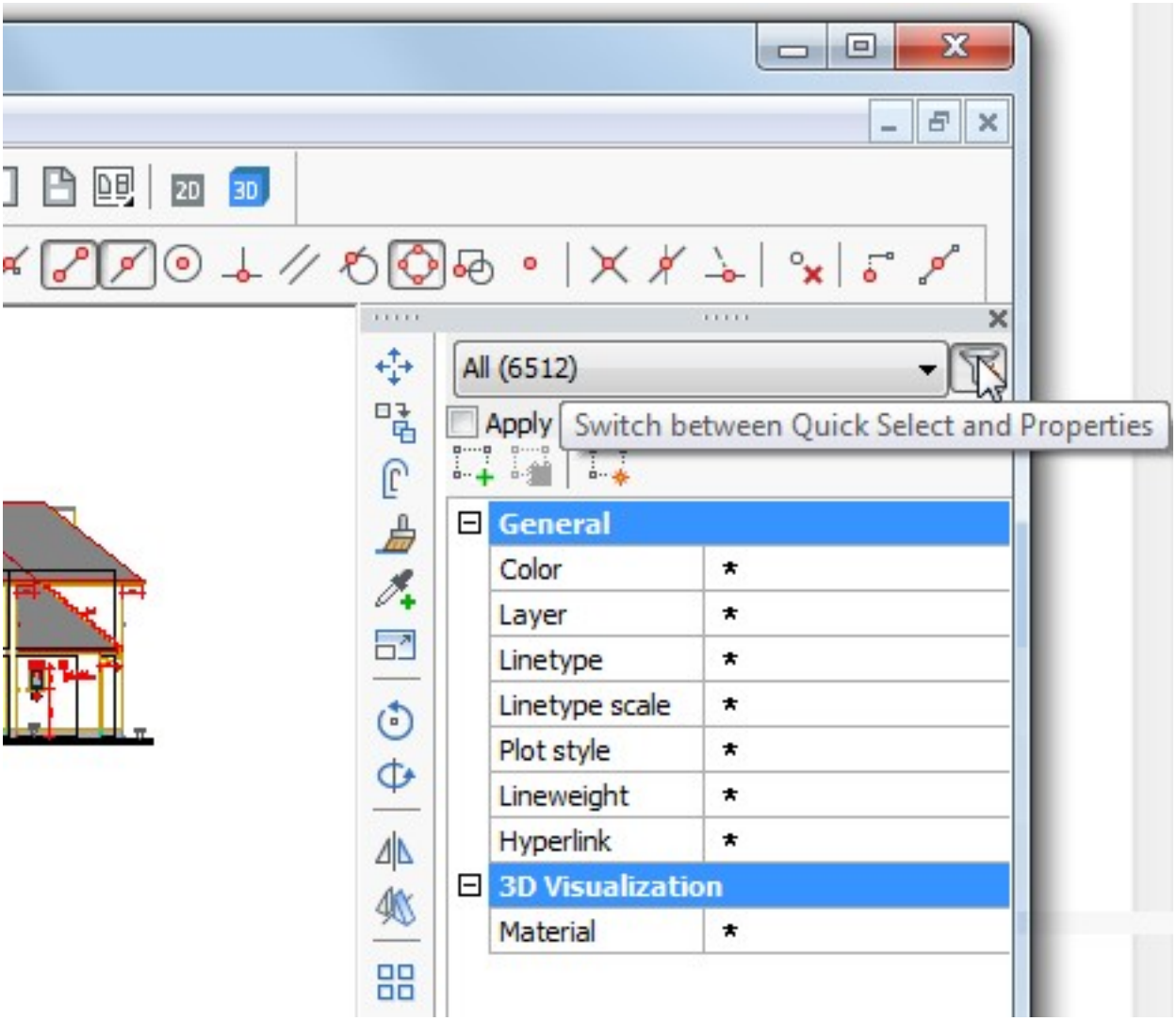
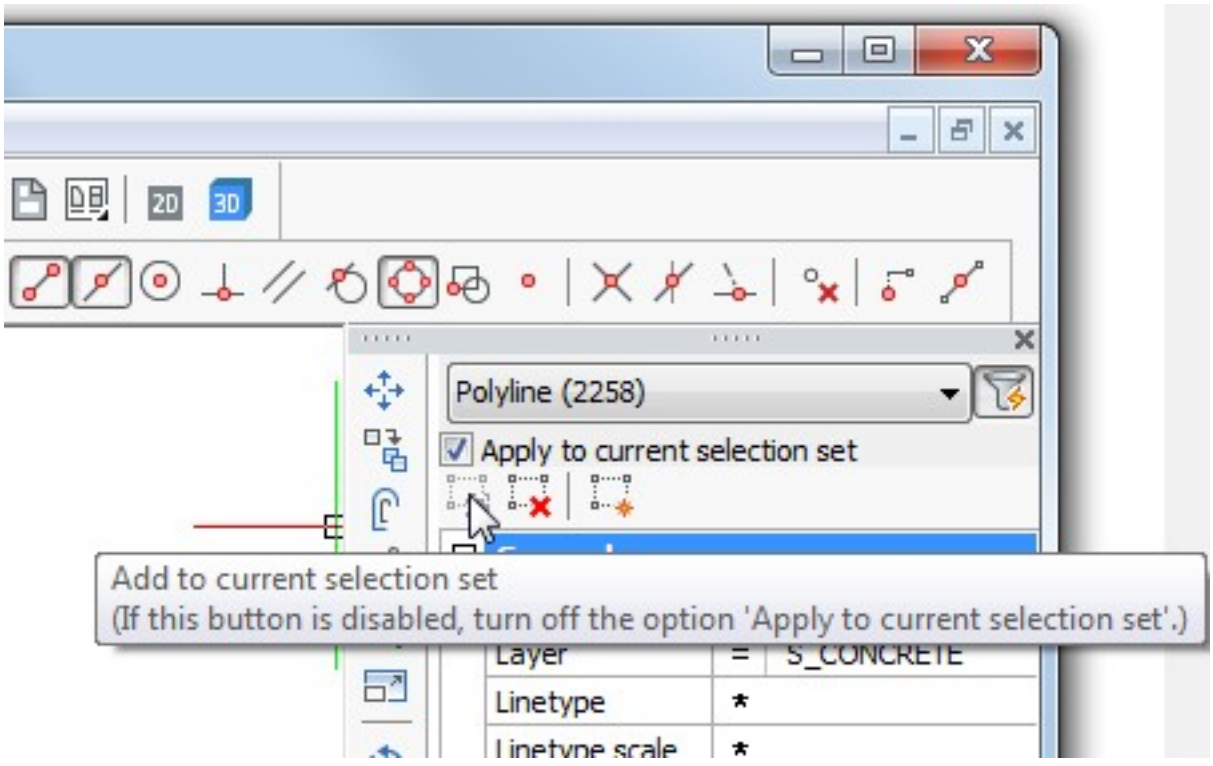


Figure 1.12



1.4.8 Secrets of the shift key in BricsCAD

https://www.youtube-nocookie.com/embed/8yyr8chvBIM

It does more than just capitalising text.

Everyday we use the shift key to write big letters or to select files, but in BricsCAD it does a lot more. Jan Bergmann will show you how you can use the shift key in BricCAD.

First secret: Deselect Objects SHIFT + Left Mouse Click Everyone knows how to select objects, but sometimes you've selected that extra object you didn't need. There's no need to select again. You can be very precise and deselectthe redundant objects while holding the shift key.

Second secret: Temporary Ortho mode or Free draw mode Hold SHIFTwhen drawing a line When you draw a line and youneed to switch between the Ortho mode or the Free mode, press andhold SHIFT and you'll see your mode changing temporarily.

Third secret: Switch between trim and extend Hold SHIFT when in Trim or Extend mode Trim what you want in the Trim mode. If you need to extend some lines, press and hold SHIFT and you switch to a temporary extend command. It also works the other way around.

Fourth secret: Snap menu sideways to the mouse Press SHIFT + Right Mouse Click If you have selected an object and want to move it. Press the SHIFT key + right click and the snap menu appears. Choose the action you want and itwill be executed with the next mouse click.

Fifth secret: Real Time Motion Press and hold Shift + Middle Mouse Button You can now move your drawing like a satellite around the earth. Back to Tutorial Overview

1.4.9 How to Align Entities

https://www.youtube-nocookie.com/embed/aEIL3uVzEkw

Forget move, scale and rotate. Use the align command.

Sometimes you get some drawings from third party companies which are wrong in location and wrong scale. With this command you can correct it very fast. Back to Tutorial Overview

1.4.10 Copying between drawings

https://www.youtube-nocookie.com/embed/Cr0ktFdYzEE

When copying entities from one drawing to another you can either paste the entities using a default base point, paste with a user-defined base point, paste to the original coordinates or paste the selection as a block.

Select the desired procedure in the Edit menu or use a keyboard shortcut. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom
content/uploads/2017/06/imageContent3-1.jpg}) Back to Tutorial Overview

Figure 1.13



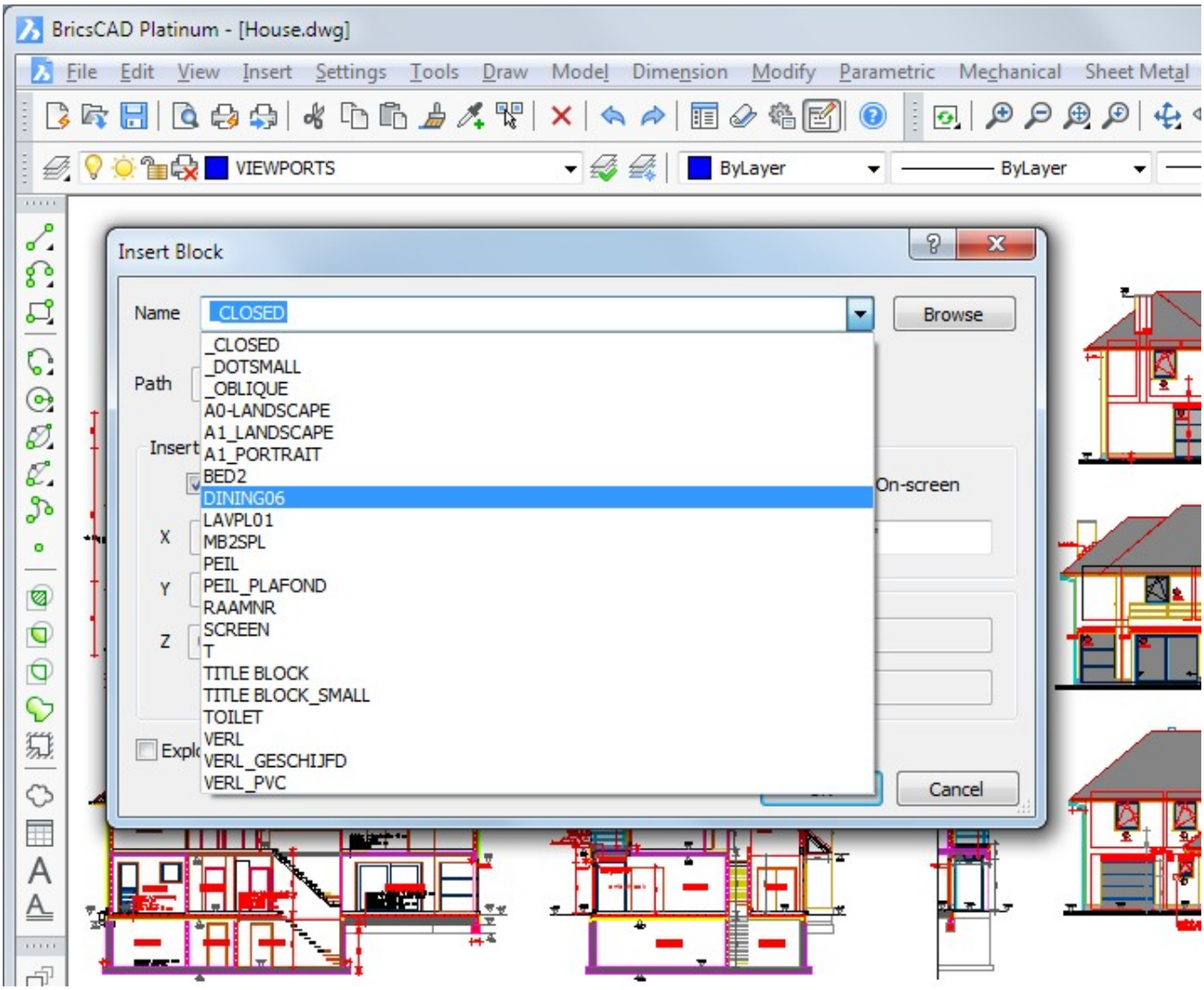
1.4.11 Manage blocks with the Drawing Explorer

https://www.youtube-nocookie.com/embed/ww0w4RnwIT4

In this movie you will learn how to insert blocks in a drawing.

In this movie you will learn how to insert blocks in a drawing. The source geometry or block definition either exists in the current drawing or in another drawing. You can also insert an entire drawing as a block in the current drawing. In BricsCAD you can insert blocks using the Insert command or using the Drawing Explorer. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent2-1-1.jpg}) Back to Tutorial Overview

Figure 1.14


BricsCAD Platinum - [House.dwg]
File Edit View Insert Settings Tools Draw Model Dimension Modify Parametric Mechanical Sheet Metal
VIEWPORTS
ByLayer
ByLayer
Insert Block
Name
CLOSED
Browse
Path
Insert
On-screen
X
Y
Z
Cancel
_CLOSED
_DOTSMALL
_OBLIQUE
A0-LANDSCAPE
A1_LANDSCAPE
A1_PORTRAIT
BED2
DINING06
LAVPL01
MB2SPL
PEIL
PEIL_PLAFOND
RAAMNR
SCREEN
T
TITLE BLOCK
TITLE BLOCK_SMALL
TOILET
VERL
VERL_GESCHIJFD
VERL_PVC

1.4.12 Package Your Drawings with eTransmit

https://www.youtube-nocookie.com/embed/-LPiaODCI_g

If you're working on drawings, there are going to be times when you need to send and receive drawing files. The bigger the project, the more important it becomes to do this as efficient as possible. In this tutorial you will learn how to use eTransmit. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent23.jpg}) Back to Tutorial Overview

Figure 1.15

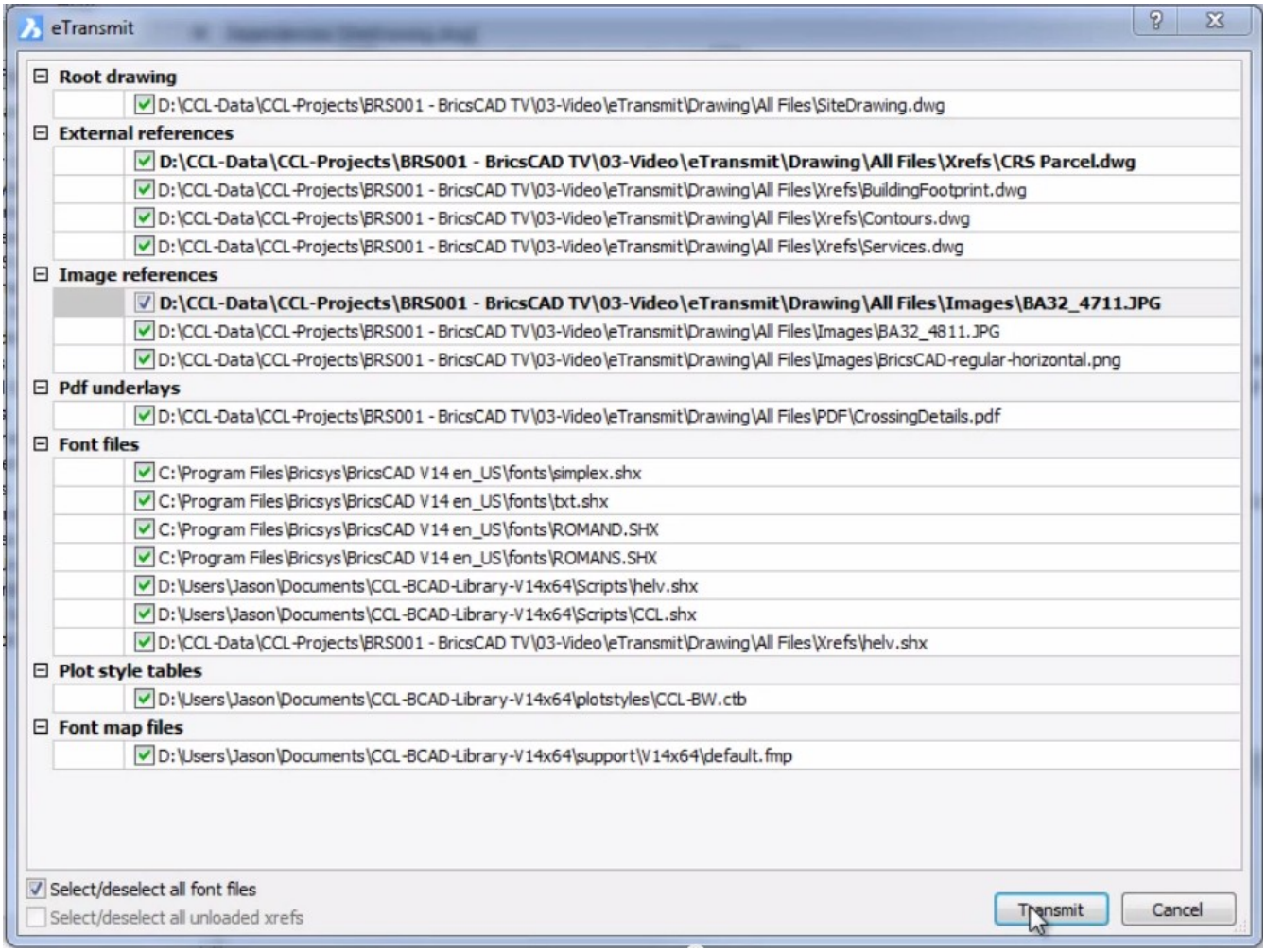

eTransmit
Root drawing
D:\CCL-Data\CCL-Projects\BRS001 - BricsCAD TV\03-Video\eTransmit\Drawing\All Files\SiteDrawing.dwg
External references
D:\CCL-Data\CCL-Projects\BRS001 - BricsCAD TV\03-Video\eTransmit\Drawing\All Files\Xrefs\CRS Parcel.dwg
D:\CCL-Data\CCL-Projects\BRS001 - BricsCAD TV\03-Video\eTransmit\Drawing\All Files\Xrefs\BuildingFootprint.dwg
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Image references
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Pdf underlays
D:\CCL-Data\CCL-Projects\BRS001 - BricsCAD TV\03-Video\eTransmit\Drawing\All Files\PDF\CrossingDetails.pdf
Font files
C:\Program Files\Bricsys\BricsCAD V14 en_US\fonts\simplex.shx
C:\Program Files\Bricsys\BricsCAD V14 en_US\fonts\txt.shx
C:\Program Files\Bricsys\BricsCAD V14 en_US\fonts\ROMAND.SHX
C:\Program Files\Bricsys\BricsCAD V14 en_US\fonts\ROMANS.SHX
D:\Users\Jason\Documents\CCL-BCAD-Library-V14x64\Scripts\helv.shx
D:\Users\Jason\Documents\CCL-BCAD-Library-V14x64\Scripts\CCL.shx
D:\CCL-Data\CCL-Projects\BRS001 - BricsCAD TV\03-Video\eTransmit\Drawing\All Files\Xrefs\helv.shx
Plot style tables
D:\Users\Jason\Documents\CCL-BCAD-Library-V14x64\plotstyles\CCL-BW.ctb
Font map files
D:\Users\Jason\Documents\CCL-BCAD-Library-V14x64\support\V14x64\default.fmp
Select/deselect all font files
Select/deselect all unloaded xrefs
Transmit
Cancel

1.4.13 Using the DimBreak Command in BricsCAD

https://www.youtube-nocookie.com/embed/W72PI-UEa7Q

DimBreak helps improve drawing clarity

If you draw in BricsCAD, youprobably need DimBreak. Have you noticed how quickly dimension and extension lines can get crossed up? It's a common situation as you add additional dimensions to your drawing sheets. Also, you'll find that some dimension annotations will lie on top of drawing geometry. When this happens, drawing clarity suffers – and that's the last thing that a designer wants! You need to ensure that the fabricators or builders who will use your drawings get the correct data from those sheets. This short video byRalph Grabowskiwill show you how to clean up these unwanted overlaps. Ralph uses theDimBreak command in BricsCADto break and heal dimension and extension lines that cross each other. Next, he'll show you how to clean up dimensions that obscure other entities. What happens when your drawing changes and a specific break is no longer needed? Well, the DimBreak command can also heal existing breaks one at a time, or across the entire drawing. Finally, Ralph shows you how to properly leverage the Auto option of the command. The dimensioning tools in BricsCAD are very powerful, and better yet, quite familiar. Do you want to learn more about dimensioning in BricsCAD, and DimBreak in particular? Click on the links below to access these specific on-line help pages atbricsys.com:

- The DimBreak command (6.11.7)
- BricsCAD dimensioning tools overview (1.21.2)
- Associative dimensions and related entity snaps (1.21.3)
- The Dimension Style Explorer (1.21.4)
- Dimension settings groups (1.21.5)
- Working with dimension styles (1.21.6)
- Linear dimension creation (1.21.8)
- Angular dimension creation (1.21.9)
- Diametrical and radial dimension creation (1.21.10)
- Ordinate dimension creation (1.21.12)
- Creating leaders and annotations (1.21.13)
- Editing dimensions (1.21.15)
- Working with geometric tolerances (1.21.16)

Back to Tutorial Overview

1.4.14 Making Smarter Drawings with Annotative Scales

https://www.youtube-nocookie.com/embed/BKsyXQ0X1hQ Annotation scaling tools can help you save you hours of time previously spent calculating scale factors and text heights. While creating, editing, and managing drawing data at different scales, stored in multiple layers. Back to Tutorial Overview

1.5 2D Drafting

1.5.1 Use the Quad to simplify working with Hatch

Youtube BricsCAD V15 introduced a number of enhancements to editing Hatch. Coupling this with the intuitive Quad and you will speed through Hatch operations. BricsCAD V15 comes with grip editing for hatch, boundary generation, and simple hatch editing using double click. In this video we will look at these enhancements, and to speed things up we will be using the Quad, which places Hatch options at your fingertips. Back to Tutorial Overview

1.5.2 Making entities transparent

https://www.youtube-nocookie.com/embed/sWriVvS8VVU BricsCAD can display entities opaque or semi-transparent. This is done with the transparency property to individual entities, to entire layers, and to blocks. A setting in the Hatch command makes hatch patterns and fills separately transparent. This video shows you how to use the transparency property with entities, layers, and hatches. It clearly explains the difference between the Transparency command and the “CeTransparency” variable. You also learn how to change the fading of locked layers, xrefs, and references being edited. Back to Tutorial Overview

1.5.3 Copying Layouts to Model Tab

Youtube

How to use the ExportLayout command

This video shows you how to use the ExportLayout command to export a layout to .dwg file, and then how BricsCAD opens the .dwg file in model space. Sometimes it is necessary to bring layouts into the model tab. For example, some CAM programs cannot read layouts. Or you might want to save a layout and all its viewports as an independent drawing file. The ExportLayout command lets you so these things. Back to Tutorial Overview

1.5.4 Creating and Editing Associative Arrays

https://www.youtube-nocookie.com/embed/aBVlovvZAvw

How to use the ArrayPolar, ArrayPath and ArrayEdit Commands

The old Array command is a powerful way to make many copies in rectangular or round patterns. But the result is a group of independent objects, and so the arrays cannot be edited except by erasing and starting over. To the rescue are associative arrays, in which the elements of the array can be interactively adjusted during their creation, as well as after the arrays are placed in the drawing.

Interactively adjust elements of the array during their creation This video shows you how to use the ArrayPolar command to interactively create and modify polar arrays; the ArrayPath command to place objects along a path, such as a spline; and the ArrayEdit command to change the array after the fact — including replacing individual items with another object. Back to Tutorial Overview

1.5.5 Transform a smartphone into a tablet

https://www.youtube-nocookie.com/embed/o05XkMG-1FQ

Using 2D Dimensional constraints in BricsCAD

Learn how to work with 2D constraints in BricsCAD by transforming a smartphone in a tablet. BricsCAD's parametric modeling allows you to change your model easily and intelligently.To keep your design intent, you need to add some geometric constraints. The overall height of our smartphone is equal to the sum of the height of its screen and the heights of both borders (upper and lower), which are equal. We can easily specify this condition with an expression in the constraints bar. Back to Tutorial Overview

1.6 3D Modeling

1.6.1 Making 3D surfaces

https://www.youtube-nocookie.com/embed/zmJd9V_HJ6M Just as BricsCAD makes 3D solid models from 2D entities, it also can make 3D surfaces from them. After completing this lesson, you will be able to:

- Understand how to create surface models using Extrude, Revolve, and Loft.
- Create your own 3D surfaces from 2D entities.
- Define the path of a loft by using guides.

1.6.2 Modify 3D surfaces

https://www.youtube-nocookie.com/embed/x5Pw8POSfn8 Once you've made 3D surfaces, BricsCAD lets you manipulate them with commands like XFaces (makes copies of individual faces), sweep profiles along paths, thicken surfaces into solids, twisting surfaces, and change the continuity of twisted surfaces. After completing this lesson, you will be able to:

- Understand the limitations in editing 3D surfaces
- Modify 3D surfaces with the Twist command
- Extract faces with the XFaces command
- Make new surfaces with the Sweep command

1.6.3 Extrude and revolve

https://www.youtube-nocookie.com/embed/B4hTPwJAzCo Extruding and Revolving 2D Entities. A third way to create a 3D model in BricsCAD is to draw entities in 2D (called a ”sketch”) and then extrude or rotate them into 3D bodies. After completing this lesson, you will be able to:

- Understand how to use Extrude and Revolve.
- Create your own 3D models from 2D entities.
- Know the difference in extruding and revolving open and closed entities.

1.6.4 Slice, section and interfere

https://www.youtube-nocookie.com/embed/22CEWcG57TY Slicing Up 3D Models. Slicing, Sectioning and Finding Interferences. After completing this video, you will be able to:

- Understand how to use Slice, Section, and Interfere.
- Slice, section and find interferences between 3D models
- Know the difference between slices, sections and interferences.

1.6.5 Slicing up 3D models

https://www.youtube-nocookie.com/embed/22CEWcG57TY **Slicing, Sectioning and Finding Interferences** After completing this video, you will be able to:

- Understand how to use Slice, Section, and Interfere.
- Slice, section and find interferences between 3D models
- Know the difference between slices, sections and interferences.

1.6.6 Loft 3D Surfaces

https://www.youtube-nocookie.com/embed/AV8wukyZ9LY The loft command creates surfaces along two or more cross-sections (open entities, such as splines) that determine the look across the length of the 3D loft entity. For additional control, you can optionally specify guidelines, which control how the surface looks along the length of the loft. After completing this lesson, you will be able to:

- Understand how to use Loft
- Create your own lofted 3D bodies from 2D and 3D entities
- Know the difference between cross-sections and guidelines

1.6.7 Thicken 3D and 3D entities

https://www.youtube-nocookie.com/embed/KjbRTkfvXj8 The Thicken command is useful for turning 2D entities into 3D bodies. As well, it copies 3D faces, thickening them into independent 3D bodies. After completing this lesson, you will be able to:

- Understand when to use Thicken
- Create new shapes by thickening existing ones
- Know the difference between thickening 2D entities and 3D faces

1.6.8 3D Manipulator

https://www.youtube-nocookie.com/embed/HfJVjIZJeT8 BricsCAD provides an on-screen widget called the Manipulator for moving, copying, rotating, resizing (scale), and mirroring entities and bodies in 3D. After completing this video, you will be able to:

- Use manipulator widget in BricsCAD.
- Activate the manipulator several ways.
- Know the actions the manipulator can do.

1.6.9 Twist 3D solids

https://www.youtube-nocookie.com/embed/DQgmdipAat0 The dmTwist command is one of the trickier ones to understand, and so this video explicitly shows you the purpose of its many options. It shows you how to do full twists, partial twists, on-axis, and off-axis twists of 3D solids. After completing this lesson, you will be able to:

- Use the dmTwist command
- Understand the meaning of its options
- Know the different results, depending on the axis

1.6.10 Deform faces

https://www.youtube-nocookie.com/embed/0xvrPDI1j8E Faces normally are flat or made of simple curves, but BricsCAD can deform them into complex shapes. The dmDeformPoint stretches a face between edges and a point in space. The dmDeformCurve command is similar, but deforms a face to stretch it between its edges and an entity in space, like a circle. After completing this lesson, you will be able to:

- Understand what it means to deform faces
- Use the dmDeformPoint and dmDeformCurve commands
- Create new kinds of 3D models as deformed ones

1.6.11 Sweep profiles along paths

https://www.youtube-nocookie.com/embed/qUuByFK0IU4 The Sweep command takes a 2D profile (a closed entity) and then extrudes it along a path (usually an open entity). The result is a 3D body that resembles a handrail or similar. After completing this lesson, you will be able to:

- Create your own 3D swept models
- Understand the difference between profiles and paths
- Know the shortcomings of the Sweep command

1.6.12 Turning 2D Entities in Simple 3D

https://www.youtube-nocookie.com/embed/U8PxsCGA_fA Turning 2D Entities in Simple 3D. The most basic way to create 3D entities in BricsCAD is to add a thickness to entities and a Z coordinate to endpoints. After completing this, you will be able to:

- Use thickness to turn some 2D entities into 3D ones.
- Change the Z elevation of some entities.
- Know which entities work with Z and thickness.

1.6.13 Section planes

https://www.youtube-nocookie.com/embed/14SglJiQtbw To look inside 3D models, like houses or gearboxes, BricsCAD provides the SectionPlane command. Once in the drawing, you can manipulate the section plane to see other parts of the model. The effect of section planes can be saved (as blocks). After completing this lesson, you will be able to:

- Understand how to use section planes and change their look
- Create your own 3D and 2D sections, and save them as blocks
- Know the difference a section entity and a section plane

1.6.14 Flatten and extract faces and edges

https://www.youtube-nocookie.com/embed/opGotp0H3RY 3D models don't have to stay 3D; BricsCAD can flatten them with a variety of commands, each with their pros and cons. The Flatten command flattens the entire 3D model to a series of 2D lines and polylines that look "3D"; the FlatShot command does similarly, but makes a copy and then lets you specify the look of hidden lines (such as dashed, colored). By contrast, the XEdges command makes copies of individual edges on 3D models as 3D splines and circles; while XFaces command extracts faces as 3D surfaces. After completing this lesson, you will be able to:

- Understand how to flatten 3D models
- Extract edges and surfaces from 3D models
- Know how to use Flatten and FlatShot, and XEdges and XFaces

1.6.15 Draw 2D Entities in 3D Space

https://www.youtube-nocookie.com/embed/rwHA_-DCE2k Drawing in 3D is 6x harder than 2D, because models have six sides to them and some parts are inevitably hidden from view. This video shows tips and tricks in seeing more than one side at a time, and you learn the names of standard engineering views. You learn the importance of normals, and how to easily draw on the sides of 3D faces, as well as rotate entities in 3D space. After completing this lesson, you will be able to:

- Understand how to use x,y,z coordinates
- Draw 2D entities on 3D models
- Use normals and commands like 3DRotate to change the orientation of 2D entities in 3D space

1.6.16 Naming 3D parts

https://www.youtube-nocookie.com/embed/g7KzMCwak_4 Naming the Parts of 3D Models Three-D modeling has its own jargon, which can be confusing to neophytes, and so this video illustrates the bit and parts of 3D models. After completing this lesson, you will be able to:

- Talk about the parts of 3D models accurately
- Identify features like the vertex, edge, and face
- Know the difference between solids, surfaces, and meshes

1.6.17 Solids, surfaces and Meshes

https://www.youtube-nocookie.com/embed/jUEt6Mp7opw Pros and Cons of Using Solids, Surfaces, or Meshes. BricsCAD provides three different kinds of 3D models -- solids, surfaces, and meshes -- and lets you convert from one to another. After completing this lesson, you will be able to:

- Know the difference between solids, surfaces, and meshes
- Understand how well each can be edited
- Convert between solids, surfaces, and polyface meshes

1.6.18 3D fillet and chamfer

https://www.youtube-nocookie.com/embed/9TOYoqlgr4o Making Edges Curved and Slanted. After completing this video, you will be able to:

- Understand how to use Fillet and Chamfer.
- Modify 3D models by curving and slanting edges.
- Know how to fillet and chamfer interactively.

1.6.19 Boolean operations

https://www.youtube-nocookie.com/embed/NXiCmOof0kY Joining, Separating, and Removing 3D Solids. Once you have several solid primitives in a BricsCAD drawing, you want to combine them, cut them, and drill into them. After completing this video, you will be able to:

- Understand how to use Boolean operations.
- Modify two or more 3D models.
- Know when and how to use Union, Intersect, and Subtract.

1.6.20 3D Primitives

https://www.youtube-nocookie.com/embed/pJZwzipiWK0 Modeling Basic 3D Bodies. Another way to create a 3D model is by assembling basic 3D entities that BricsCAD calls ”primitives.” After completing this video, you will be able to:

- Understand what 3D primitives are
- Create 3D primitives
- Know how to draw with Box, Pyramid, Wedge, Sphere, Cylinder, Cone, Torus. and PolySolid.

1.6.21 Solid edit: part 1

https://www.youtube-nocookie.com/embed/zjoMlZJaqKo The many faces of Solid Edit. After completing this video, you will be able to:

- Understand how to use the SolidEdit command.
- Copy and color faces of 3D models.
- Shell a 3D model to make it hollow.

1.6.22 Solid edit: part 2

https://www.youtube-nocookie.com/embed/HXf4Px-fyKw Copy, Color Edges, Move, Extrude Faces. After completing this video, you will be able to:

- Understand how to copy and color edges.
- Use the SolidEdit options Move and Extrude on faces.
- Know the difference between extruding and moving faces.

1.6.23 Static UCS, Set 2D Drawing Planes in 3D

https://www.youtube-nocookie.com/embed/4hGRvMM9FYo While dynamic UCS is useful in BricsCAD for modeling in 3D on existing faces, sometimes a static UCS (user-defined x,y-plane) is more convenient. After completing this video, you will be able to:

- Understand how to use UCSs in BricsCAD.
- Orient your own coordinate systems.
- Know how to reset the UCS.

1.6.24 Dynamic UCS, Drawing on any 3D Face

https://www.youtube-nocookie.com/embed/WpnLG91wD94 Almost all drawing in 3D is done on a 2D plane (a.k.a. an x,y-plane), and so to make 3D modeling easier BricsCAD provides you with the dynamic UCS. After completing this video, you will be able to:

- Understand what dynamic UCS is
- Draw 2D and 3D entities on any 3D face
- Know how lock a face to draw away from it

1.6.25 Viewing in 3D

https://www.youtube-nocookie.com/embed/I5-Y2a9XjFM Viewing 3D models is trickier than 2D plans, because of the sides and backs that you normally do not see in 3D, and so BricsCAD provides quick ways to change your view of the model. After completing this video, you will be able to:

- Zoom, pan, and orbit 3D models in BricsCAD.
- Use the LookFrom widget to change views.
- Create multiple views of a 3D model at the same time.
- Save specific viewpoints by name for reuse.

1.6.26 3D Direct Modeling Basic Commands

https://www.youtube-nocookie.com/embed/FI7Q4XweiSw

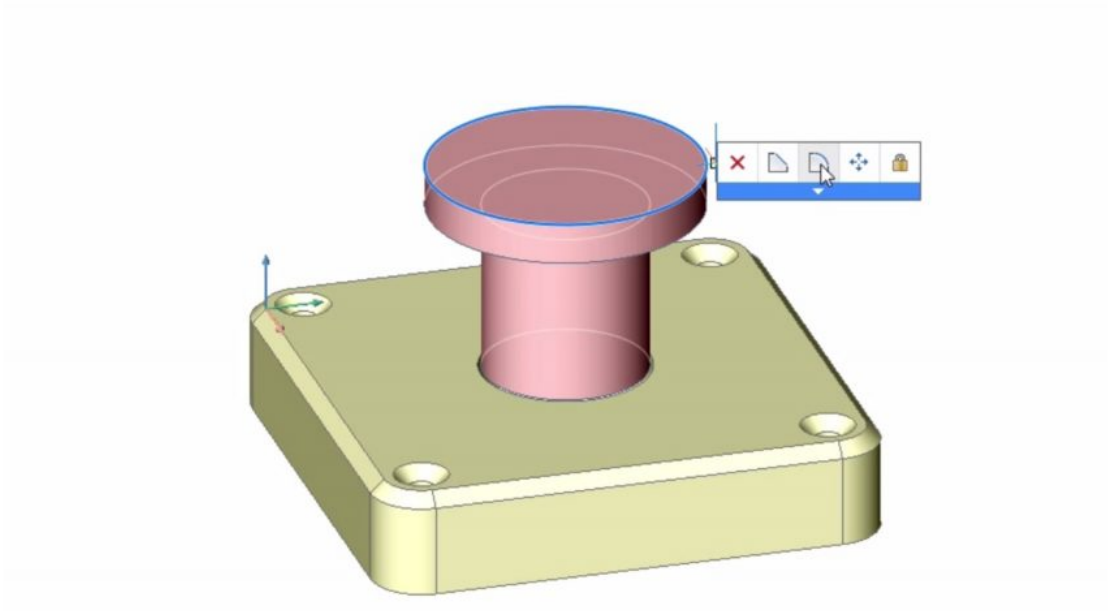
This tutorial shows how you can create complex models using powerful direct modeling technology that combines the best of two worlds: parametric and direct modeling.

Commands like chamfer, fillet, push/pull and extrude are all just one click away with the newly designed quad cursor.All designed bodies can be save as separate components. Bricscad also comes with a library which contains more than 30 thousands standard parts.

Using the context-sensitive quad cursor, your mouse movements are significantly reduced.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009695733/imageContent3-2-1024x572.jpg
Back to Tutorial Overview

Figure 1.16



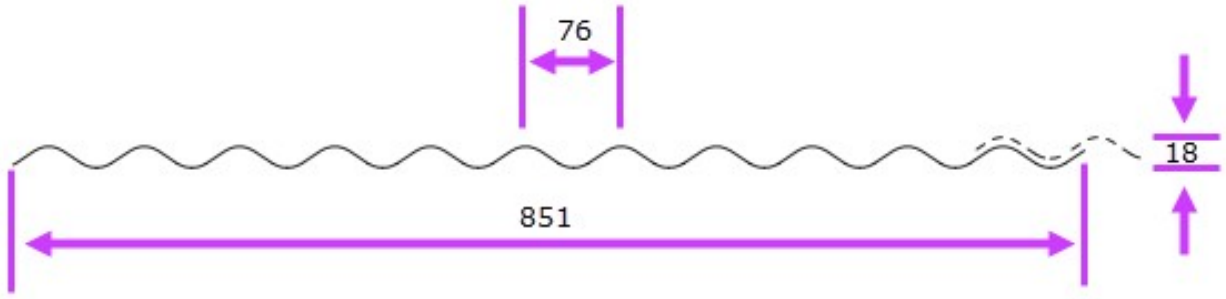
1.6.27 Extending onto the curve

https://www.youtube-nocookie.com/embed/kg9e3z7P5NY With BricsCAD V16.2, dynamic UCS (DUCS) has been enhanced to allow you to work with curved surfaces.In this example we use this extension to DUCS to quickly populate the connections on a pressure vessel. Back to Tutorial Overview

1.6.28 Making Beautiful Shapes from Lofts

Youtube The Loft command is fascinating for the fantastical shapes it creates. It takes two or more 2D objects (like circles or ellipses at different elevations) or the faces of 3D solids, and then creates a solid that smoothly transitions between them. This video shows you how to use the Loft command between three ellipses, and then between two 3D solids. It shows the surprising effects you can achieve with lofts when you change the settings between ruled, smooth, normal, and draft-angle values. Back to Tutorial Overview

Figure 1.17



1.6.29 Project Geometry – Corrugate Iron

https://www.youtube-nocookie.com/embed/ujiS_02O9VU BricsCAD V16.2 introduces Project Geometry. This command projects 2D geometry onto a region, a surface or a 3D solid; creating linework that you can use in the development of your model.In this tutorial we are going to use PROJECTGEOMETRY to put a “twist” on an method given on the Bricsys Forum to create a Corrugated Iron profile from a HELIX. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/ar

Back to Tutorial Overview

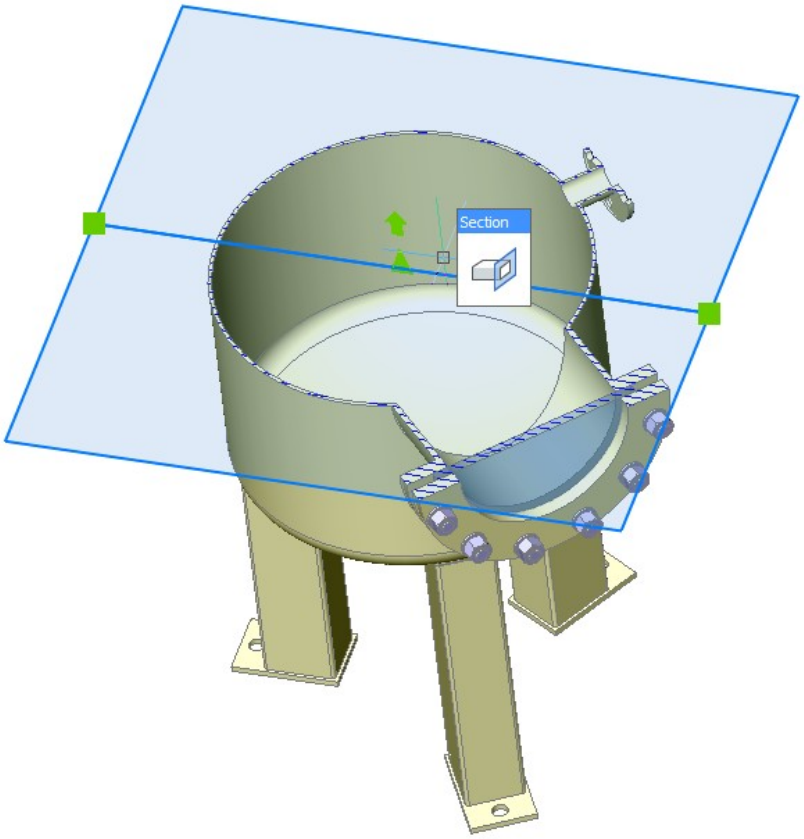
1.6.30 Project Geometry – Vessel

https://www.youtube-nocookie.com/embed/YHsk2B266kI

Use PROJECTGEOMETRY to markout your models

BricsCAD V16.2 introduces Project Geometry. This command projects 2D geometry onto a region, a surface or a 3D solid; creating linework that you can use in the development of your model.In this tutorial we are going to use PROJECTGEOMETRY to mark out potential nozzle positions on a pressure vessel. Back to Tutorial Overview

Figure 1.18



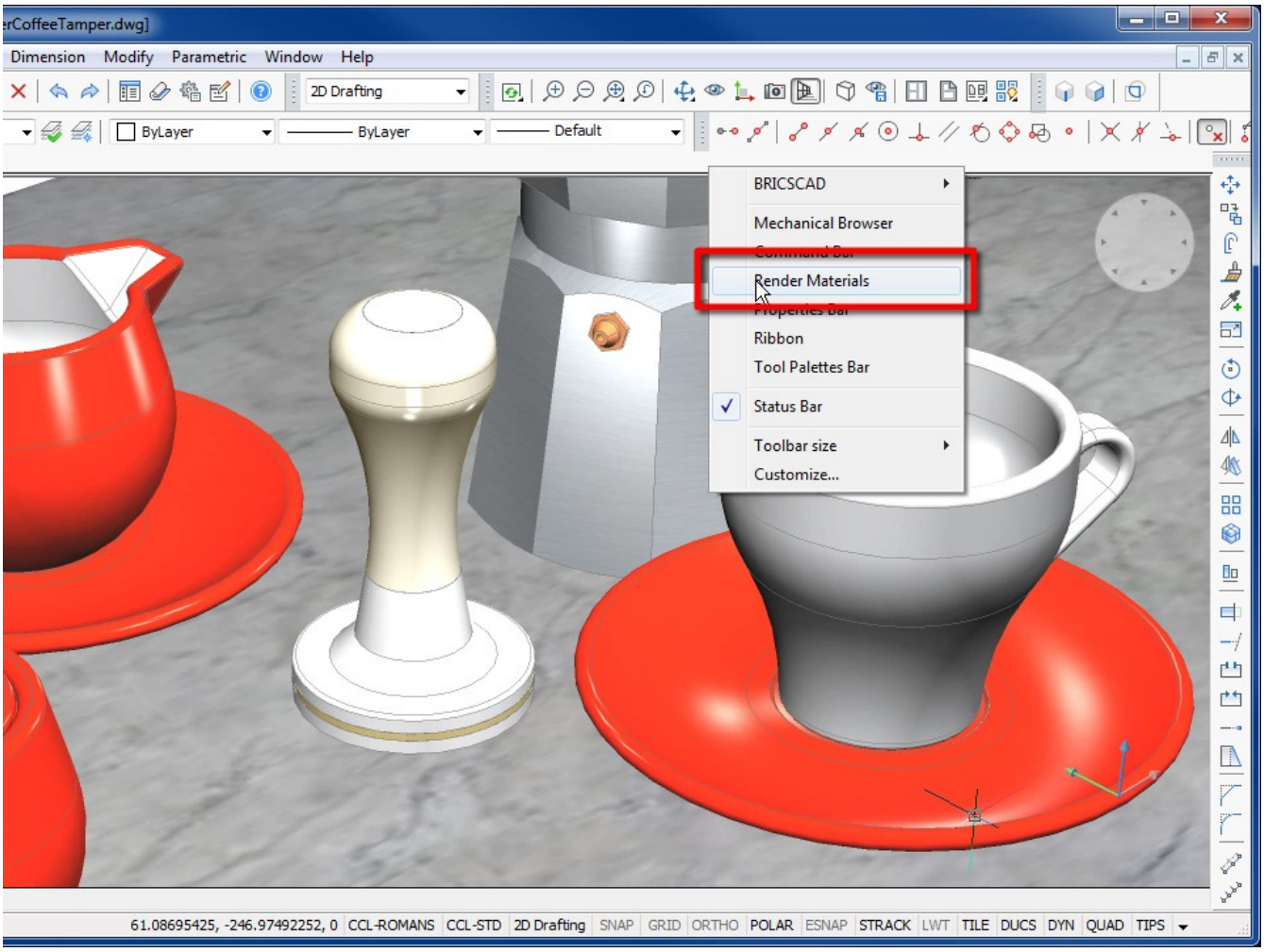
1.6.31 3D Modeling at a Clip

https://www.youtube-nocookie.com/embed/BX5xVd4jFWE BricsCAD 16.2 includes an exciting new feature called Clip Display (CLIPDISPLAY (6.9.24)). Quite simply it's going to change how you work in 3D. If you're familiar with Live sections, then Clip display is similar, but there are some important differences.Firstly you can work with multiple SECTIONPLANE (6.27.12)s, or BIMSECTION (6.7.27)s at the same time. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2016/12/imageContent2-153950.jpg})

Continue to use all modeling and editing tools Clip display also only CLIPS the DISPLAY of the model, where a Live section temporarily replaces the model with cut solids. This means you are still working directly with your model, and can continue to use all modeling and editing tools, including object snaps. What's more Clip display will also allow you to snap to points created by the cutting plane itself, providing you with an additional set of options.

Clipped sections are updated in real time Finally Clipped sections are updated in real time as you move them, allowing you to quickly change focus around your model. Back to Tutorial Overview

Figure 1.19



1.6.32 Bring life to your BricsCAD models with Render Materials

Youtube BricsCAD offersaccess to a huge render material library to bring your models to life. In fact the library has over 600 materials that you can choose from. In this video we'll work with render materials to demonstrate how you can add realism to your BricsCAD models. Opening render materials is simple. Just right click over any toolbar then select it from the menu. Alternatively you can type MATBROWSEROPEN (6.21.6) in at the command line. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009638214/imageContent3-1-1.jpg}) **Simply drag and drop the material from the browser** Applying materials is easy too. Simply drag and drop the material from the browser onto the object that you want to apply it to. This allows you to quickly try out a number of different materials. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009638234/imageContent5-1.jpg}) (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009638254/imageContent6.jpg}) Back to Tutorial Overview

Figure 1.20

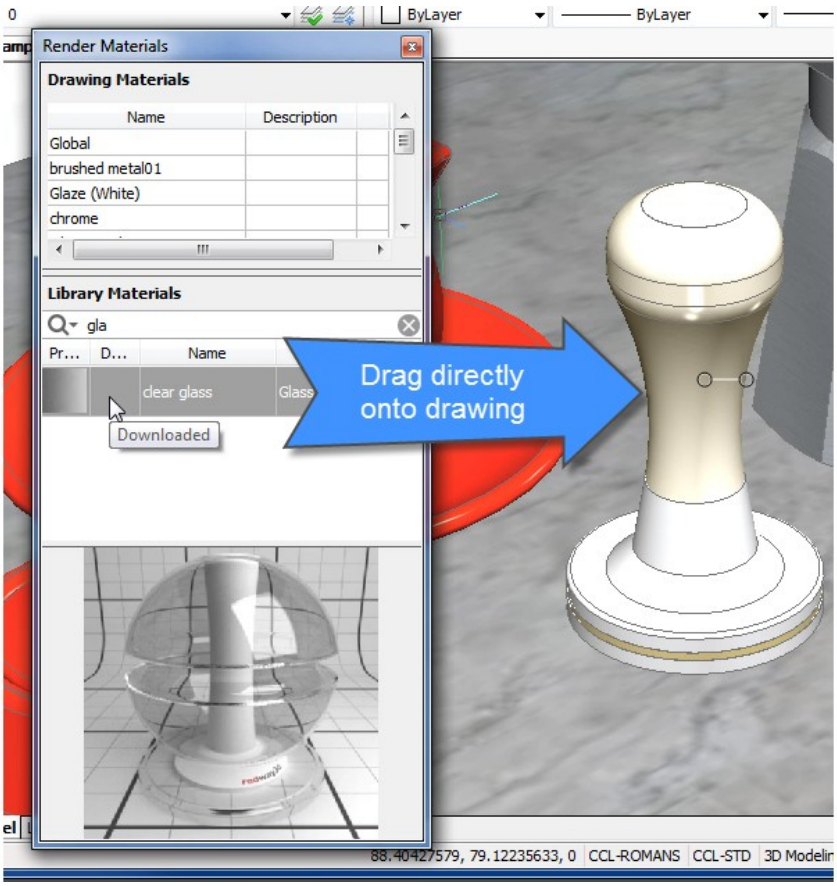
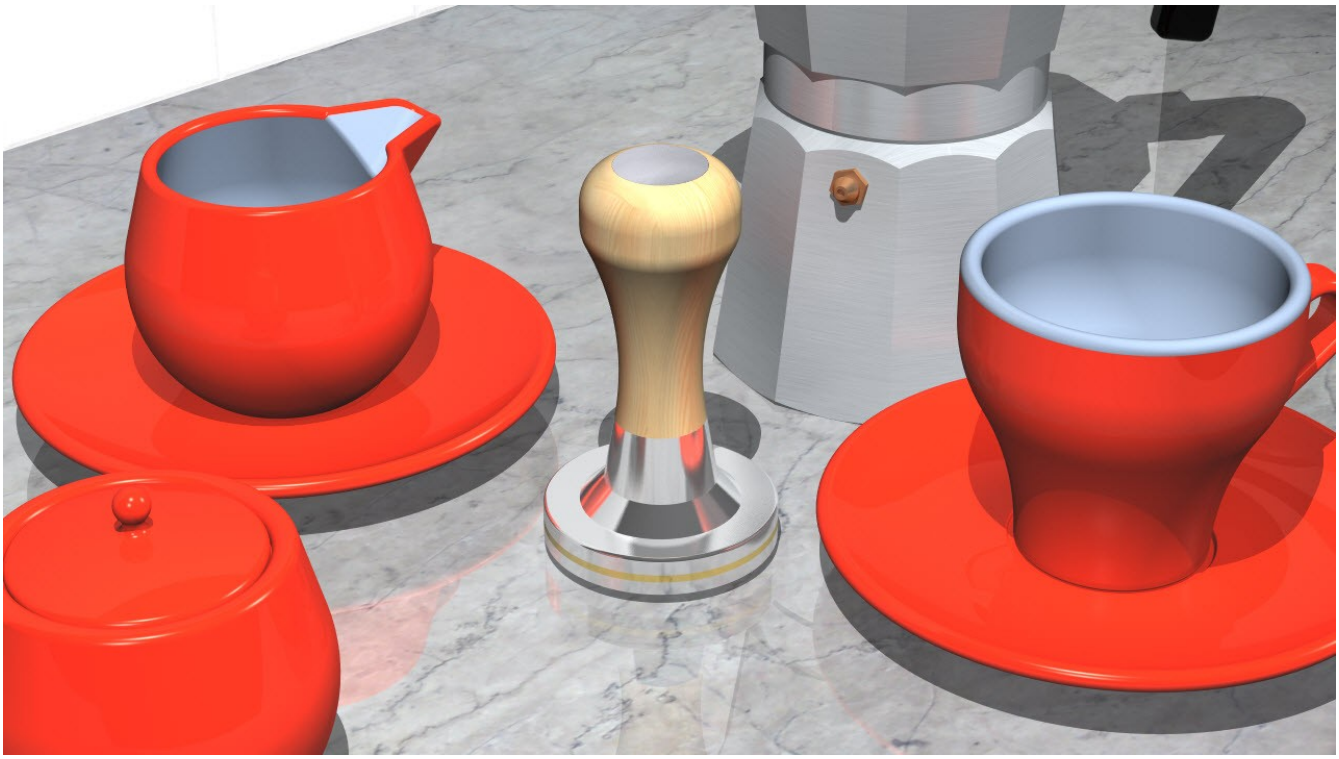


Figure 1.21



1.6.33 Generate drawing views in BricsCAD

https://www.youtube-nocookie.com/embed/COkW63wpheM

Drawing views from any 3D model

In BricsCAD, you can generate drawing views from any 3D model. These views are updated automatically when the original model is changed.

Base View Use the "**base views**" command to generate 2D drawing views from your 3D model. Ever click isa new and different view placed on your sheet.You can change the properties of one or multiple views in the properties Bar. For example, disable hidden and tangent lines.

Drawing View The type of Section View can be changed in the drop-down menu in the "**drawing views**" section, which can be found in the Ribbon. Similarly, you can also create a half section from the bottom base view. You can see that the hatch angle is identical for different projections of the same solid. It changes according to the view section style of the solids. You can adjust the depth of the section by creating a section of the connecting flange of the valve and after placing thedrawing view.

Detail view Throughthe "**detail view**" feature to zoom in on certain details of your drawing views. You can choose out of two option: create circular or rectangle detail views.If needed, you can also adjust the scale of your detail section.

Custom styles BricsCAD has a standard section style, but if that doesn't suit your needs, you can add new custom styles in the "**view section styles**" panel of the drawing explorer.

Customize your style Gothe "**identifier**", where you can change the color of the identifier, its position and more. The preview images allows you to see all changes before they are applied. You can change color of the hatch, its scale, type and angles in the"**hatch**". Finally, in the "**cutting plane**", the linetype of the section lines, their color and lineweight can be changed.For the end and bend lines, you can change the position, color, lineweight and linetype. Similarly, you can create a custom View Detail Style and adjust all properties.To change the lineweight of the hatch, go to the drawing explorer and select "layers".You can adjust the current section view style to your own custom one in the **properties bar**.The detail section view styles can be changed in a similar way.

Finish your drawing To specify the necessary dimensions in your drawing, you can use the linear dimension tool.As finishing touch, you can add the Bill of Materials, which gathers data about all the materials used in your model. Back to Tutorial Overview

1.6.34 How to setup the 3DConnexion 3D mouse in BricsCAD

Youtube BricsCAD V14 now supports 3Dconnexion 3D mice. Users can simultaneously pan, zoom and rotate 2D/3D content while the standard mouse is free to select menu items of parts of the model. Back to Tutorial Overview

1.6.35 Use surface modeling and deformable modeling

Youtube This video shows how the deformable modeling and surface modeling tools can be used to edit imported geometry. The imported camera body is going to be edited with the Direct modeling tools. First of all remove the redundant hole. Then make a new hole to mount the new camera lens using the Extrude tool.To remove the camera flash, select the faces that need be removed and apply the Delete tool. BricsCAD intelligently removes the set of faces and restores the underlying shape. Secondly, use the Repair tool to check the model on geometric inconsistency. Switch to the Report tab in Mechanical Browser (3.2.2) to see all the issues that have been found. Fix all issues of the model. Then use the deform Point tool to edit the surface. This tool applies force to the specified base point in a specific direction. The base point, direction and target deformation position can be specified. Next, use the Extrude tool with the Subtract option. Thenreduce the camera body using Move Point. With the Dynamic UCS switched off, Move Point will align deform direction with z axis of the current UCS.To create the mount for the camera flash, use the Extrude tool. Thirdly, use the Thicken tool to thicken the surface and insert the lens. Then you usethe Simplify tool to replace the geometry of the faces and edges under the given entity by analytic surfaces and curves Finally, align the lens using a concentric constraint. Back to Tutorial Overview

1.7 Images

1.7.1 Backgrounds for visualizing your 3D models

https://www.youtube-nocookie.com/embed/kDv8uWQbA6U

Backgrounds make your 3D visualizations better

A huge benefit of modeling in 3D is using your model for sharing and visualization. You can use backgrounds to enhance the visual quality of your 3D models in BricsCAD. This video byRalph Grabowskishows you how to use the Background command to specify colors, gradients, and image parameters. First, set your Visual Style to “Model”. The Background command needs avisual style set to something other than“2D wireframe” mode. Next, simply start the BACKGROUND command. Go to the top of the dialog, and you’ll see your type choices – solid colors, gradients or images. Solids are pretty self-explanatory. Gradients transition from one color to another – for example, from blue sky to green grass. You have the ability to use a two-color or three-color gradient in BricsCAD. Using images as backgrounds gives you the ability to “place” your model in a scene. Select your image file, and you can move and scale it to fit your graphics window. Then, you can zoom, pan and adjust the viewing angle to better fit the model into the image. You have additional options that are available when you use an image as a background: stretch, center or tile the image to fit on the screen. Next, you can lock the aspect ratio of the image to keep it from distorting. Finally, the sliders in the Background dialog give you the choice of adjusting the background image’s scale or offset.

Want more info?

If you want to learn more about 3D modeling, rendering and using backgrounds for visualization, just click on the links below. They’ll take you to the related page in the BricsCAD on-line help system:

- The BACKGROUND command (6.6.1)
- MatchPerspective (6.21.7)–match the current MSPACE perspective view with a background image
- VisualStyles (6.31.13)–create and control visual style definitions in Drawing Explorer
- Materials (6.21.11)–create materials and edit their properties through the Drawing Explorer
- MaterialMap (6.21.10)–control texture mapping in BricsCAD Pro and Platinum

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1.7.2 Image Geolocator

https://www.youtube-nocookie.com/embed/HlrQCXgNGUE

Placing Geocoded Images accurately is easy with BricsCAD

Utilizing aerial imagery with your drawings has a numerous benefits, and has become a common expectation in the production of good project documentation. But how do you go about aligning these types of images with your vector data? The good news is that BricsCAD has long been able to attach geocoded images in the correct location on your drawing. This tutorial shows you how painless this is to do with BricsCAD. Back to Tutorial Overview

1.8 Printing & Plotting

1.8.1 Output As Pdf

Youtube

How to save single drawings or groups of drawings in PDF format.

PDF is the most popular way to view drawings, because they can be viewed on just about any computer or mobile device today. PDF allows viewing in a way that can't be edited. BricsCAD provides two ways to convert drawings to PDF: 1) One drawing at a time through the Export command 2) Groups of drawings through the Publish command. This video shows you both approaches. Back to Tutorial Overview

1.8.2 Paper Space Layouts in BricsCAD

https://www.youtube-nocookie.com/embed/tIwY8xao-Q0

Creating Paper Space Layouts in BricsCAD

Why use paper space? Here's the #1 Rule of CAD: all geometrygets created at full (1:1) scale. You drawin model space (MSPACE), and you can use the VPORTS command to create different views of your drawing / model on the screen. This is a great strategy when you're creating a drawing. It's not so easy when you need to fit yourfinished drawings into a standard sized title block. Yes, it is possible to create a drawing in model space and make it fit on a printed page. But why?**Using paper space (PSPACE) is the correct way to create accurate, reusable drawing layouts!** Creating drawing sheet layouts is super easy in paper space. In this shortvideo by CADexpertRalph Grabowski, you'll get started creating layouts that update automatically as your model geometry changes. This alone is a huge reason to use PSPACE. Think of a layout as a scale-able, zoom-able set of windows ("viewports") that helpyou see your model from different points in space. You can control the view, the scale, the orientation and the properties of each viewport in a layout. Ralph's video starts simply, helping you create and move viewports in a new layout tab. Next, you'll see how to change what you see in each new viewport. Then, he'll show you how you can modify, scale or change the properties of each individual viewport in a layout tab. You'll use visual styles to change the way that your drawing is displayed in each PSPACEviewport. Then, he will show you how tonon-rectangular and clipped viewports, how to turn any closed entity into a viewport border and how to hide viewport boundaries using LAYERs. Back to Tutorial Overview

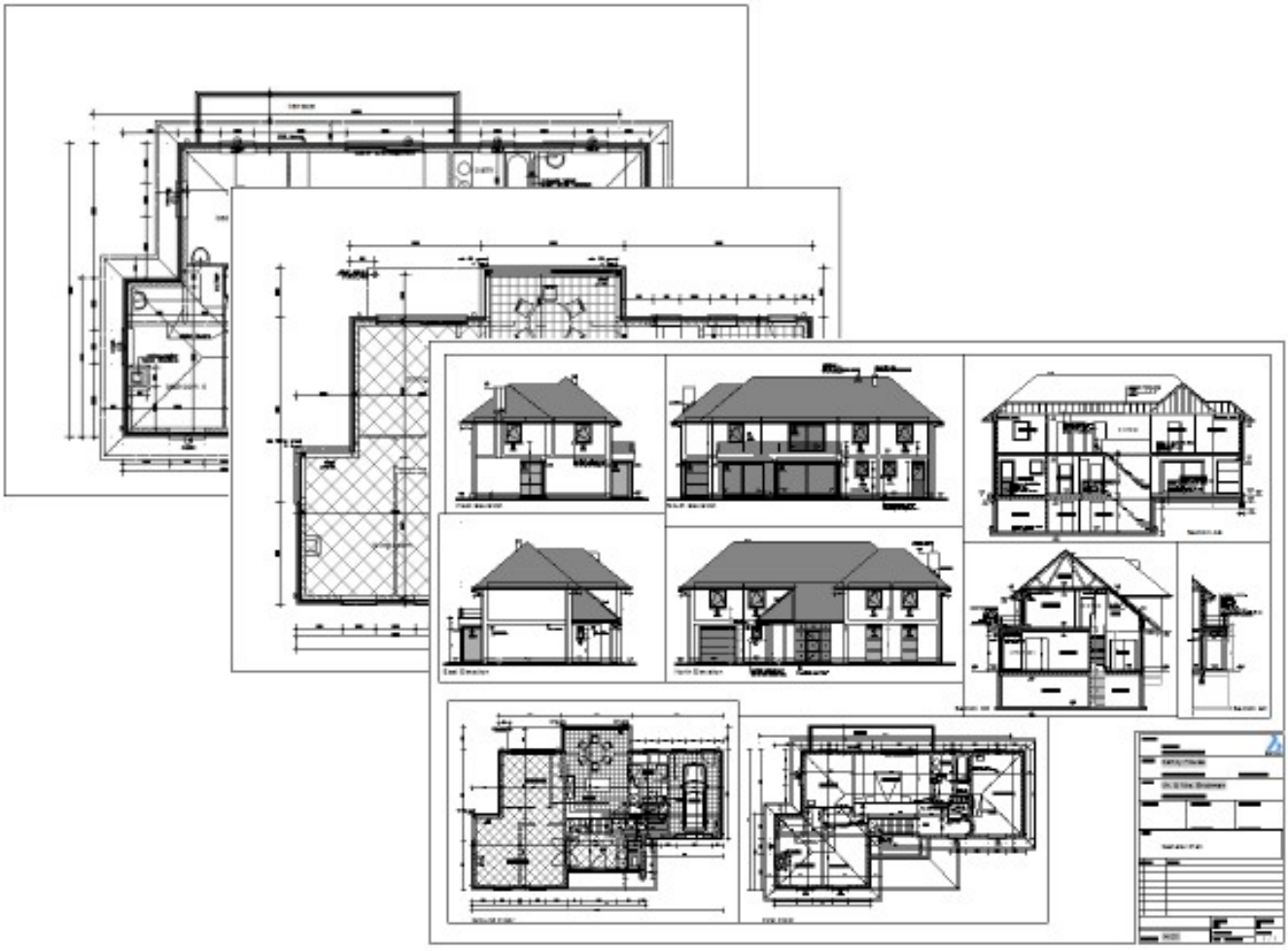
1.8.3 Printing in BricsCAD: Paper Space Layouts

https://www.youtube-nocookie.com/embed/t6zwMl8xmLI This is a four series tutorial about printing in BricsCAD. We'll start with the paper space layouts.Although it is perfectly possible to print a drawing from model space, paper space layouts are especially designed for printing purposes.

Multiple layouts

Each drawing can have multiple layouts, with multiple viewports in each layout. The print settings are controlled by a page setup, while viewports define the portion of the model space to be printed. The layer visibility and scaling can be different for each viewport. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360010091474/i
1-2.jpg})

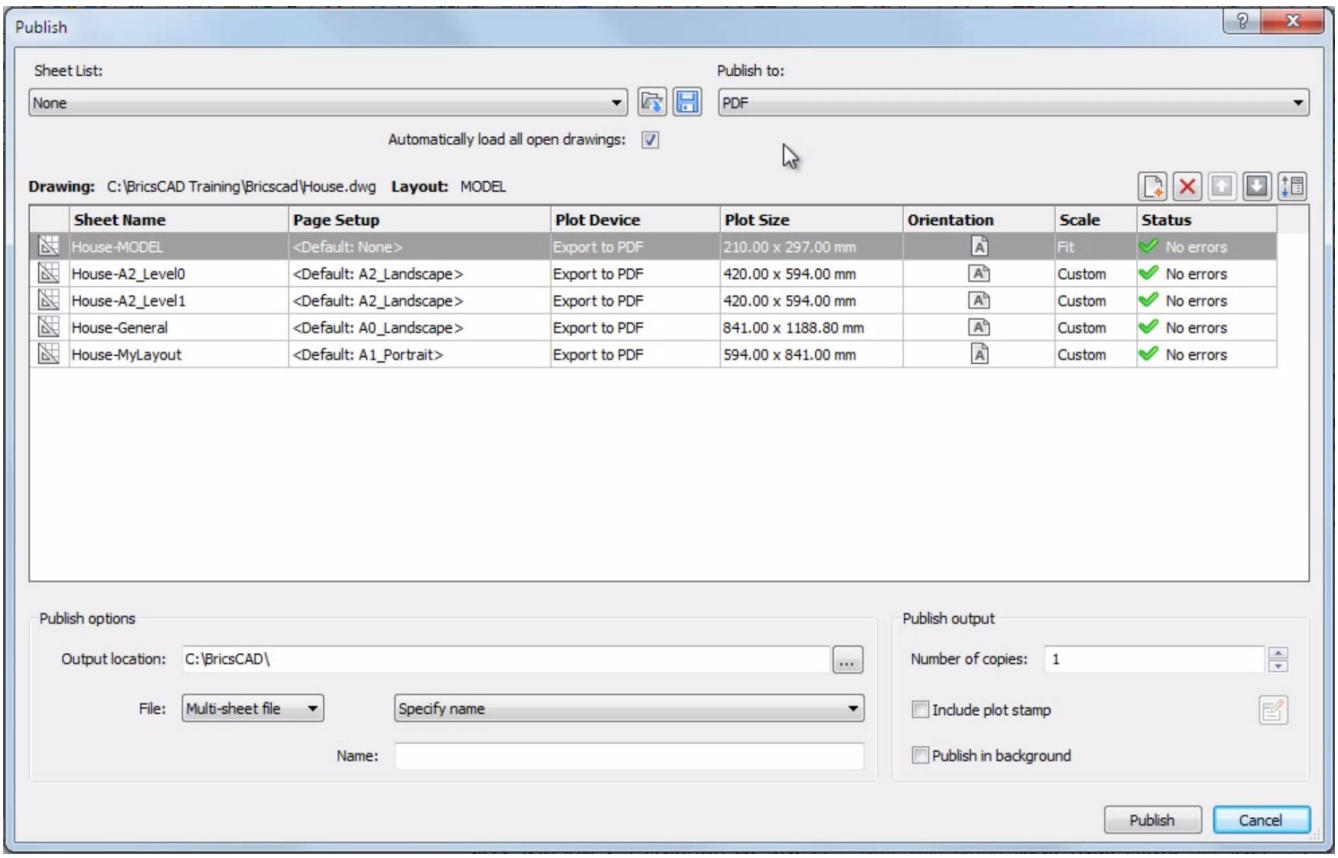
Figure 1.22



Publish to PDF

Print all layouts in a drawing to a single, multi-page pdf. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachm
1-2.jpg}) Back to Tutorial Overview

Figure 1.23


Publish
Sheet List:
None
Publish to:
PDF
Automatically load all open drawings:
Drawing: C:\BricsCAD Training\Bricscad\House.dwg Layout: MODEL
Sheet Name
Page Setup
Plot Device
Plot Size
Orientation
Scale
Status
House-MODEL
<Default: None>
Export to PDF
210.00 x 297.00 mm
Fit
No errors
House-A2_Level0
<Default: A2_Landscape>
Export to PDF
420.00 x 594.00 mm
Custom
No errors
House-A2_Level1
<Default: A2_Landscape>
Export to PDF
420.00 x 594.00 mm
Custom
No errors
House-General
<Default: A0_Landscape>
Export to PDF
841.00 x 1188.80 mm
Custom
No errors
House-MyLayout
<Default: A1_Portrait>
Export to PDF
594.00 x 841.00 mm
Custom
No errors
Publish options
Output location: C:\BricsCAD\
File: Multi-sheet file
Specify name
Name:
Publish output
Number of copies: 1
Include plot stamp
Publish in background
Publish
Cancel

1.8.4 Printing in BricsCAD: Viewports

https://www.youtube-nocookie.com/embed/cGFD_GOuc7w

Create page setups and layouts using viewports.

Use page setups to define print settings Page Setups control:

- Printer configuration
- Paper size and orientation
- Plot style table, which specifies how entity colors are used to control color and line weights.

Freeze layers in a viewport In the VPFreeze column of the Drawing Explorer – Layers dialog, select the layers you do not want to be printed in the current viewport. Back to Tutorial Overview

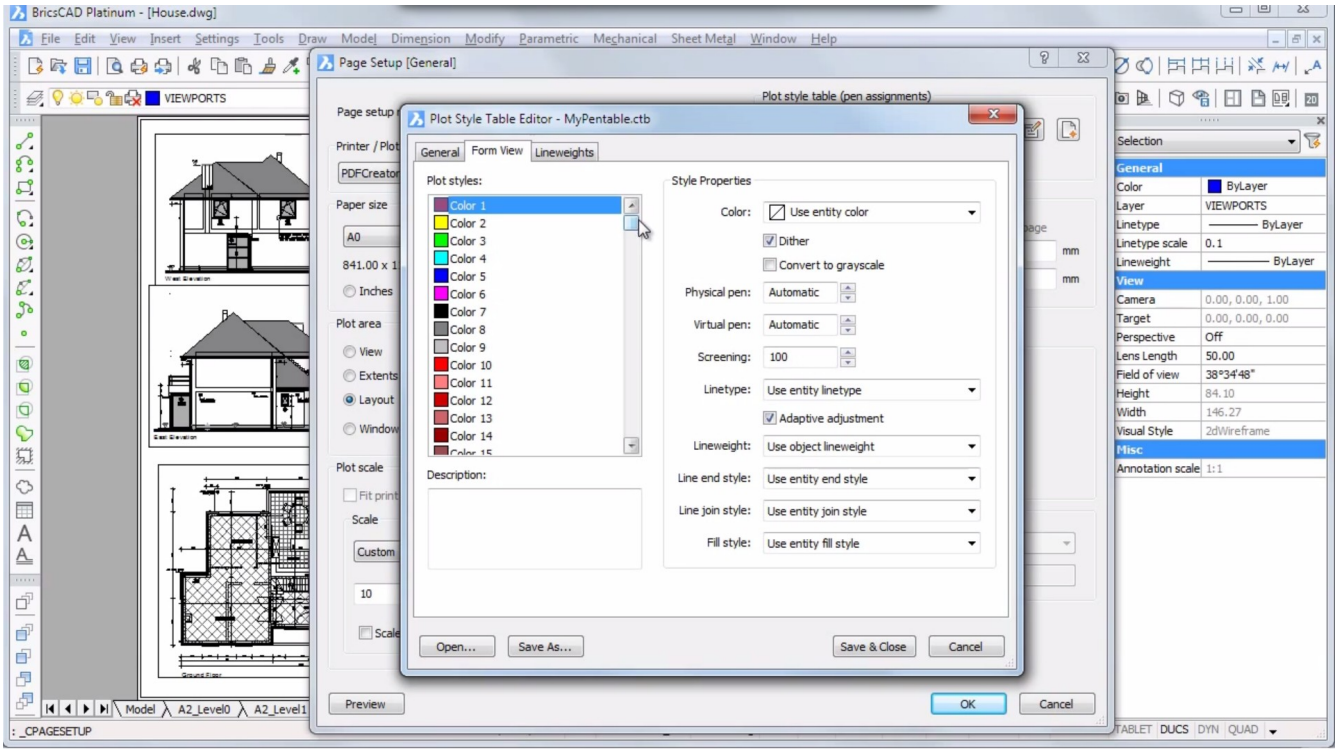
1.8.5 Printing in BricsCAD: Plot Style Tables

https://www.youtube-nocookie.com/embed/DLVvwPWPhCg

Control the color and line-weight of your printout with Plot Style Tables.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360010143433/imageContent2-4.jpg})

Figure 1.24



Editing line weights

Select a line weight, then click the Edit button to modify the line weight. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/h
2.jpg}) Back to Tutorial Overview

Figure 1.25

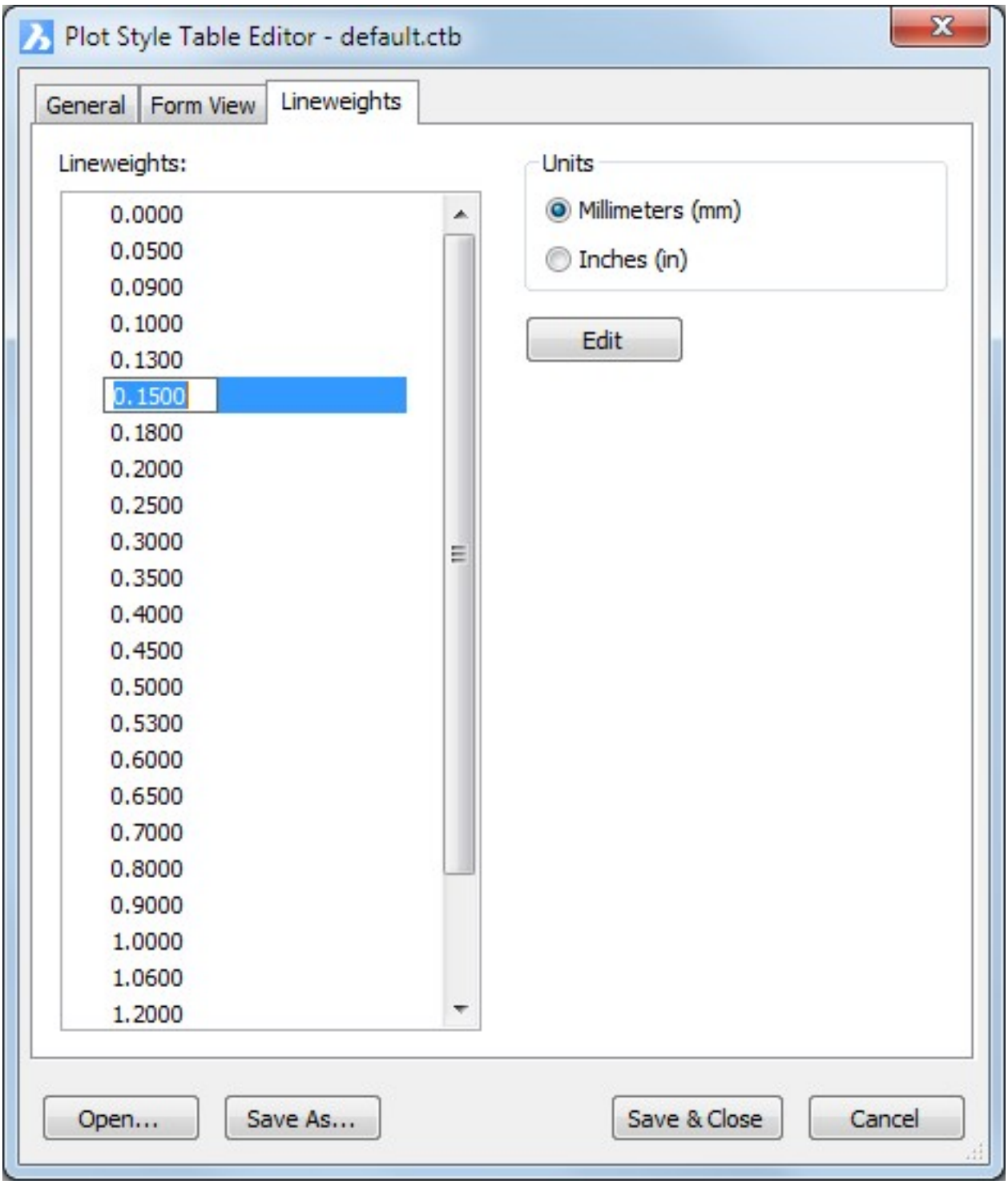
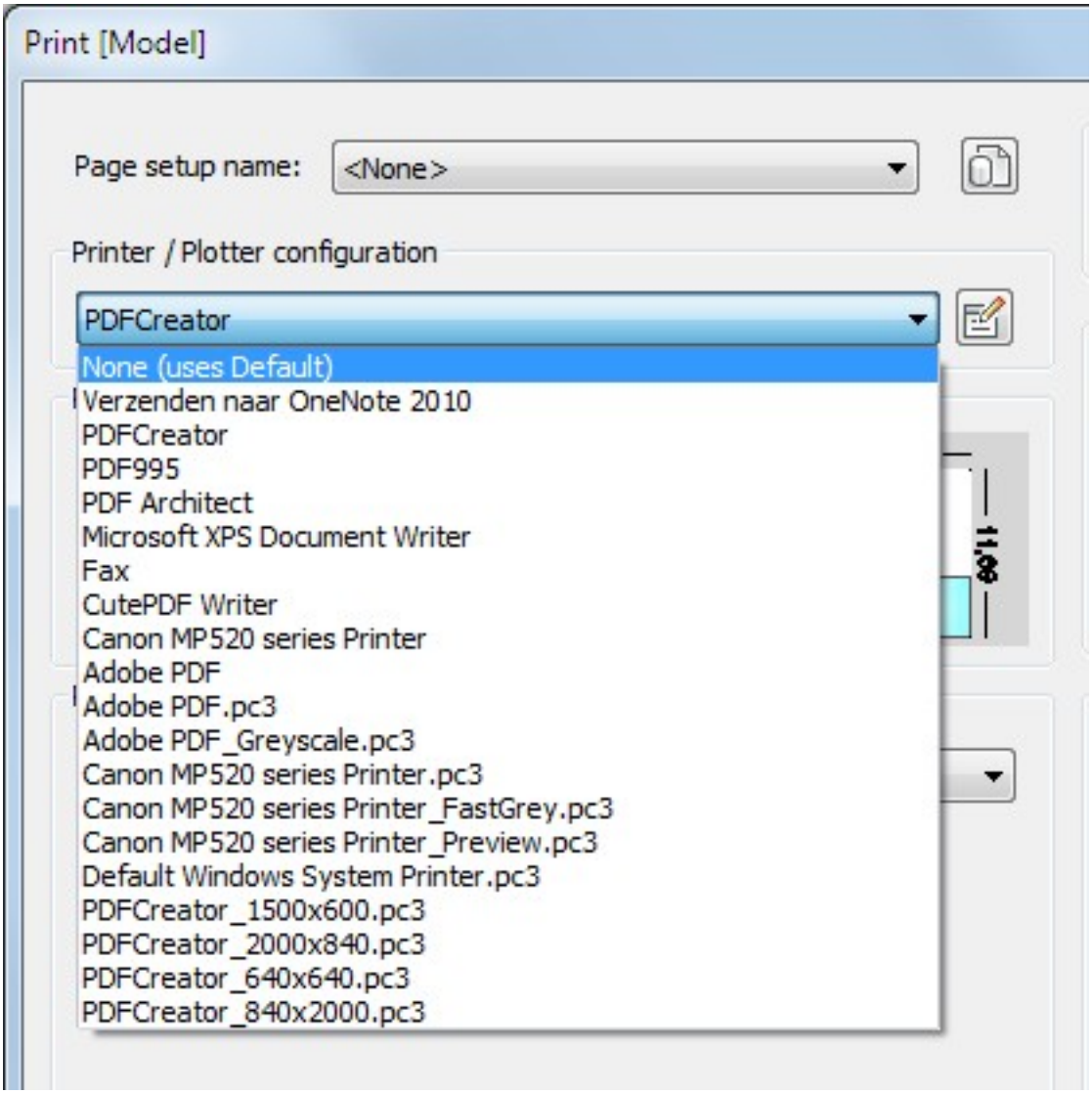

Plot Style Table Editor - default.ctb
General
Form View
Lineweights
Lineweights:
0.0000
0.0500
0.0900
0.1000
0.1300
0.1500
0.1800
0.2000
0.2500
0.3000
0.3500
0.4000
0.4500
0.5000
0.5300
0.6000
0.6500
0.7000
0.8000
0.9000
1.0000
1.0600
1.2000
Units
Millimeters (mm)
Inches (in)
Edit
Open...
Save As...
Save & Close
Cancel

Figure 1.26



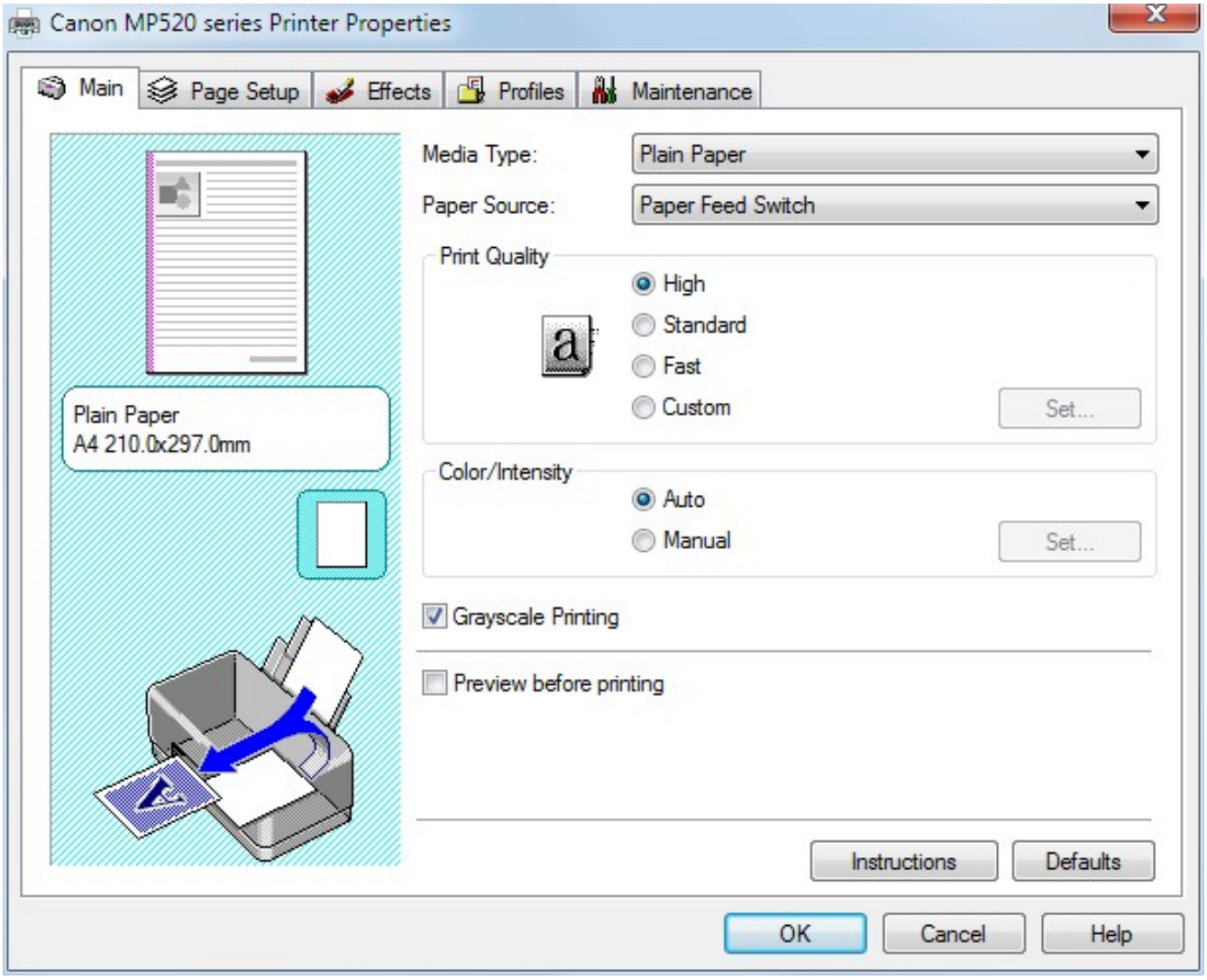
1.8.6 Printing in BricsCAD: Plotter Configurations

https://www.youtube-nocookie.com/embed/TqJVK0GE2SY When printing a layout, the available paper sizes and print settings are controlled by a Plotter Configuration File.The Printer/Plotter Configuration list is composed of all printer drivers that are installed on your computer. The Printer Configuration are the files in the folder which is specified by the 'Plotter Configuration Path' user preference. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachr 1-2.jpg})

Create a new plotter configuration

To create or edit a Plotter Configuration click the 'Edit Plotter Configuration' button on the Print or Page Setup dialog. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360010091434/imageContent2-1-2.jpg})

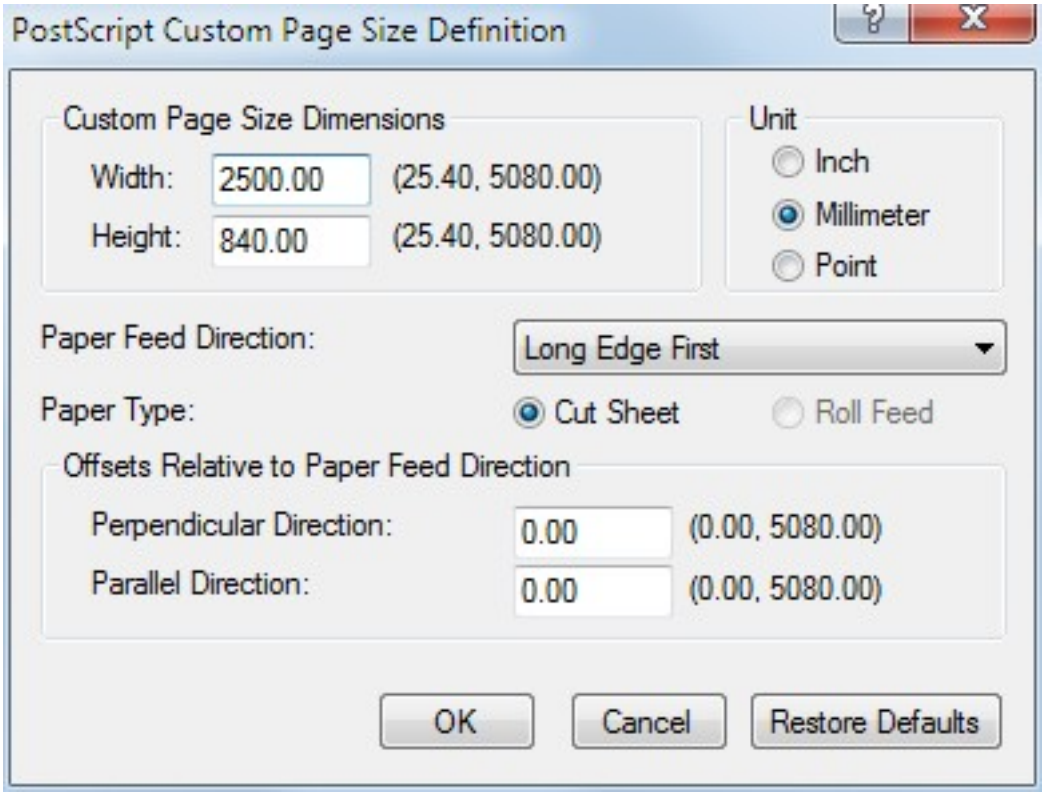
Figure 1.27


Canon MP520 series Printer Properties
Main
Page Setup
Effects
Profiles
Maintenance
Media Type:
Plain Paper
Paper Source:
Paper Feed Switch
Print Quality
High
Standard
Fast
Custom
Set...
Color/Intensity
Auto
Manual
Set...
Grayscale Printing
Preview before printing
Plain Paper
A4 210.0x297.0mm
Instructions
Defaults
OK
Cancel
Help

Define a custom paper size

Type the dimensions of the custom paper size in the Width and Height fields. (\vref {fig:C:/Dev/Task-2448/images/helpbricsysc
1-2.jpg}) Back to Tutorial Overview

Figure 1.28


PostScript Custom Page Size Definition
Custom Page Size Dimensions
Width:
2500.00
(25.40, 5080.00)
Height:
840.00
(25.40, 5080.00)
Unit
Inch
Millimeter
Point
Paper Feed Direction:
Long Edge First
Paper Type:
Cut Sheet
Roll Feed
Offsets Relative to Paper Feed Direction
Perpendicular Direction:
0.00
(0.00, 5080.00)
Parallel Direction:
0.00
(0.00, 5080.00)
OK
Cancel
Restore Defaults

1.8.7 Creating Plot Stamps

https://www.youtube-nocookie.com/embed/wxbc5RCivms Plot stamps are text that appears on drawings only when plotted. They provide information like the date and time the plot was made, the file name, and so on.BricsCAD adds plot stamps through an option in the Print dialog box. For greater flexibility, however, consider using field text. This video shows you how to use both methods. Back to Tutorial Overview

1.9 Programming

1.9.1 Unlock BricsCAD's extra features

https://www.youtube-nocookie.com/embed/7Pg4OEQRwN4 BricsCAD has quite a lot of commands that don't appear on the menu, ribbon, or toolbars.If you don't know about these, then you're missing out on features. In this video we'll show you how to find these commands, and then highlight a couple that you may find useful.

Don't miss out on features!

Back to Tutorial Overview

1.9.2 The Case for Using LISP Today, Part 1

LISP: Whatis old isstill very new

Welcome to Part 1 of my first post on the Bricsys Blog! I'll be writing about LISPprogramming and transitioning your existing code into the BricsCAD environment. I'll review implementation, differences from other environments, tips & tricks, and other topic requests from our readers. Or, if you haven't started looking at LISP, maybe I can give a little encouragement andlet you know that LISP is alive and well. It is definitely a language worth considering for BricsCAD customization. First, let me state that I've written a couple hundred thousand lines of LISPover the last few decades. But I still would not call myself an expert. I don't know allof the vl-commands, orsome of the more "advanced" functions. Please forgive me if I present a method that isn't the most efficient, or if I suggest a paragraph of code that could be replaced by a single function. Chances are good that I had no idea that the "better" function existed! I still stumble across "new" (to me) commands on a regular basis. When I do, I often wonder how much timethat newly-discoveredfunction mighthave saved me. A while back I was questioned by a client about my continued coding in LISP. I actually thought it may have become a skill set that had become obsolete. Think about(manual) drafting skills and mastery of the Vemco Drafting Machine – now, those areobsolete! But whenI looked at my customer's real needs, I found more and more reasons to update or develop new LISP functions. I think that it will be fun to share some of these concepts with you to help you get started.

Is LISP the Rosetta Stone of CAD customization?

The name "LISP" comes from the words "List Processing". Data is stored in .dwg files as lists, and LISP reads, writes, changes and stores that data. Best of all, thiscode is platform independent. It's efficient, small and flexible. I prefer thesimplicity of developing code in a non-compiled language. WhileLISP code does not need to be compiled, you can encrypt your programs to protect your source code. Many programsthat I wrote twentyyears ago on "vintage" hardware/software/OSes still work today in current versions of BricsCADwith little or no modifications. Recently, I was asked to solve a migration issue for a client who was transitioning from AutoCAD™ 2014 to 2016. This company's standard CAD setup included over 400 program files with 123,000 lines of LISP code, with around 4,000 functions defined. After a bit of research, I discovered that the primary issue was related to a couple of (compiled) ARX applications that were not compatible with AutoCAD™ 2016. **Their LISP code, much of which was written in the late 90s, ran perfectly.** When I'm asked to consult on CAD standards migration, I first determine how much of a company's workflow is dependent on 3rd party tools. Then, I check to see if their tools needspecific software versions or platforms. Often, the only issues revolve around hard-coded folder paths that need to be updated.

Why use LISP for custom CAD programs?

So, what are some reasons to create custom programs in your CAD environment? Typically, I write codeto improve process or workflow automation, relating to specific CAD entities or CAD standards. Routines can be as simple as a few lines of code. These functions can perform layer manipulations, change entity properties, automate drawing and XREF relationships, or help standardize plotting and publishing workflows. Or, theseroutines can be as complex as a parameter-driven design process, or an application to automate database access. As I mentioned earlier, a key benefit to using LISP is simply the small size and great efficiency of the language in a CAD environment. In the example we'll explore in Part 2 of this post, the LISP code to do simple geometry manipulation (move and rotate) can be a fraction of what one might have to write in a compiled language (.NET,C, C++, et al). For the sake of page space, I'm not including a .NET example here. But, if you really want to see what I'm talking about, just Google: **"Create a move command with .NET API"** and you'll see a few examples. These compiled languages often require hundreds to thousands of lines of code to create the graphical behaviors and intuitive interfaces that users expect in a day-to-day CAD environment. (This postwill be continued in Part 2 (1.9.3)) You can customize BricsCAD by adding custom programs written in any of several programming languages that run within BricsCAD.See here which API's are available**. Back to Tutorial Overview**

1.9.3 The Case for Using LISP Today, Part 2

Copy & rotate the easy way with LISP

In contrast to the .NET example I pointed you to in the last episode of this blog, here is a simple example of a LISP command used to Copy & Rotate objects: Now that's a short program! Like I mentioned above, this "command" combines the Move and Rotate commands in to a single command.

The same program, with comments:

In LISP, the semi-colon marks a comment line, so anything after is not processed. To improve the usabilityof our function, we could add a "while" loop (to keep asking for more selections), and give angular options, turn Osnaps or Ortho on/off, etc…

Is a LISP routine too much?

Having presented this, I will tell you that there are many cases where a LISP routine can be over-kill. If you are simply executing a string of commands that don't require "manipulation" of the data between functions, or where the entry of data is consistent, then menu macros can be even more efficient than LISP. But, menu macros can't be designed with choices and they can be "brittle" – that is, easy to break – as they can't handle errors. My recommendation? Here's a bit of a hybrid – it's a one-line LISP routine: ***^c^c(command "Select" pause "copy" "p" "" "@" "@" "move" "p" "" pause pause "rotate" "p" "" "@" pause)** This simple, yet elegant LISP "macro" functions exactly as the program above, with the added automatic function repeat (*^c^c), and gives all the dynamic graphical user feedback of a sophisticated program. Simply add an icon to one of your menus using the CUI interface and plug the line above into the command string, and you have a super cool copy / rotate function.

The right tool for the job

You might be thinking that it's just as easy to use Grips in BricsCAD and right-click to cycle through all the options. My reply would be to remind you that I'm (really) old school, and while I use Grips often, I also know the Esc key has always been the first key to die from abuse on my keyboard (OK, not really). The main point here is to make sure you are using the right tool for the job, and while there are certainly valid uses for .NET applications, I can't bring myself to write pages of compiled code when I can do the job cleanly with a few lines of LISP. Hopefully, this introduction has peaked your interest. I think that you will find some useful tips/suggestions as you dive into the world of LISP. Remember, it's all about getting your design work done quickly and accurately. What better way than using the features built into BricsCAD to make it work your way? Back to Tutorial Overview

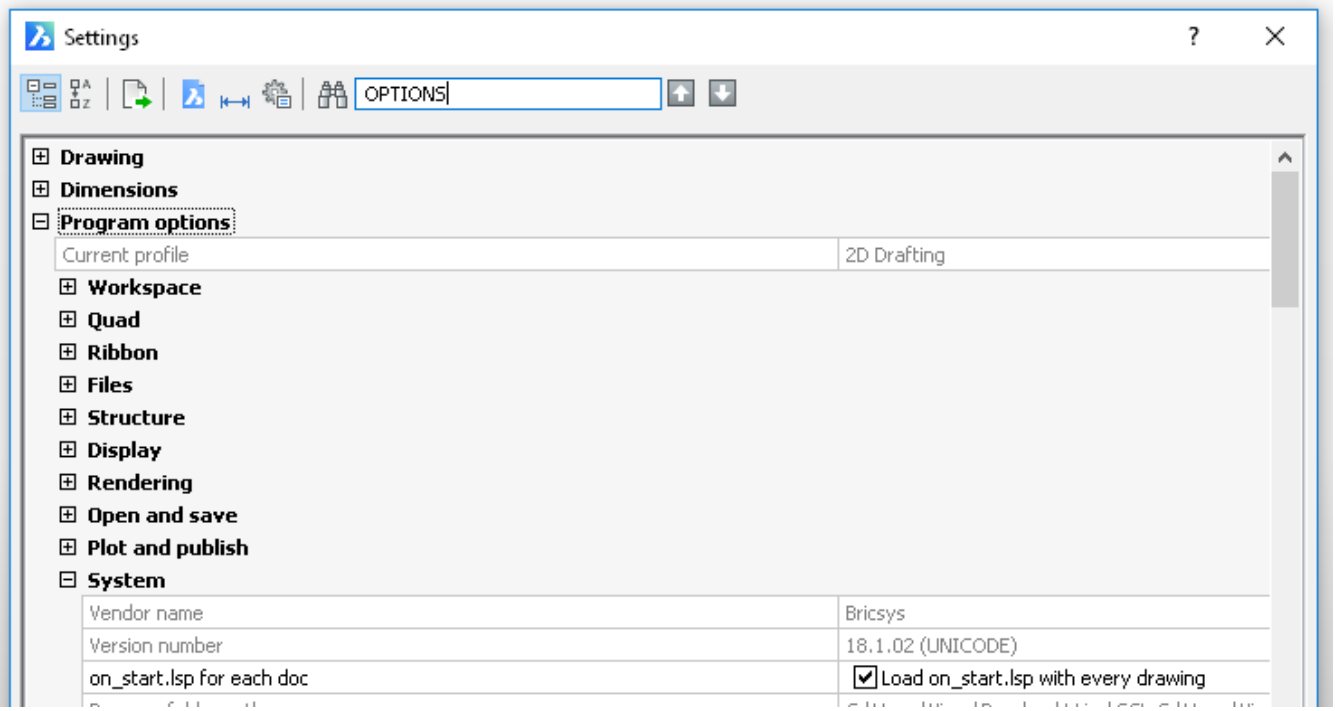
1.9.4 BricsCAD Lisp – Migrating from OtherCAD

BricsCAD LISP versus OtherLISP

If you've read my previous posts (**part 1**and **part 2**), you know that BricsCAD fully supports theLISP tools that you and your company have relied upon for years (perhaps decades). Now, as you move to BricsCAD, you are ready to migrate your environment and standards. What are some of the items to consider? For the sake of this discussion, we are going to focus on LISP customization and not on menu and custom user interface (CUI) modifications. However, I would first recommend using thedefaultBricsCAD interface for a week or so beforeyou make any changes to the menus. For me, I found that I actually preferred the standard layout and command access of BricsCAD over the "other" environment. Once I discovered where all commands were located, I found that I wasn't toggling between tool menu palates as often as I did in OtherCAD. As a side note… you know you are truly "old school" when you prefer to type short-cut commands with your non-mouse hand, rather than picking through 2 or 3 levels of menus. The more I use BricsCAD, the less I seem to rely on those keyboard short-cuts. The streamlined user interface and the quad cursor actually make it easier to use menus than to type. Try it, you'll love it.

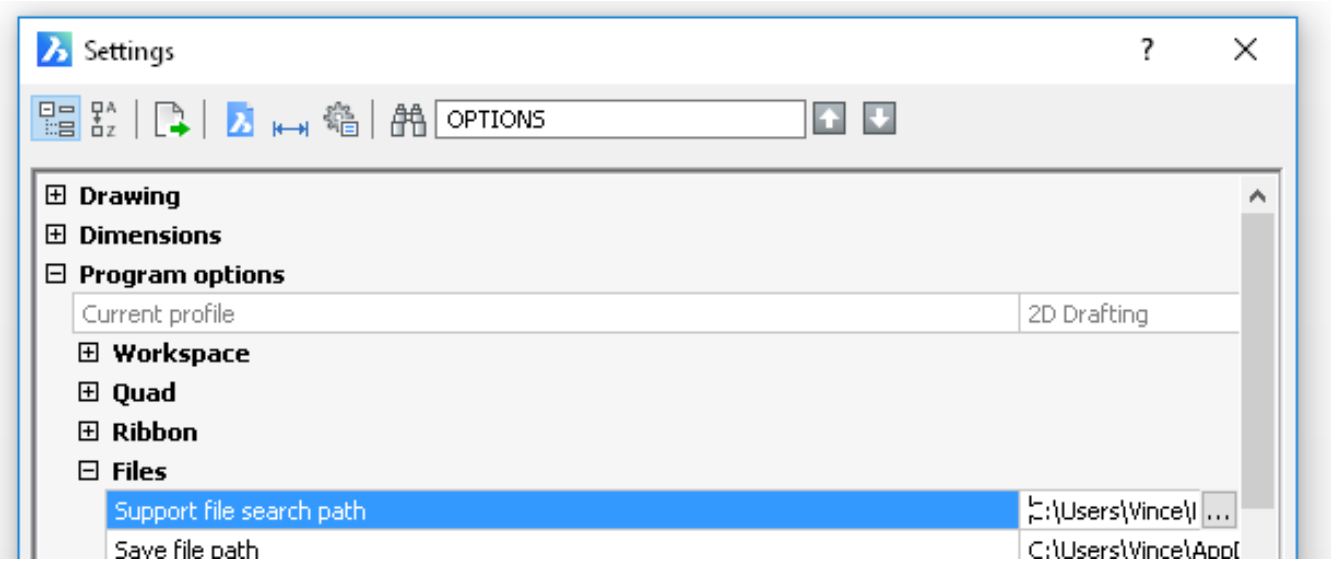
Almost no differences When you're ready to start migrating, you'll find that there are virtually no differences in BricsCAD LISP versus OtherLISP. Your codeloads and runs, and the functionality is identical. The primary differences when "porting" your apps will be a few minor setup steps, command line structure, and possibly file locations. To start, let review the setup and launch of your existing programs in BricsCAD. If you are auto-launching your programs using **Acad.lsp** or **Acaddoc.lsp**, you will simply need to rename and or combine those files into a single file named "**on_start.lsp**". To enable this feature, simply toggle the option in Settings / Program options / System: (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/11/LispP3SettingsDialog01.png}) I prefer to always enable this option.

Figure 1.29



Enjoy the BricsCAD Settings Panel I would also recommend that you prepare to spend a little time familiarizing yourself with the Options/Setting menu in BricsCAD. As you dive in, you will find a ton of settings and options you probably wish you had in your "other" CAD environment. Welcome to BricsCAD! The next step in the settings dialog would be to add your support folders to the Files search path, which is found within Settings / Program Options / Files: (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/11/LispP3SettingsDialog02.png}) As a rule, I never hard code any paths unless absolutely necessary. The support paths I use will typically be added as in the manner above and/or located one level below the current project folder. Typically, I will use a "findfile" function to validate the file exists before proceeding with my commands, as in the method below: `(if (findfile "custom-code.lsp") (load "custom-code"))` In fact, this is one of the methods I've used to demand-load my functions within the **on_start.lsp** file: `(defun c:my-function () ; this defines the command(if (findfile "custom-code.lsp") ; check if the file exists(load "custom-code") ; if True, launch(alert "Custom-code not in Support Path!") ; not True, so alert) ; end If statement) ; end function` The command defun (DEfine FUNction) is the same name as the function inside ofyour **on_start.lsp**file and that function redefines the calling function when it's launched. This is another cool reason for using this method, as it demonstrates how dynamic LISP can be!

Figure 1.30



Take advantage of your move to BricsCAD... When your engineering workflow is running, you usually build along list of maintenance items that you keep putting off. Migrating to a new CAD environment offersan opportunity to make some of those updates andupdateyour documentation standards. Almost every client that I've helped to migrateneed to address old-style hard-coded paths, 8.3 syntax filenames, old layer naming conventions and other no-nos within their standards. As we move forward in this series, I will share some of the methods I use to store user settings and defaultsexternal to yourLISP code, so that they can be updated more easily. Until next time, happy coding! Back to Tutorial Overview

1.10 Getting Started

1.10.1 Welcome to BricsCAD

https://www.youtube-nocookie.com/embed/s57w237tmBM Whether you're a novice user or an experienced CAD professional, you'll soon be creating drawings using the numerous productivity-enhancing features in the software. With hundreds of third party solutions, BricsCAD is the powerful CAD software platform unifying 2D CAD and 3D direct modeling for .dwg on Windows, Mac and Linux. These are just a few of the many features that we have included in the program to make the transition into CAD-based drawing a smooth one for new users. We have also incorporated many features, that experienced CAD users have been requesting for years, in a drawing package. You have chosen an affordable, high-quality software program to produce your drawings. We are convinced you will be pleased with the results! This help file describes all the BricsCAD features. Some features do not apply to the Classic and/or Pro versions; some commands are not supported on the Linux and/or the Mac platforms. Please see the Command Reference (6.36.1) to check the availability of a command on the various platforms (Windows, Linux, Mac) and feature levels (Platinum, Pro or Classic).

1.10.2 Differences between platforms and levels

WINDOWS MAC LINUX Platinum Pro Classic Platinum Pro Classic Platinum Pro Classic CAD functionality □ □ □ □ □ □ □ □ □ □ **2D constraints solving** □ □ □ □ □ □ □ □ □ □ **Full LISP support** □ □ □ □ □ □ □ □ □ □ **Bricsys 24/7 connectivity** □ □ □ □ □ □ □ □ □ □ **Direct modeling** □ □ - □ □ - □ □ - **Generated views** □ □ - □ □ - □ □ - **Rendering** □ □ - □ □ - □ □ - **VBA** □ □ - - - - - - - **BRX** □ □ - □ (1) □ (1) - □ (1) □ (1) - **3D constraints solving** □ □ - □ □ - □ □ - **3D constraints creation** □ - - □ - - □ - - **Design intent recognition** □ - - □ - - □ - - **Assembly modeling** □ - - □ - - □ - - **BIM** □ (2) - - □ (2) - - □ (2) - - **Sheet metal** □ (2) - - □ (2) - - □ (2) - - **Communicator** □(3) □(3) - □ (3) □(3) - □(3) □(3) - (1) Without MFC classes (2) Optional - Requires a specific license. (3) Optional - Requires a specific license. Communicator is an additional module which allows to import and export 3D CAD data from and to all the major MCAD programs. View the complete comparison charton the Bricsys website.

Related topics

Licensing (1.11.3) Activation (1.11.5)

1.10.3 License Options

Choose your license level

BricsCAD licenses are available in 6 levels: Classic, Pro, Platinum, BIM, Mechanical and BricsCAD Ultimate.

BricsCAD Classic If you create general 2D designs, the Classic license may be right for you. Compared to AutoCAD® LT, it also allows to view and edit 3D models and running Lisp routines.

BricsCAD Pro. If you do any 3D modeling or run third-party CAD applications BricsCAD Pro is the right choice.

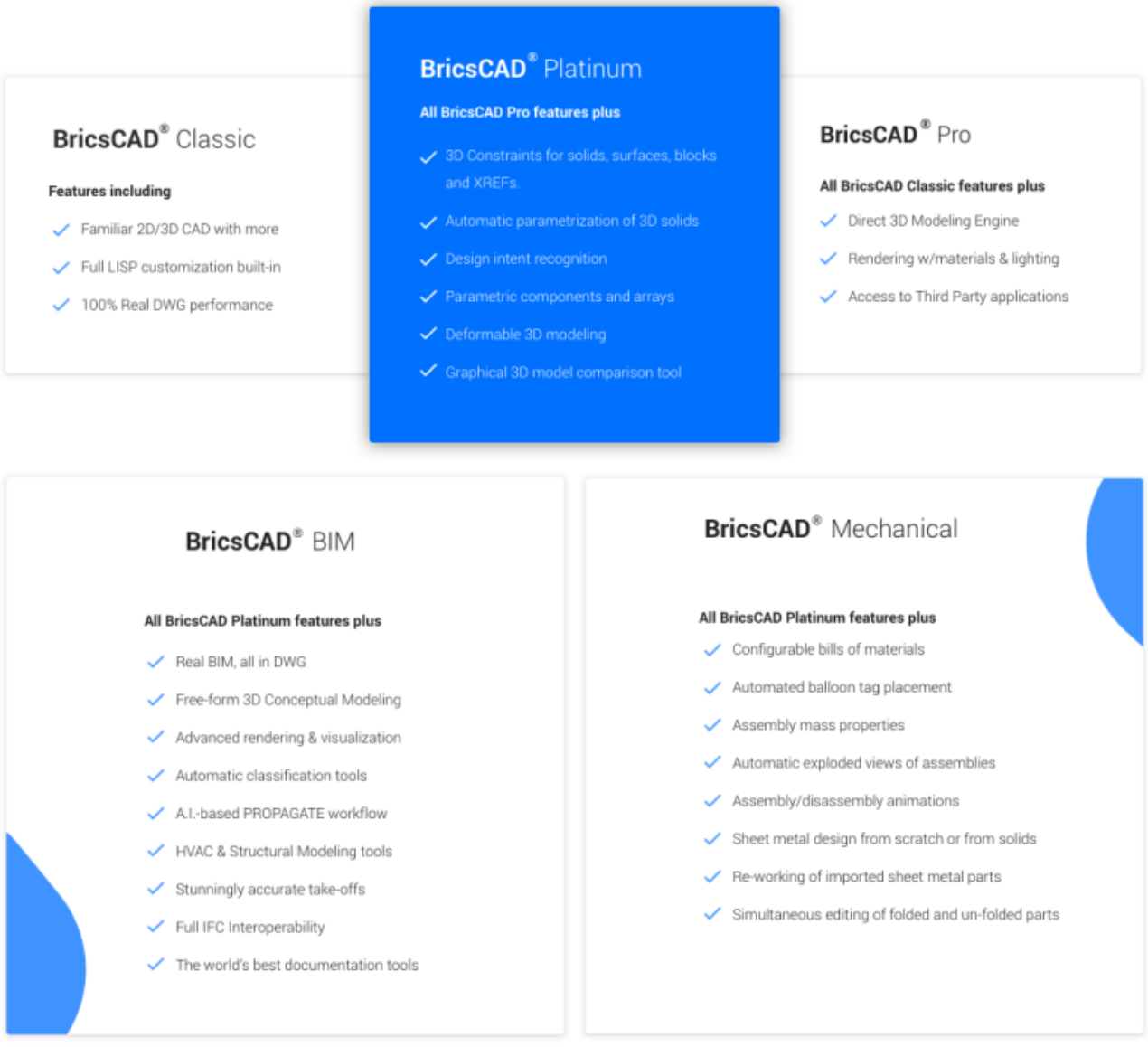
BricsCAD Platinum BricsCAD Platinum includes direct 3D modelling with a full 3D constraint system, Rendering with materials and lighting and access to third party applications.

BricsCAD BIM A BricsCAD BIM license includes all Platinum features. You can use your current CAD skills to smoothly move to creating real Building Information Models in record time. One product, one workflow, all in industry-standard DWG.

BricsCAD Mechanical On top of all Platinum features, a BricsCAD Mechanical licenseoffers a complete mechanical design toolkit, all in .dwg. , including sheet metal functionality.

BricsCAD Ultimate This license level combines the BIM and Mechanical licenses. (\vref {fig:C:/Dev/Task-2448/images/static

Figure 1.31

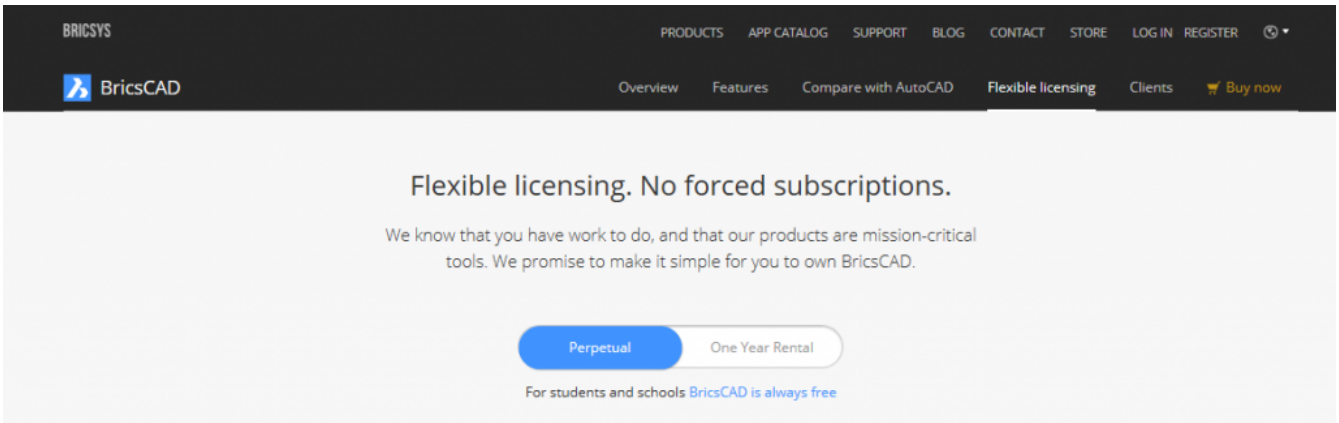

BricsCAD® Classic
Features including
Familiar 2D/3D CAD with more
Full LISP customization built-in
100% Real DWG performance
BricsCAD® Platinum
All BricsCAD Pro features plus
3D Constraints for solids, surfaces, blocks and XREFs.
Automatic parametrization of 3D solids
Design intent recognition
Parametric components and arrays
Deformable 3D modeling
Graphical 3D model comparison tool
BricsCAD® Pro
All BricsCAD Classic features plus
Direct 3D Modeling Engine
Rendering w/materials & lighting
Access to Third Party applications
BricsCAD® BIM
All BricsCAD Platinum features plus
Real BIM, all in DWG
Free-form 3D Conceptual Modeling
Advanced rendering & visualization
Automatic classification tools
A.I.-based PROPAGATE workflow
HVAC & Structural Modeling tools
Stunningly accurate take-offs
Full IFC Interoperability
The world's best documentation tools
BricsCAD® Mechanical
All BricsCAD Platinum features plus
Configurable bills of materials
Automated balloon tag placement
Assembly mass properties
Automatic exploded views of assemblies
Assembly/disassembly animations
Sheet metal design from scratch or from solids
Re-working of imported sheet metal parts
Simultaneous editing of folded and un-folded parts

Communicator for BricsCAD If you design for the manufacturing industry and need to exchange data with other popular manufacturing applications, you should consider adding Communicator to your Pro, Platinum, BIM or Mechanical license. Communicator running on BricsCAD Pro imports parts. When running on Platinum, BIM or Mechanical it imports full assemblies with embedded PMI (**Product** and **Manufacturing Information**).

Choose your license type

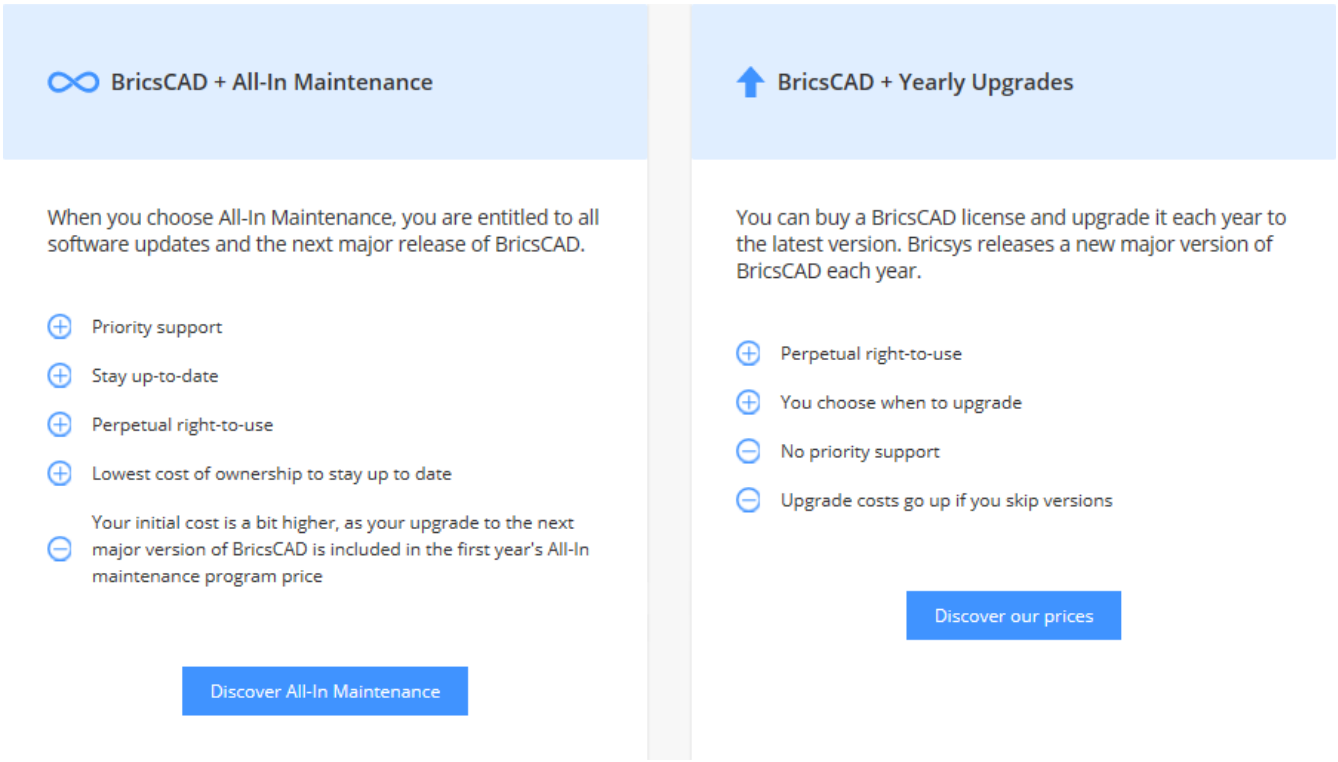
BricsCAD offers flexible licensing options so you can purchase the right software with terms that best meet your needs. You can buy a perpetual license or an annual subscription. Additional options include volume, network, and academic licensing. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2018/05/HHBricsCAD-License-Options-03-1024x324.png})

Figure 1.32



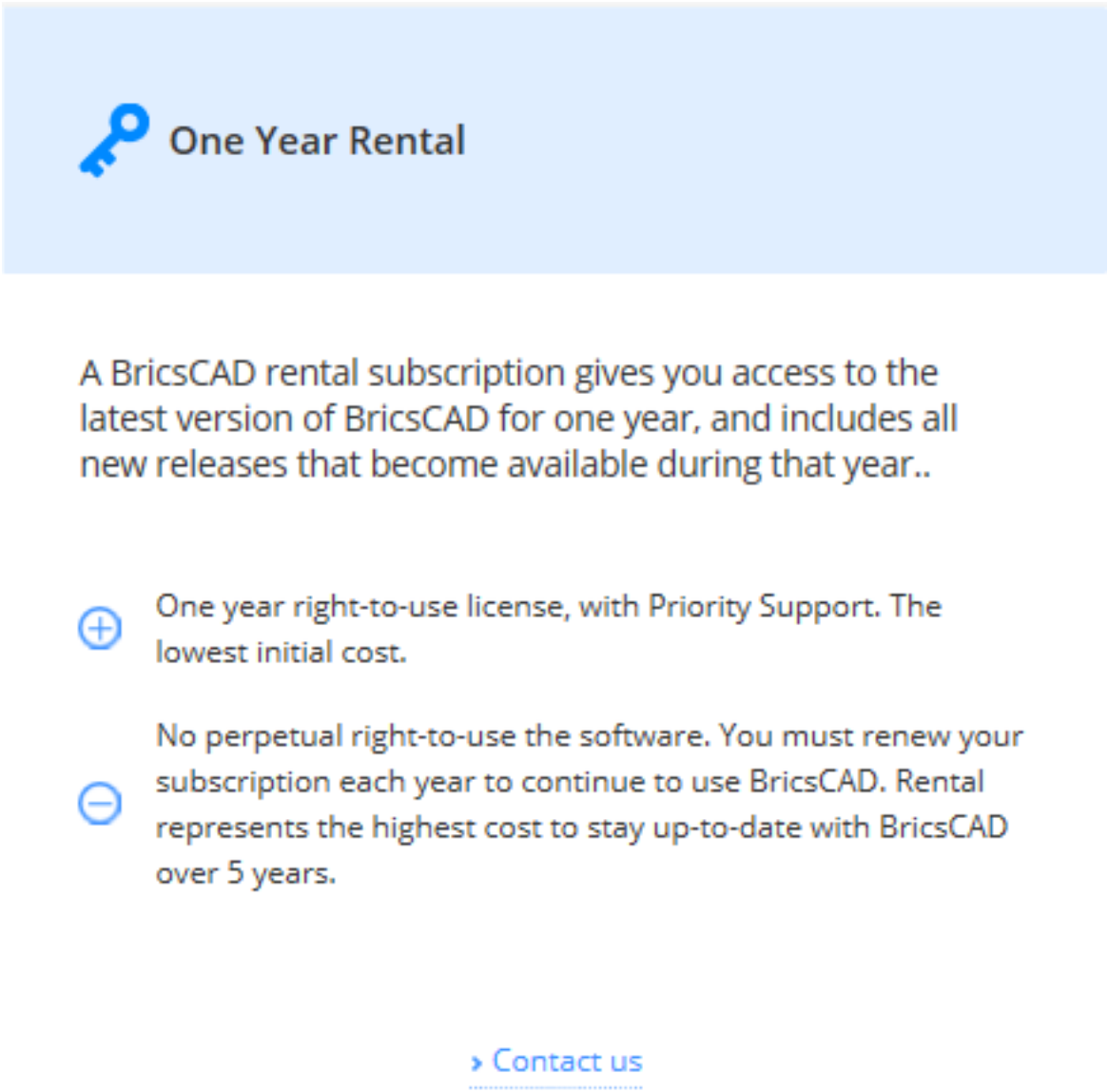
Perpetual license When you purchase a perpetual license, you own the software. And remember, BricsCAD editions run on Microsoft Windows, macOS and Linux – your choice! Pretty amazing, right? You can choose from two options. The All-In Maintenance option includes automatic upgrades and support. It's the least expensive way to stay up to date with BricsCAD. Or, if you prefer to purchase upgrades at your discretion, you have that option! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/Maintenance-UpgradeComparison.png})

Figure 1.33



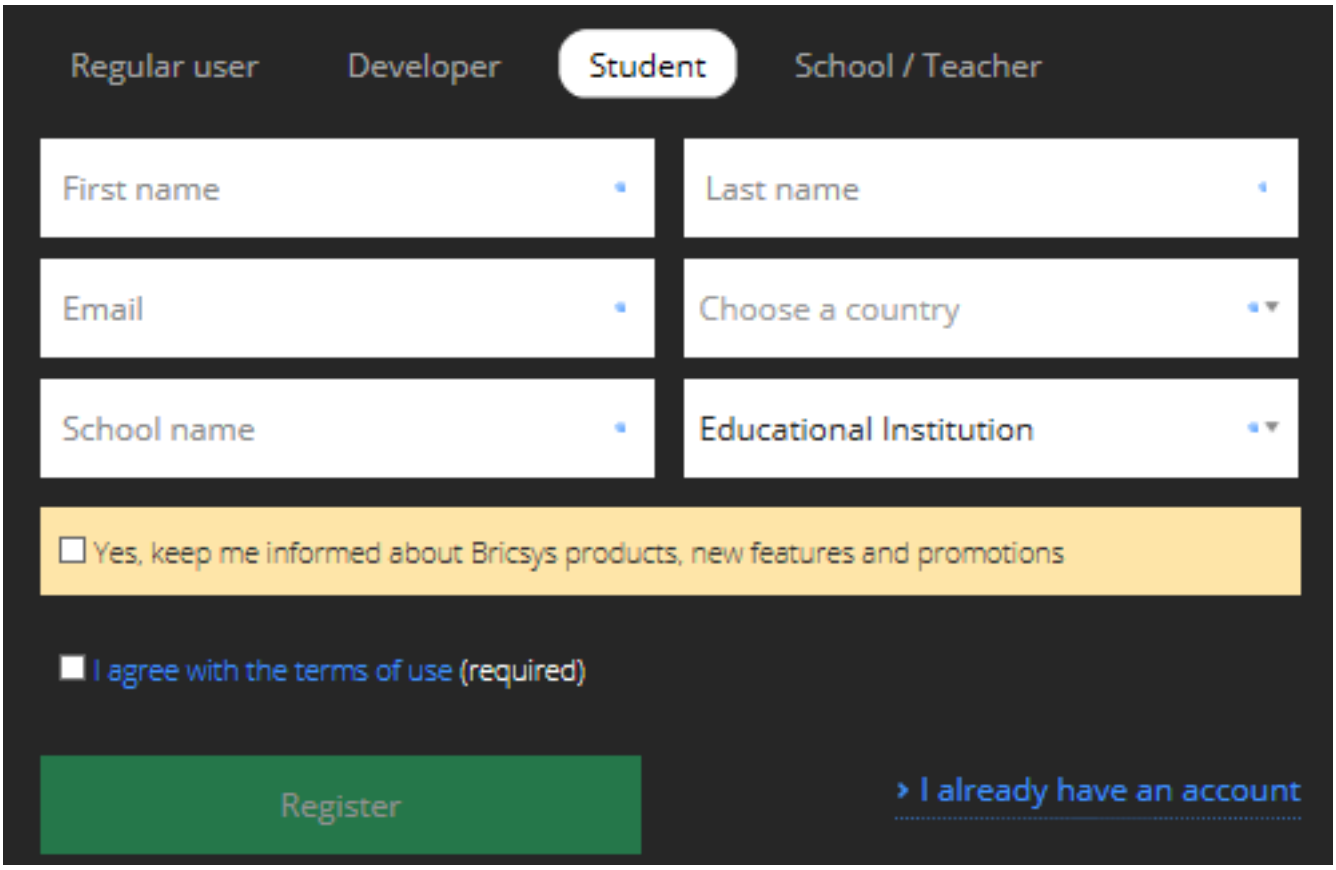
Rental license When you choose the rental option, you have access to the latest version of BricsCAD for a year. It includes Priority Support and access to all new versions that are released during the time your subscription is active. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/OneYearRental.png})

Figure 1.34


One Year Rental
A BricsCAD rental subscription gives you access to the latest version of BricsCAD for one year, and includes all new releases that become available during that year..
One year right-to-use license, with Priority Support. The lowest initial cost.
No perpetual right-to-use the software. You must renew your subscription each year to continue to use BricsCAD. Rental represents the highest cost to stay up-to-date with BricsCAD over 5 years.
› Contact us

Academic license Students, faculty, instructors, and educational institutions have free access to all BricsCAD software for 12 months. And, you can renew it yearly with a current student ID. Simply register as a student or school/teacher with your academic information. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCA

Figure 1.35



Additional license options The most common type of license is for a single-user. However, BricsCAD is also available with network or volume licenses.

Single user license The single user license agreement is for a single user. However, each single user license allows two activations. You can install it on two machines, for example a workstation in the office and a laptop in the field. But, only one can be in use at any time. Single user licensing is available for perpetual or rental licensing. Academic licenses for students are always single-user.

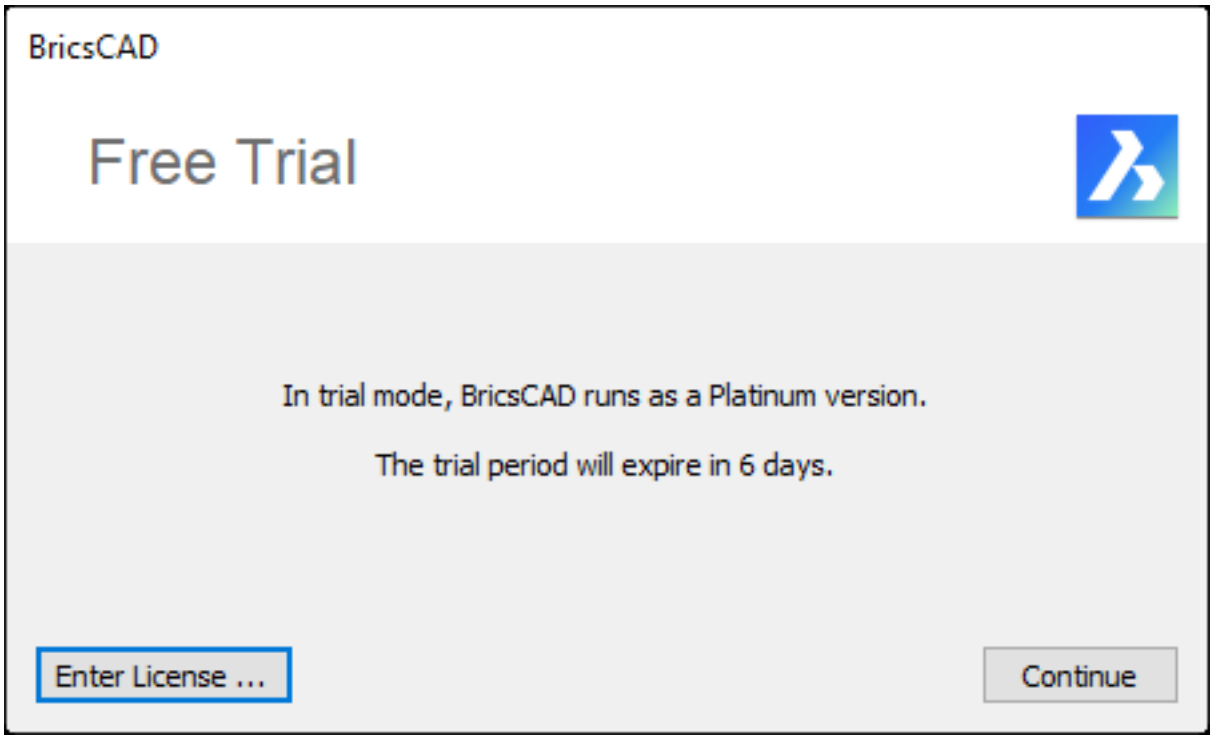
Network license The network licensing agreement enables multiple users to access BricsCAD within a LAN (Local Area Network). The number of available network licenses defines the number of users who can concurrently use the software. The network license option is available with perpetual or rental licensing and includes a one year maintenance contract to get you started. Network licensing is also available to institutions using an academic license.

Volume license The volume licensing agreement allows you to use BricsCAD software within a defined user base. You can install the software using a single license key valid for every user. This greatly simplifies license management. The volume license option is available with perpetual or rental licensing and includes a one year maintenance contract to get you started. And, it's the default for an institutional academic license.

Activate your trial license

Regardless of which licensing option you choose, you'll receive a license key to activate your trial. First, launch the software and then choose **Enter License**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Getti
Enter your BricsCAD license key. If you're using a network license, you may have to contact your administrator for relevant server information. See the LicEnterKey (6.20.23) command for detailed information.

Figure 1.36



Manage your Licenses

Commands: LicenseManager (6.20.22), RunAsLevel (6.26.46) Inside BricsCAD you can view and manage your software licenses in the License Manager. It indicates your active license and enables you to upgrade to a higher license level. Choose Buy to purchase a license for any of the modules directly from the Bricsys store. After you obtain a license key, choose Details. Then, choose Modify and enter the key to activate the module. You must relaunch BricsCAD for the new license level to load. Even with BricsCAD and all the modules activated, you can continue to use RunAsLevel to try the software at lower levels. If you're a CAD Manager, this can help you determine which editions and modules are best suited for various users.

1.10.4 Download and install BricsCAD trial

Download the BricsCAD Free Trial

The download form is simple, without requiring you to provide a lot of excess information. Just enter your email address and then select the options for your download! You can select from Windows, Mac or Linux operating systems in 18 different languages! If you select the Windows operating system, you can also choose whether you want the 32 or 64-bit version. After specifying your download options, agree to the Terms of Use and choose Download. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/DownloadBricsCAD.PNG}) If you haven't yet created a Bricsys account using the email address you entered, you can create one or enter a different email address for which you've previously created a Bricsys account. Then you're ready to download! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/SignInRegister.PNG}) You may have the option to Run the install as soon as it downloads or save the download to install later. Based on experience with other CAD software, you may automatically want to save the download first and then install it another time. Why do you do that? The download and install of some popular CAD applications takes a significant amount of time. So, you separate them into two tasks that you can squeeze into your busy workday. No need to squeeze with BricsCAD! It's fast! You won't even have time to grab a coffee during the few minutes it takes to download and install BricsCAD! You'll just have to find another excuse for your coffee break. In fact, BricsCAD downloads and installs so fast that many newbees, like me, try to download it multiple times thinking it's failed! Rest assured it's just THAT fast! So, feel free to save the download and install later, but don't let time constraints be the reason why! I prefer to get it done, so I choose Run!

Figure 1.37

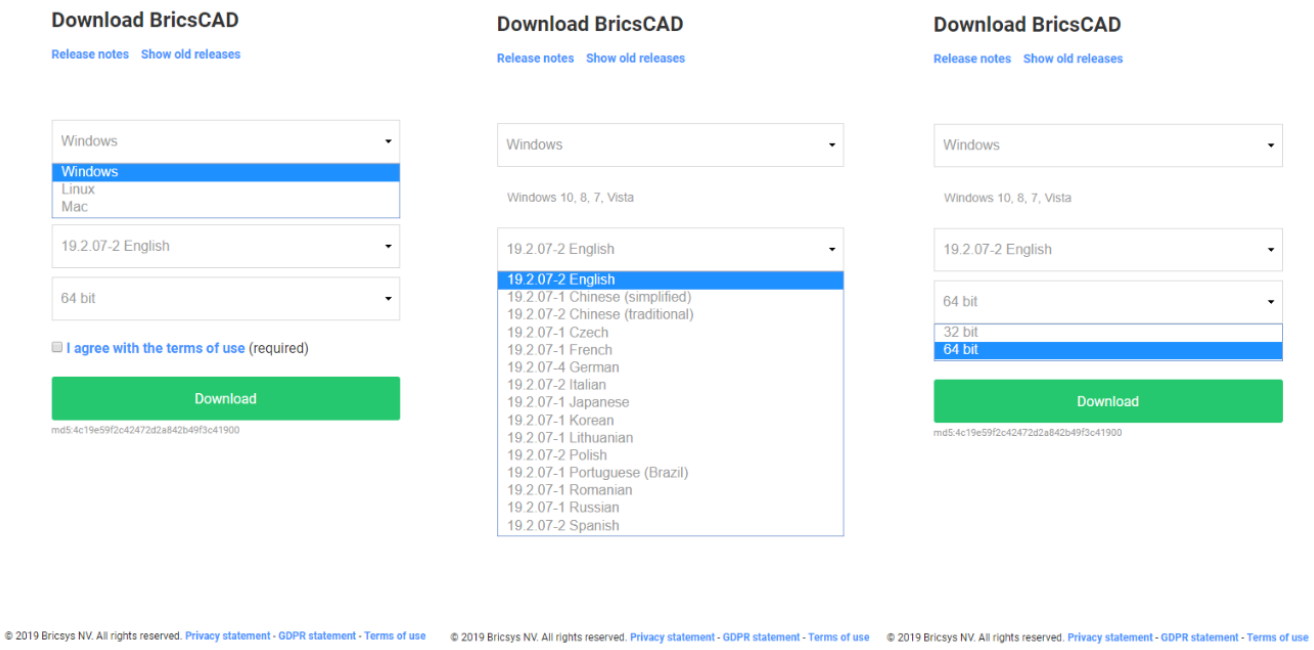
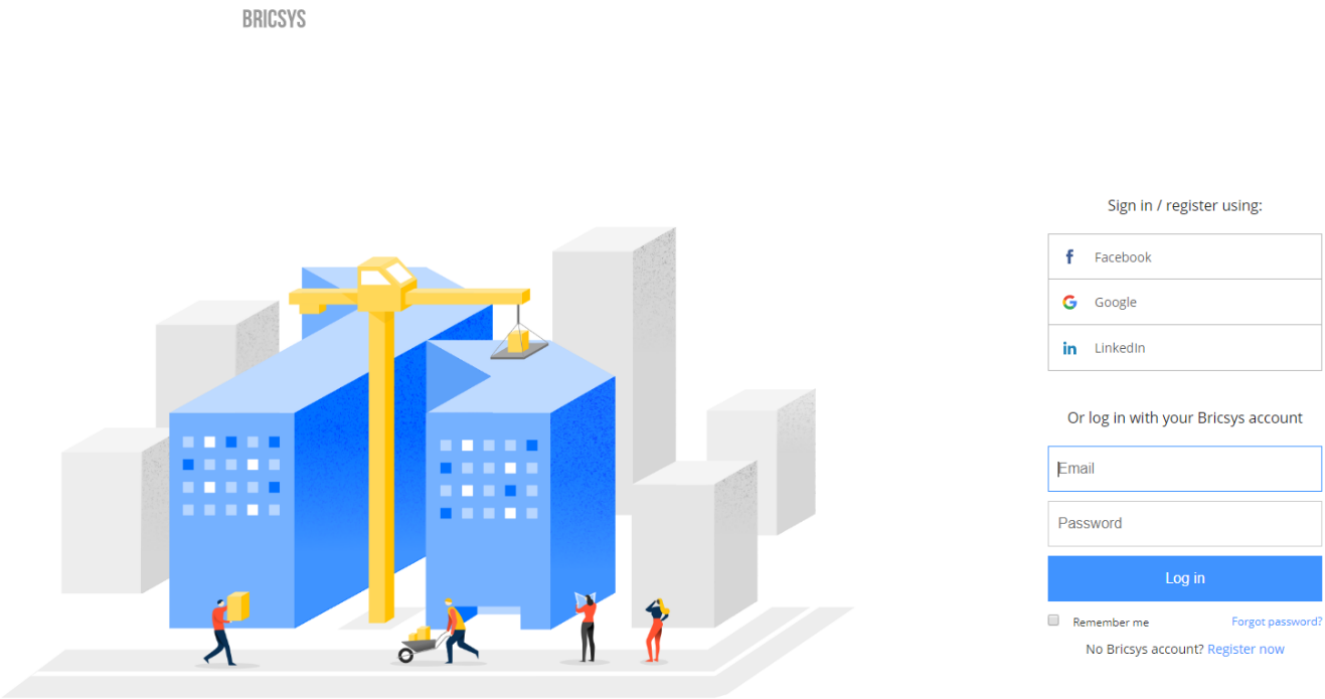


Figure 1.38



Install BricsCAD Software

On my system, with a wireless connection, it took about 1 minute to download. When it finishes downloading, the installer prompts you to accept the license agreement and install location. You can also choose to add a shortcut to the desktop and automatically display release notes when the install completes. And for those of you considering a move from AutoCAD, notice the options for file associations. I expect those look familiar to you!
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/Install1.PNG})
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/Install2.PNG})
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/Install3.PNG})
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/Install4.PNG})
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/Install5.PNG})
If you're sitting right there ready to respond to those prompts, the install only takes about one minute. So, don't be tempted to take that coffee break just yet! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Insta
When BricsCAD finishes installing, you can choose to activate it using the 30 day trial or enter a valid license key.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/ActivateBricsCAD.PNG})
If you choose to activate the trial, each time you launch BricsCAD you're reminded how many days are remaining in your trial and you're offered another opportunity to enter a valid license key. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc

Figure 1.39

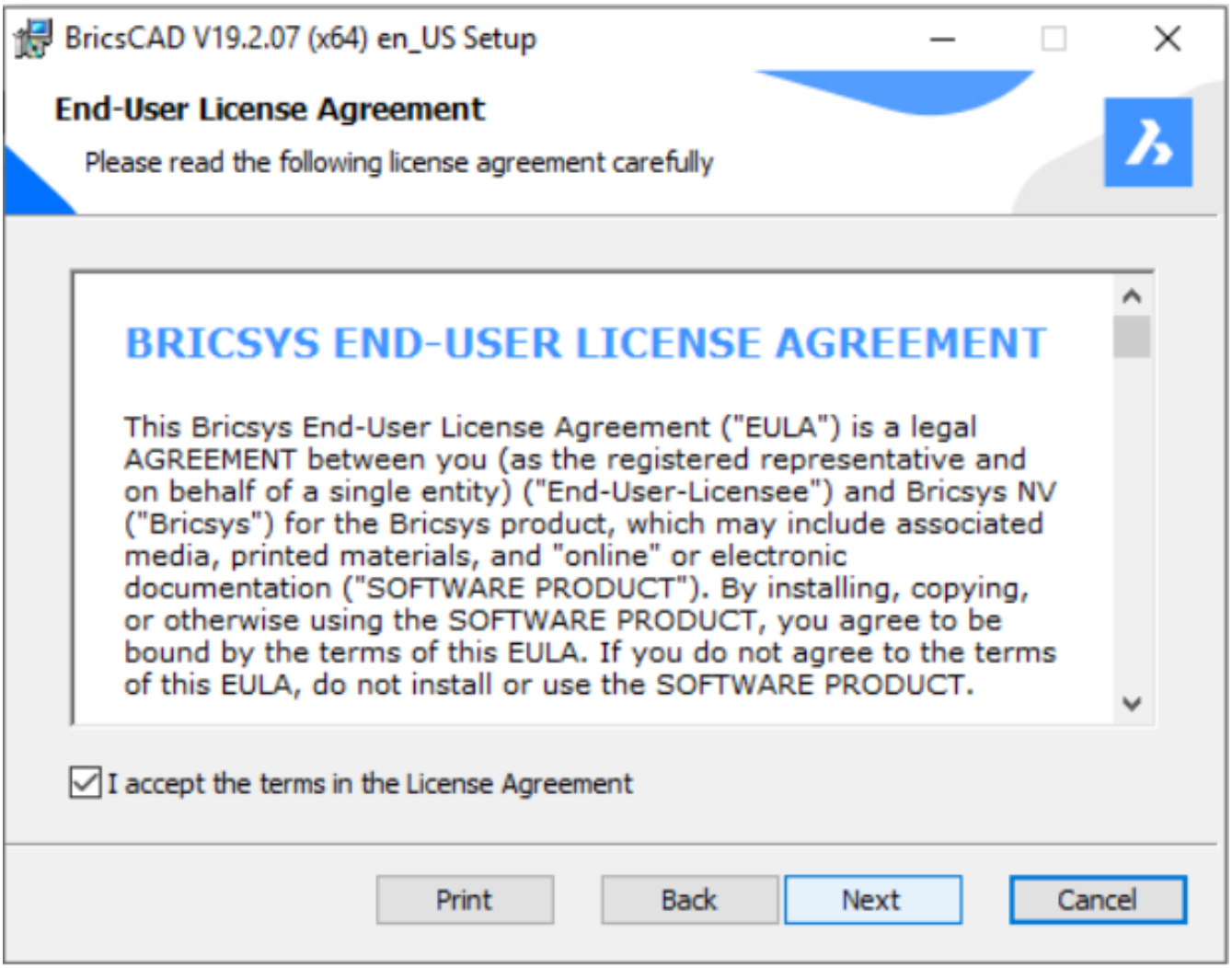

BricsCAD V19.2.07 (x64) en_US Setup
End-User License Agreement
Please read the following license agreement carefully
BRICSYS END-USER LICENSE AGREEMENT
This Bricsys End-User License Agreement ("EULA") is a legal AGREEMENT between you (as the registered representative and on behalf of a single entity) ("End-User-Licensee") and Bricsys NV ("Bricsys") for the Bricsys product, which may include associated media, printed materials, and "online" or electronic documentation ("SOFTWARE PRODUCT"). By installing, copying, or otherwise using the SOFTWARE PRODUCT, you agree to be bound by the terms of this EULA. If you do not agree to the terms of this EULA, do not install or use the SOFTWARE PRODUCT.
I accept the terms in the License Agreement
Print
Back
Next
Cancel

Figure 1.40

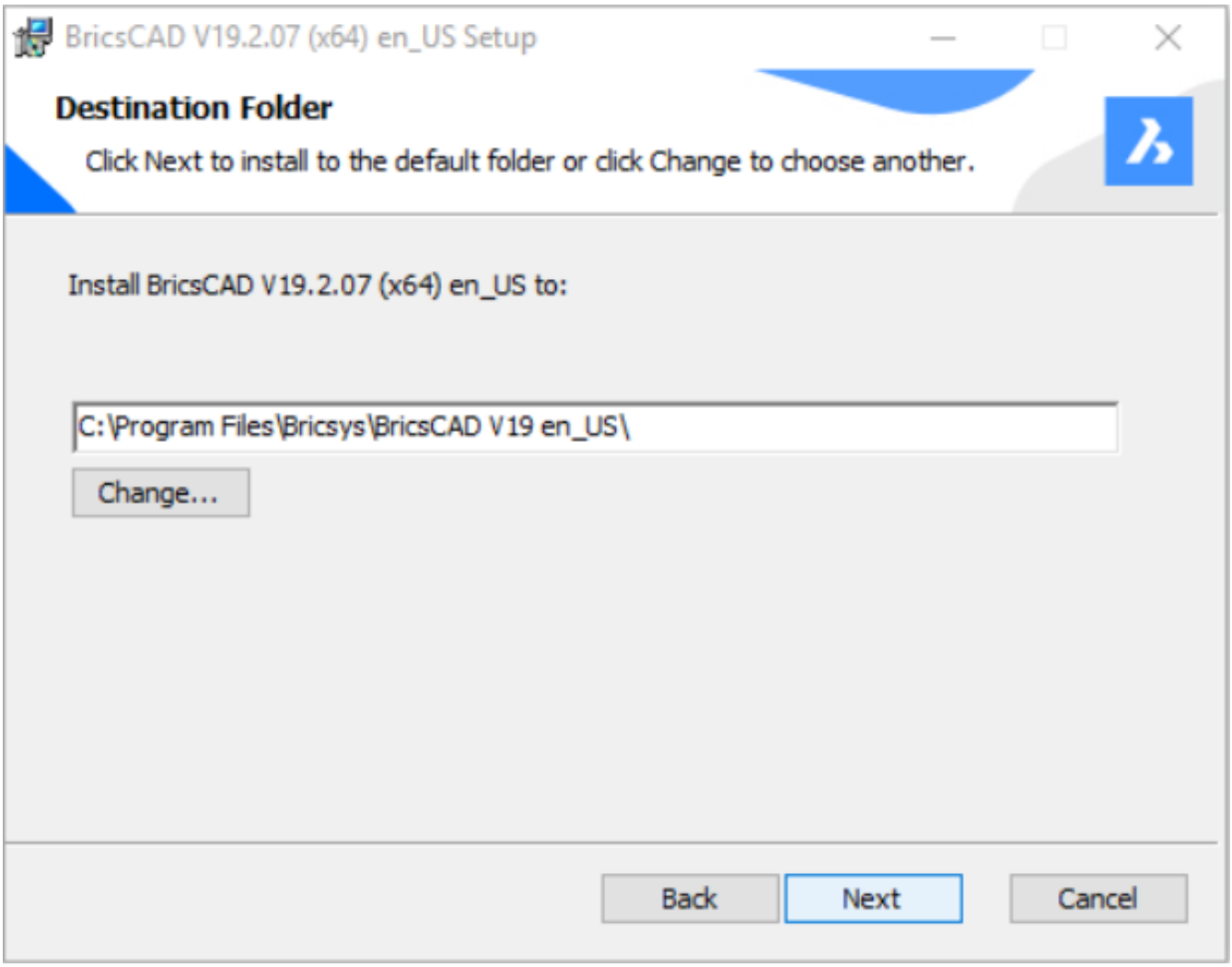

BricsCAD V19.2.07 (x64) en_US Setup
Destination Folder
Click Next to install to the default folder or click Change to choose another.
Install BricsCAD V19.2.07 (x64) en_US to:
C:\Program Files\Bricsys\BricsCAD V19 en_US\
Change...
Back
Next
Cancel

Figure 1.41

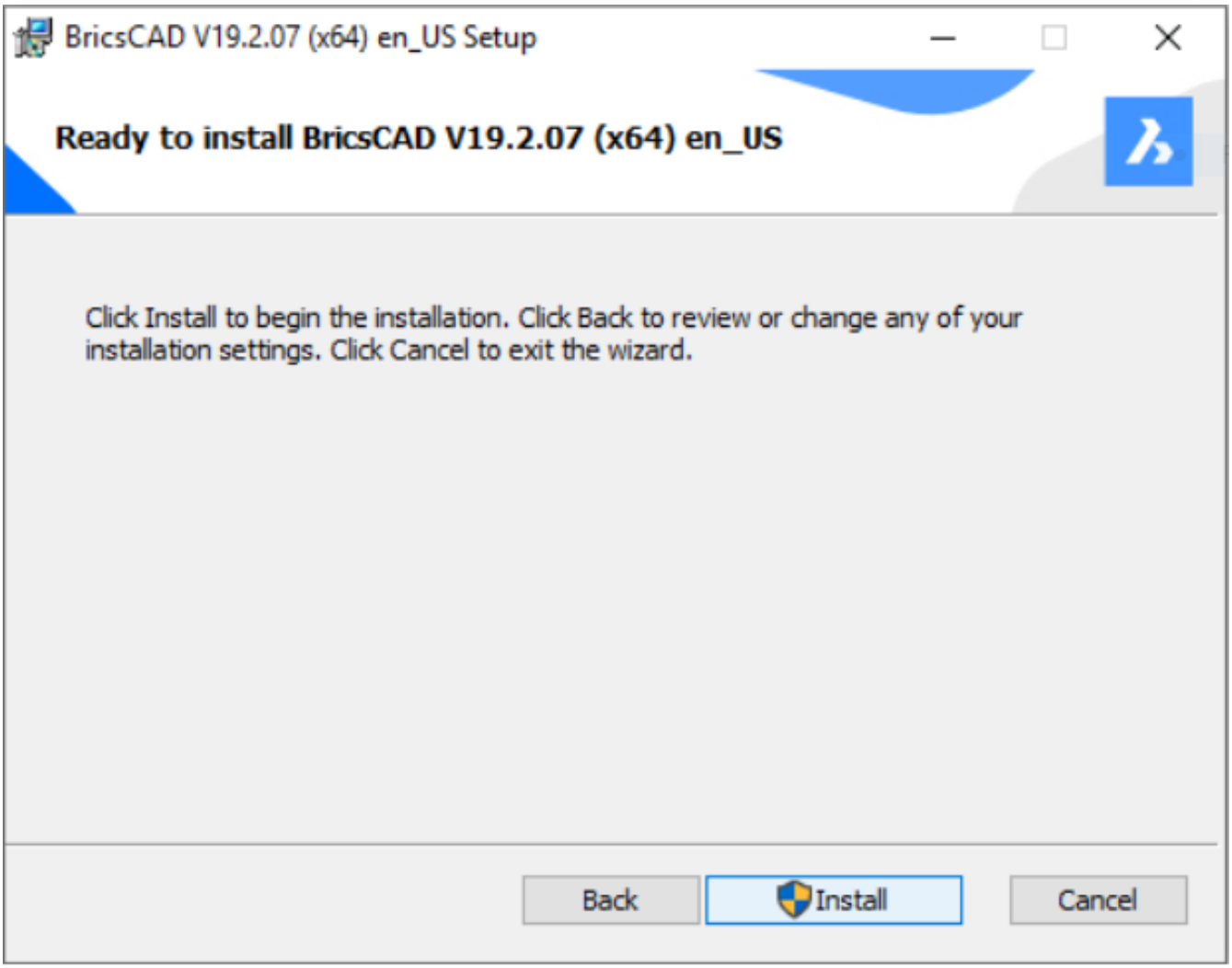

BricsCAD V19.2.07 (x64) en_US Setup
Ready to install BricsCAD V19.2.07 (x64) en_US
Click Install to begin the installation. Click Back to review or change any of your installation settings. Click Cancel to exit the wizard.
Back
Install
Cancel

Figure 1.42

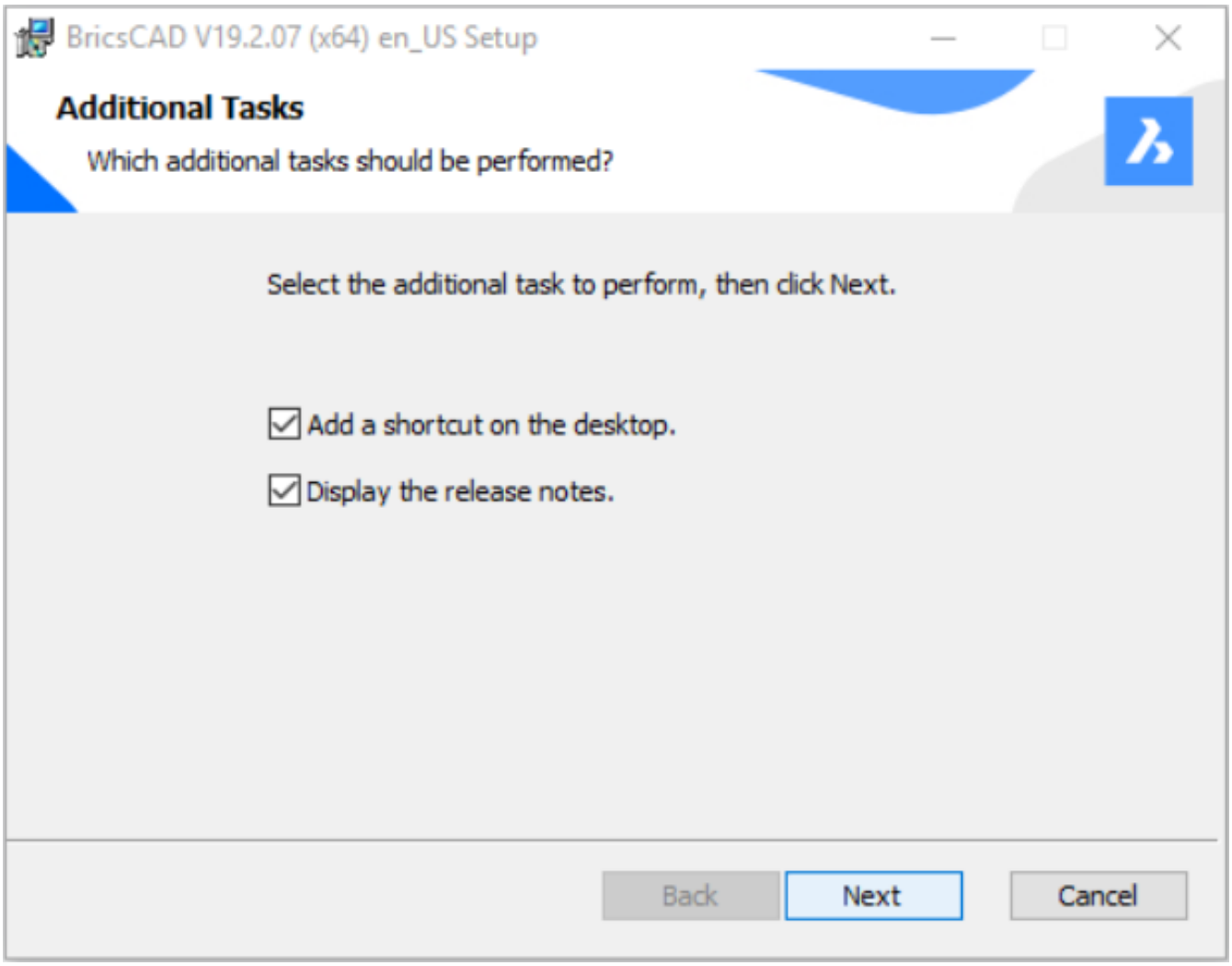

BricsCAD V19.2.07 (x64) en_US Setup
Additional Tasks
Which additional tasks should be performed?
Select the additional task to perform, then click Next.
Add a shortcut on the desktop.
Display the release notes.
Back
Next
Cancel

Figure 1.43

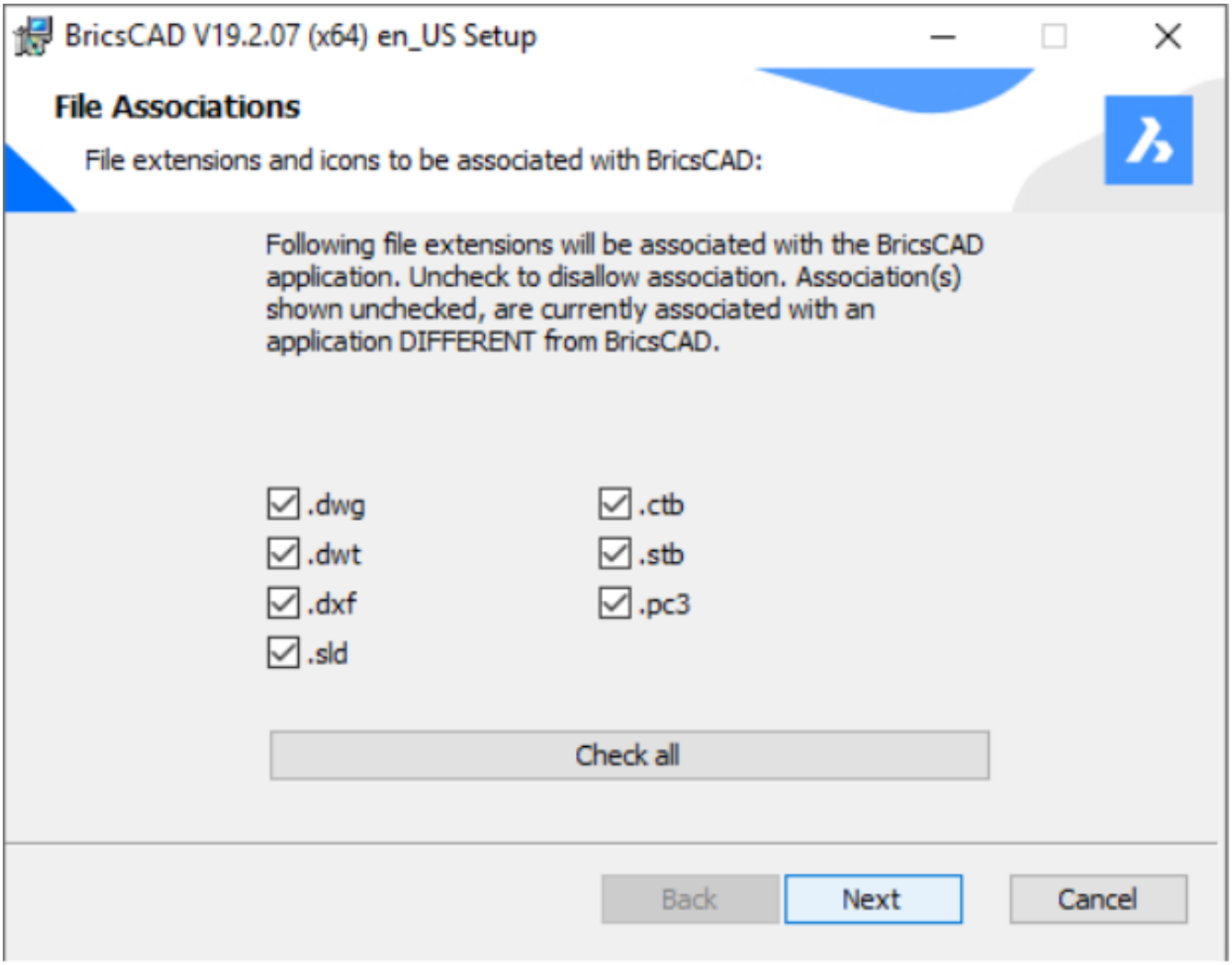

BricsCAD V19.2.07 (x64) en_US Setup
File Associations
File extensions and icons to be associated with BricsCAD:
Following file extensions will be associated with the BricsCAD application. Uncheck to disallow association. Association(s) shown unchecked, are currently associated with an application DIFFERENT from BricsCAD.
.dwg
.dwt
.dxf
.sld
.ctb
.stb
.pc3
Check all
Back
Next
Cancel

Figure 1.44

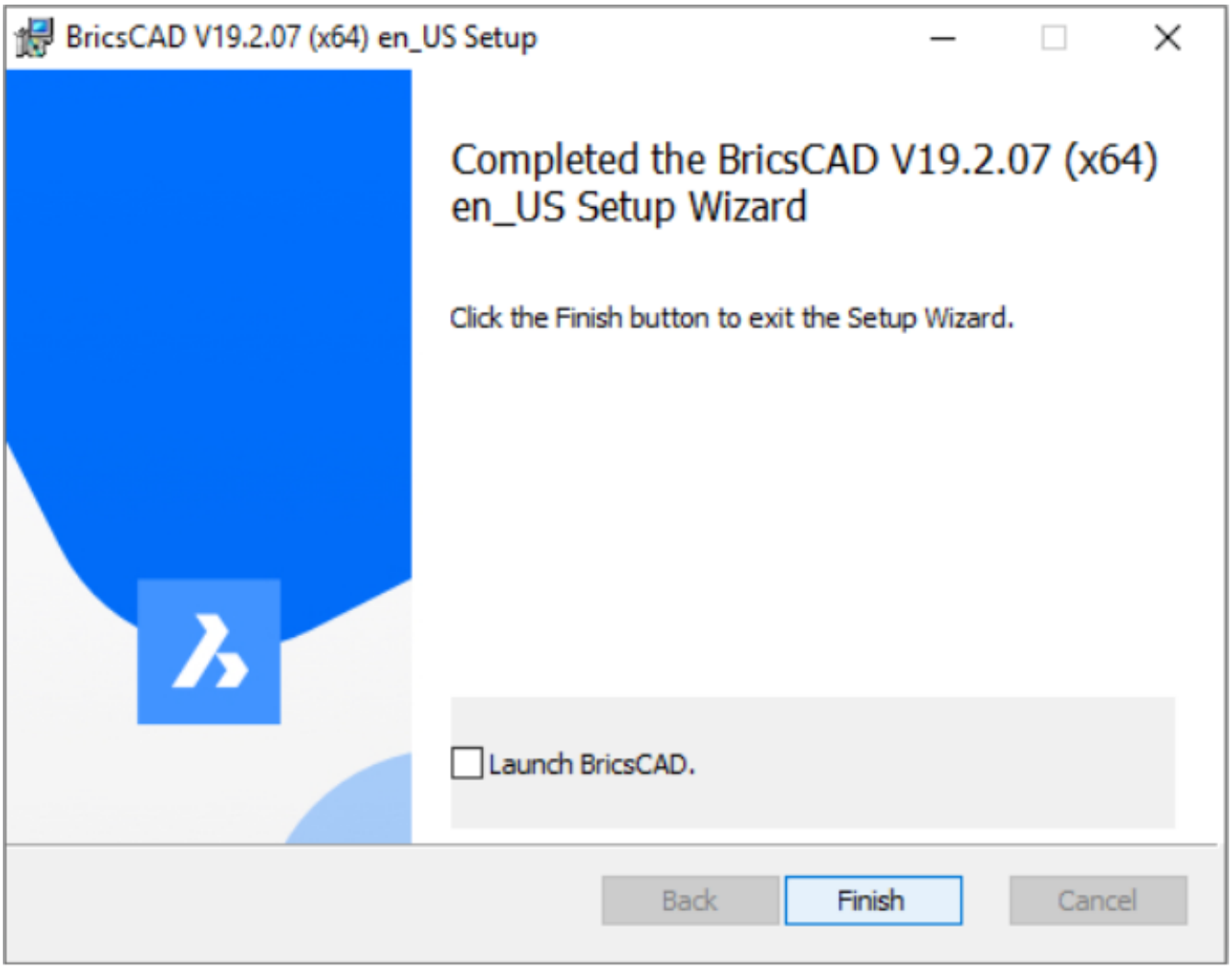


Figure 1.45

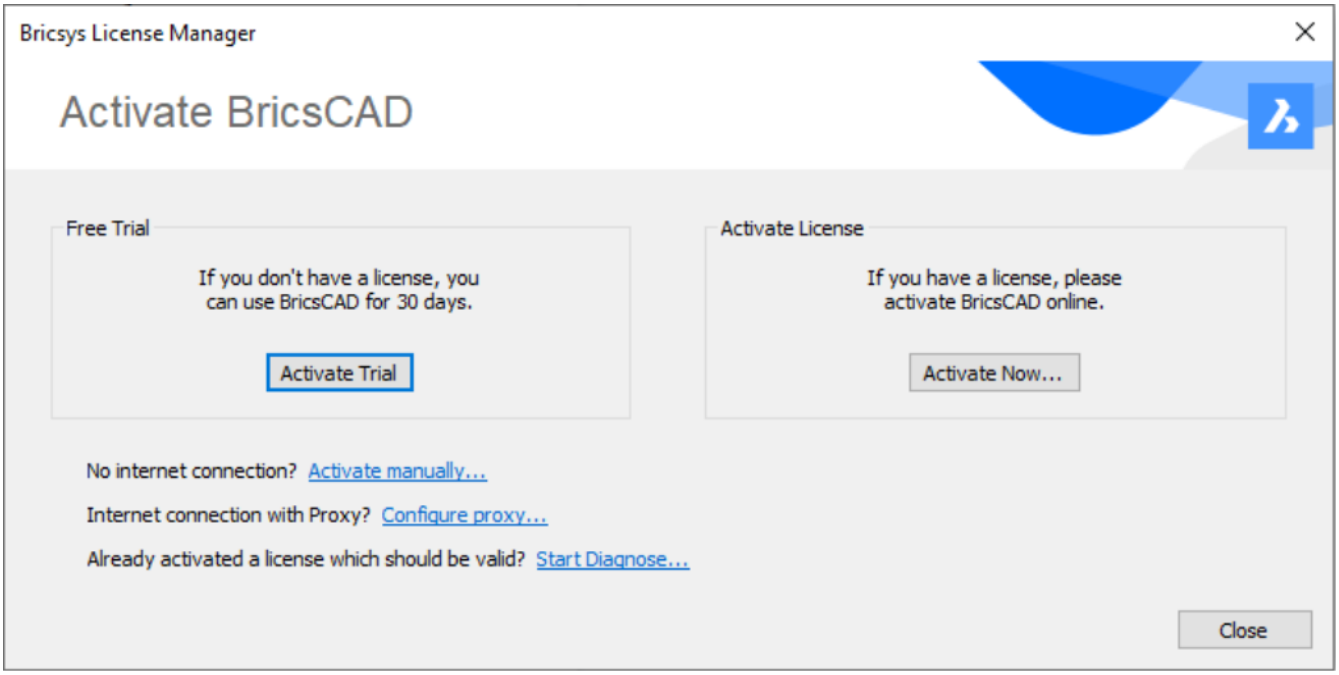
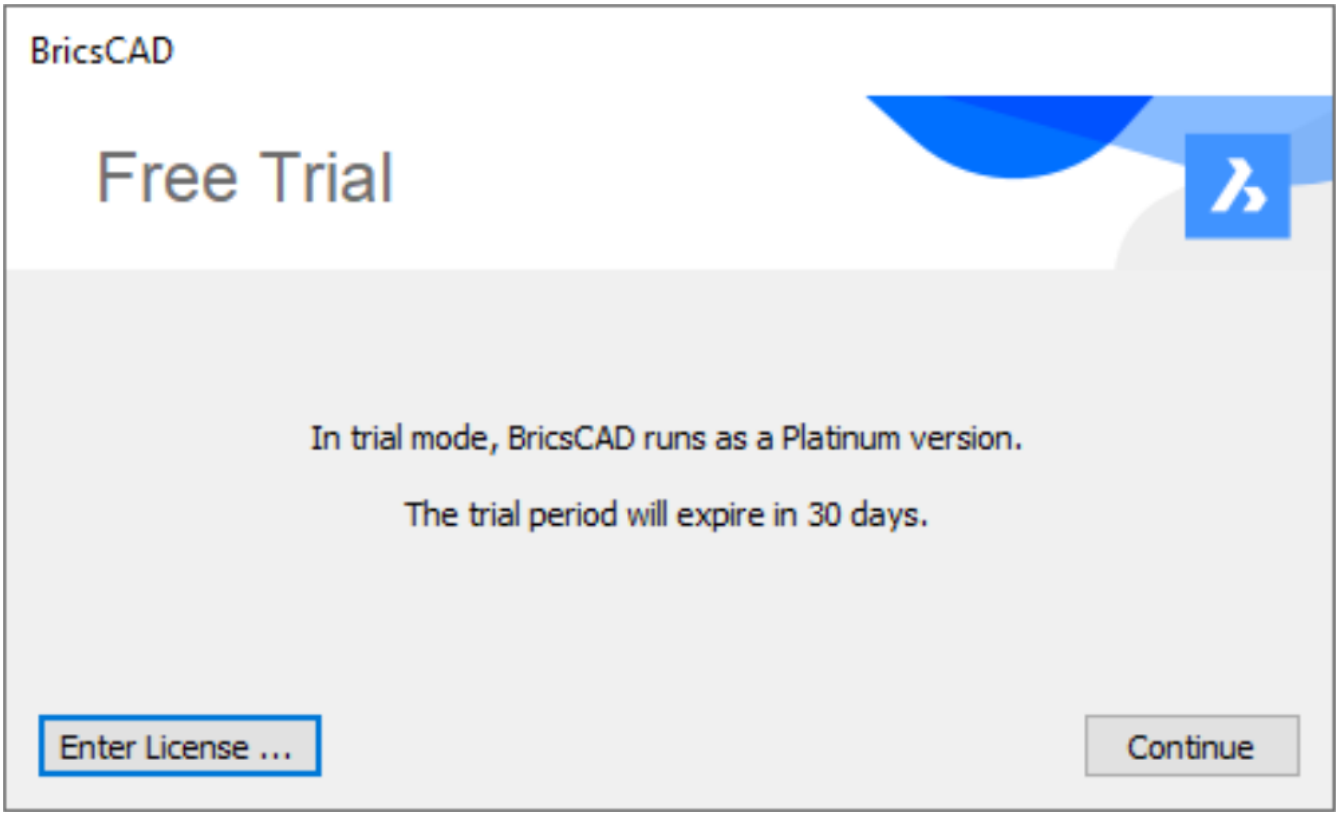


Figure 1.46

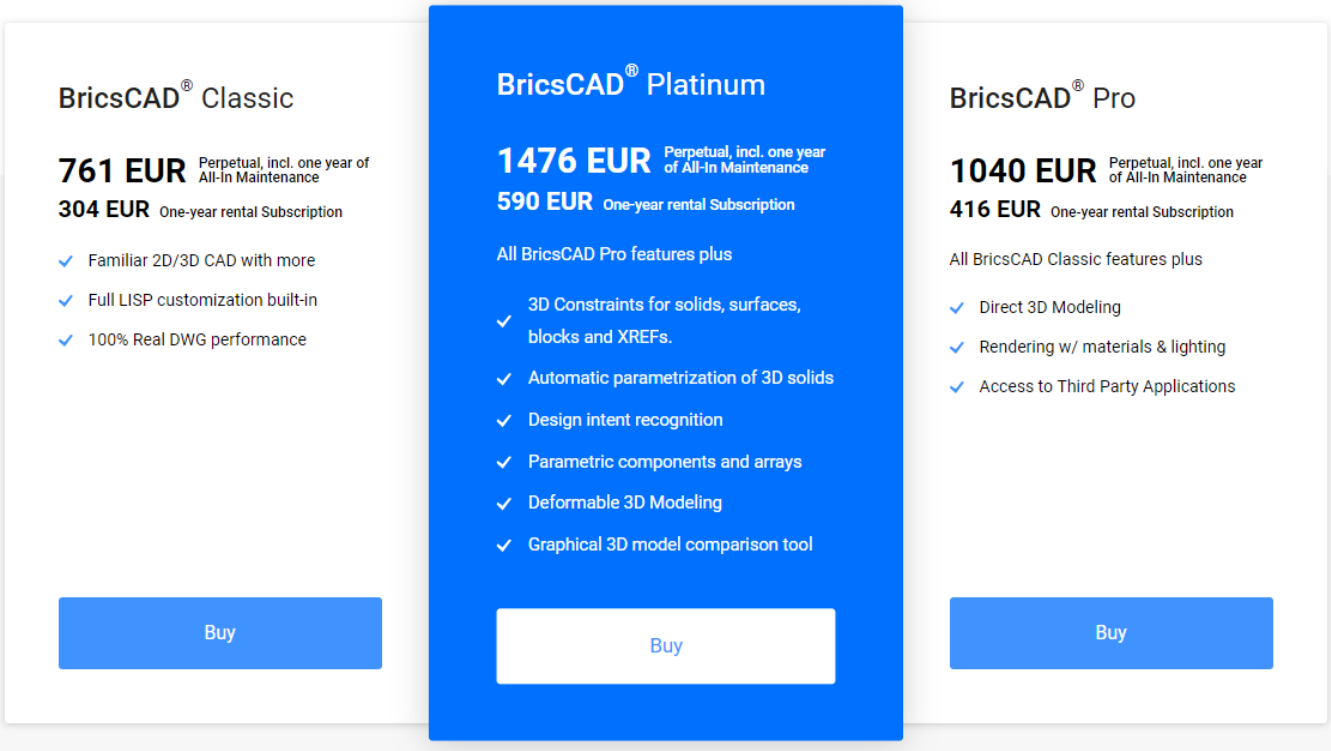

BricsCAD
Free Trial
In trial mode, BricsCAD runs as a Platinum version.
The trial period will expire in 30 days.
Enter License ...
Continue

1.10.5 Exploring Trial Levels

BricsCAD Editions

You can purchase BricsCAD available in three different editions: Classic, Pro, and Platinum. The BricsCAD Classic edition is for2D drafting and is comparable to AutoCAD LT®. However, it also includes full LISP support! The Pro edition adds 3D modeling, visualization tools, and access to third-party applications. It's comparable to AutoCAD®. BricsCAD Platinum adds 3D constraint creation, design intent recognition, assembly modeling, deformable modeling and 3D Compare. I don't know what to compare that to! I guess that makes it incomparable! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/TrialLevels.PNG}) Visit https://www.bricsys.com/bricscad/ to learn more about the BricsCAD purchase options.

Figure 1.47



BricsCAD Modules

While BricsCAD Pro and Platinum offer significant CAD functionality to typical designers, you may want more. Therefore, Bricsys offers three optional modules that are built on top of BricsCAD: BIM, Mechanical, and Communicator. You can add any or all of these modules to BricsCAD Platinum. The BIM module enables you to create full-featured building information models. The Mechanical module enables product design and manufacturing for assembly modelling. It also enables you to import, create, unfold, rework and export sheet metal parts. The Communicator module allows you to import and export 3D CAD data from popular manufacturing file formats including CATIA, Inventor, and SolidWorks. Unlike BIM and Mechanical, the Communicator module is not installed with the Trial download of BricsCAD. However, you can download and install a trial version of Communicator with either the BricsCAD Pro or Platinum editions.

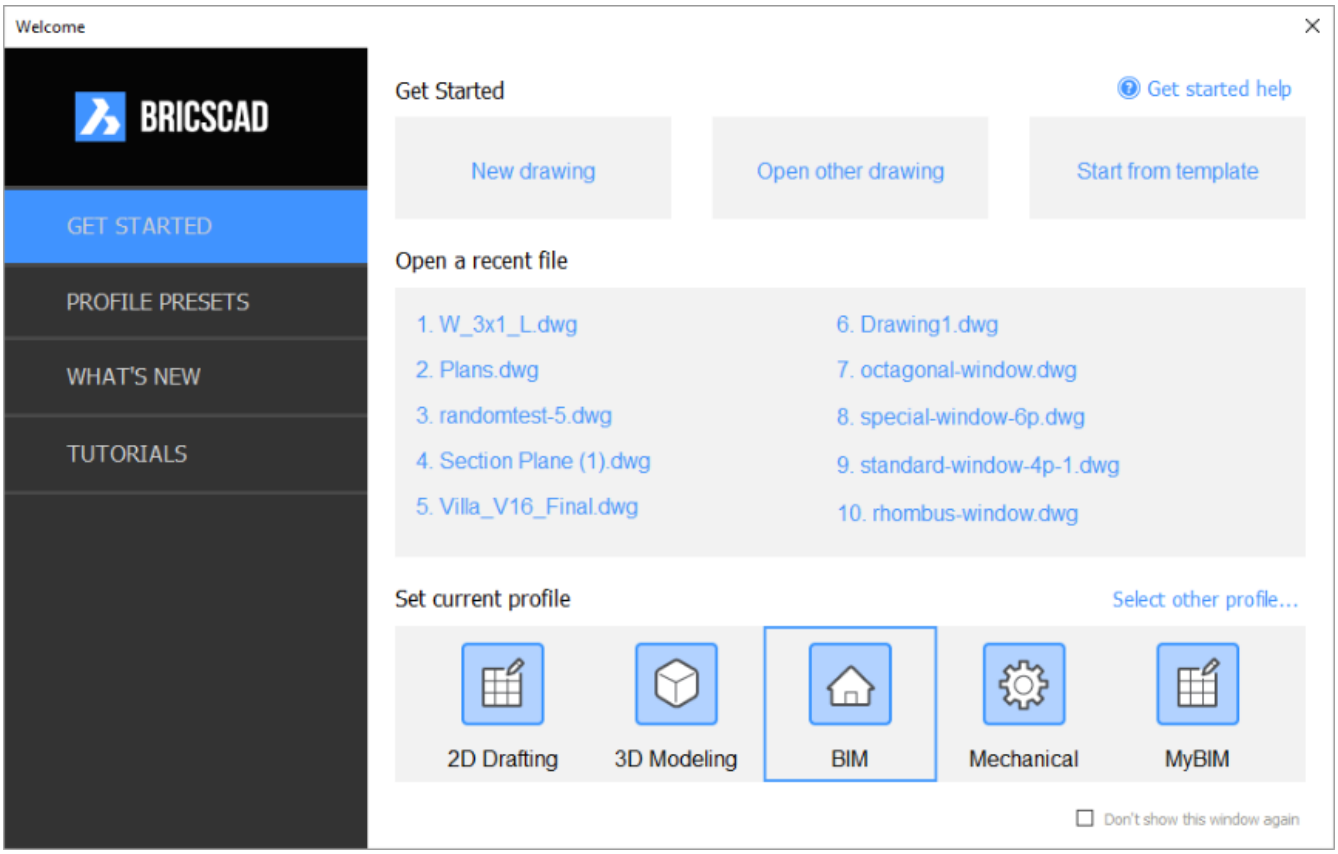
Figure 1.48



Trial Levels

When you install BricsCAD, it automatically installs the Platinum edition. If you access it using a trial license, you can also try the BIM andMechanical modules. You may wish to test the differences between editions and modules to determine which combination is right for you. Let me introduce you to the RunAsLevel (6.26.46) command! While in trial mode you can use the RunAsLevel command to change the behavior of BricsCAD. First, choose the BricsCAD option and then enter which edition you want to try: Classic (C), Pro (P) or Platinum (PL). You'll have to close and restart BricsCAD to see the changes. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingSt
Changing the BricsCAD level allows you to experience each of the editions during your trial period! Similarly, use the RunAsLevel command to enable or disable BIM and Mechanical tools. You can try as much, or as little, BricsCAD functionality as you want without having to install or uninstall additional software!

Figure 1.49

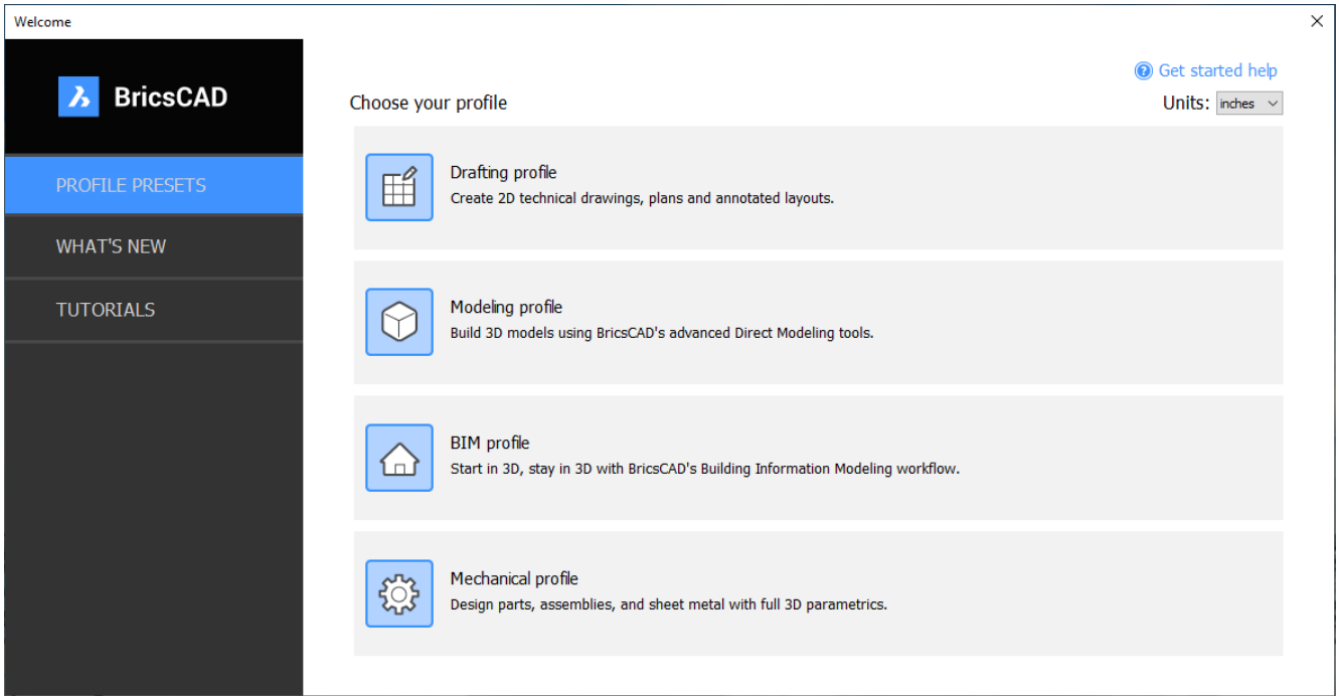


1.10.6 Get Started dialog

If the GetStarted system variable is **ON**, the Welcome dialog box displays when you launch BricsCAD. Here you can choose the user profile (1.13.9) for the current session. You can choose between a number of presets or user defined profiles. The most recently used profile is indicated by a blue frame. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

Note: The very first time BricsCAD is launched the Get Started option is missing on the Welcome dialog. The Profile Presets page is shown instead. After selecting a profile preset the Get Started dialog displays with the new profile selected in the Set current profile list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingSt

Figure 1.50



To start a BricsCAD session

1. Accept the currently selected profile or select a profile in the **Set current profile** list.
2. Do one of the following:

- Click the **New drawing** button to create a new drawing using the default template (1.14.1) of the current profile.
- Click the **Start from template** button to create a new drawing using a template of your choice (1.14.1).
- Click the **Open other drawing** button to open an existing drawing (1.14.6).
- Click a drawing in the **Open a recent file** list. The number of files in this list is controlled through the **Recent file list** (NFILELIST (1.13.11)) system variable. Up to 20 files can be listed here.

To select the startup user profile

Do one of the following:

- Click a profile in the **Set current profile** list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD
By default the five most recently used profiles display. If necessary, drag the right hand edge of the Welcome dialog box to see more profiles.
- Click **Select other profile**, then select a profile in the **All profiles** dialog and click the **OK** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/SelectOtherProfile.PNG})
The selected profile is added to the **Set current profile** list.

Figure 1.51

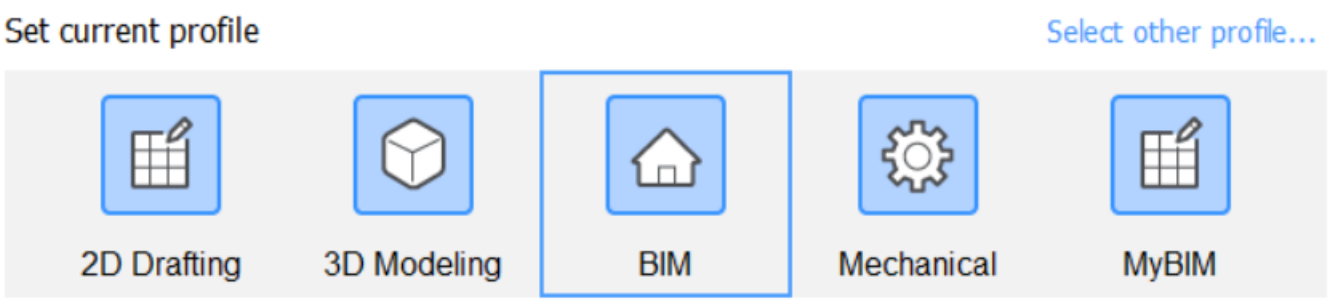
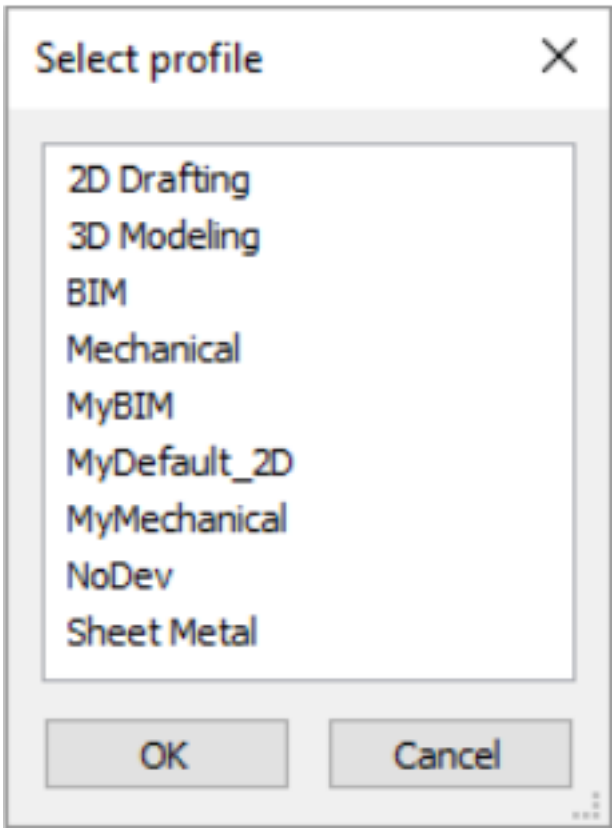


Figure 1.52



Using a Profile Preset

BricsCAD comes with a set of built-in hard coded profiles. You can use these Preset Profiles as templates to create your own set of user profiles. Each of the profile presets has the name of the Workspace (1.13.10)they use.

1. Click **PROFILE PRESETS** on the **Welcome** dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
2. Choose either **Imperial** or **Metric** in the Units list button to set the Measurement system variable in the profile.
3. Select a **Profile Preset** , e.g. **2D Drafting** . A dialog box displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h
4. Type a new name in the **Profile name** field to create a new profile using the hard-coded settings of the selected Profile Preset. If you type the name of an existing profile a dialog box prompts you to confirm to overwrite the existing profile using the hard-coded settings of the selected Profile Preset. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/NameExists.PNG})

The **Get Started** dialog displays with the new profile selected in the **Set current profile** list.

Figure 1.53

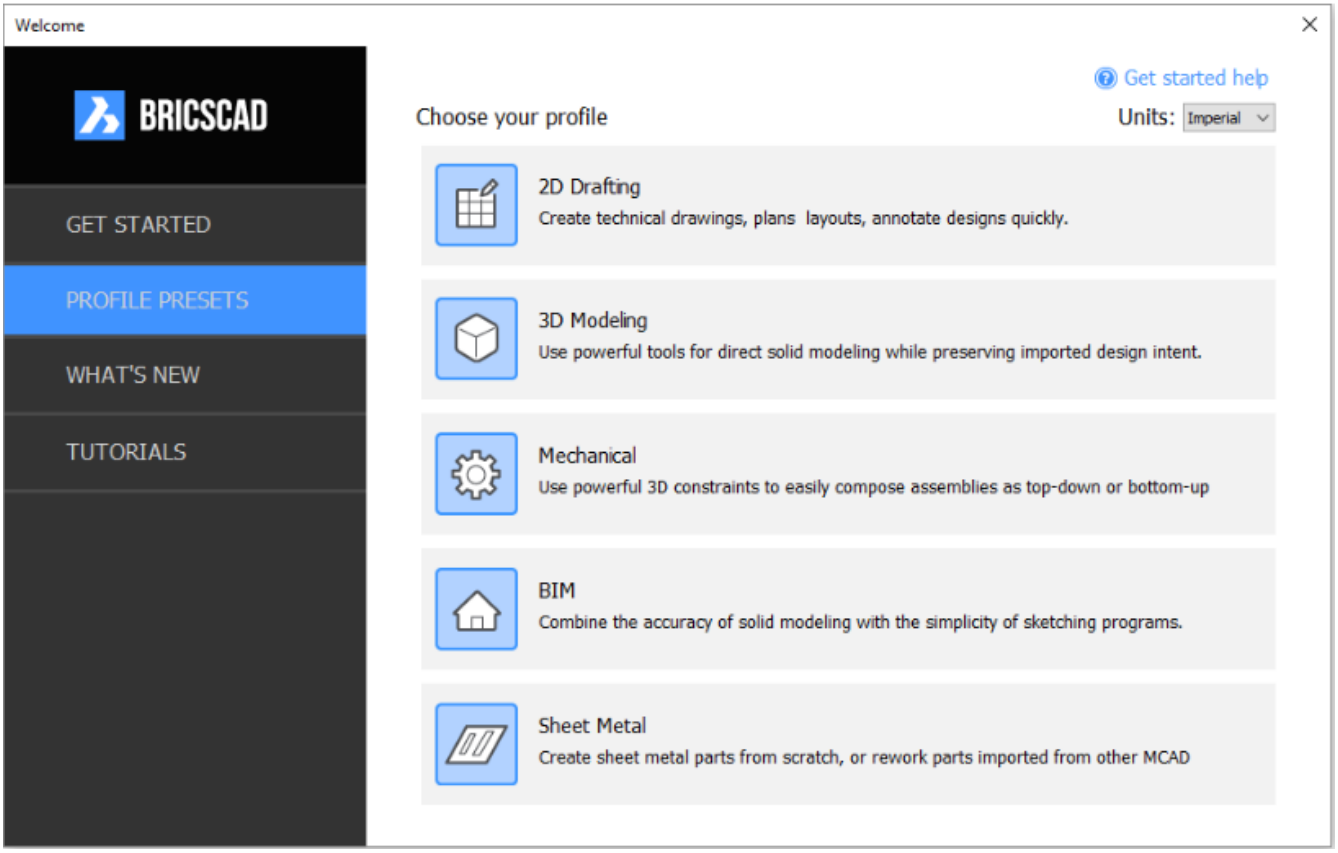


Figure 1.54

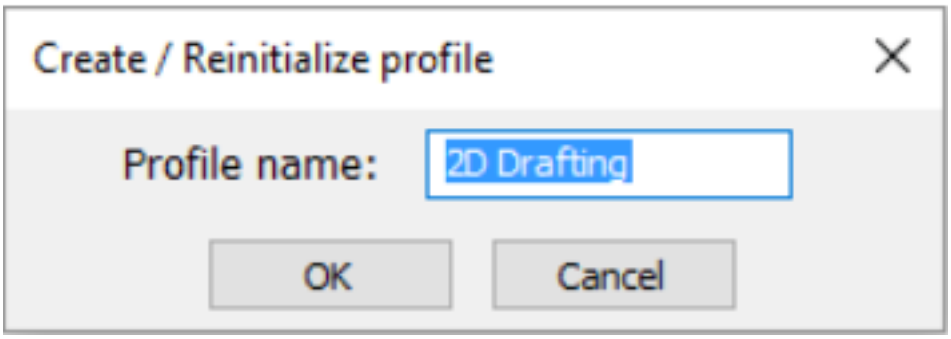


Figure 1.55

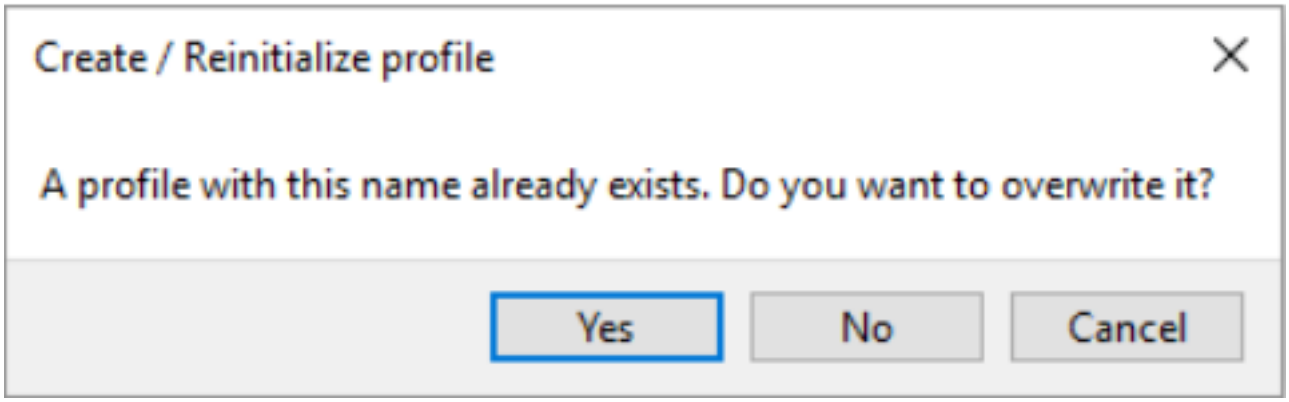
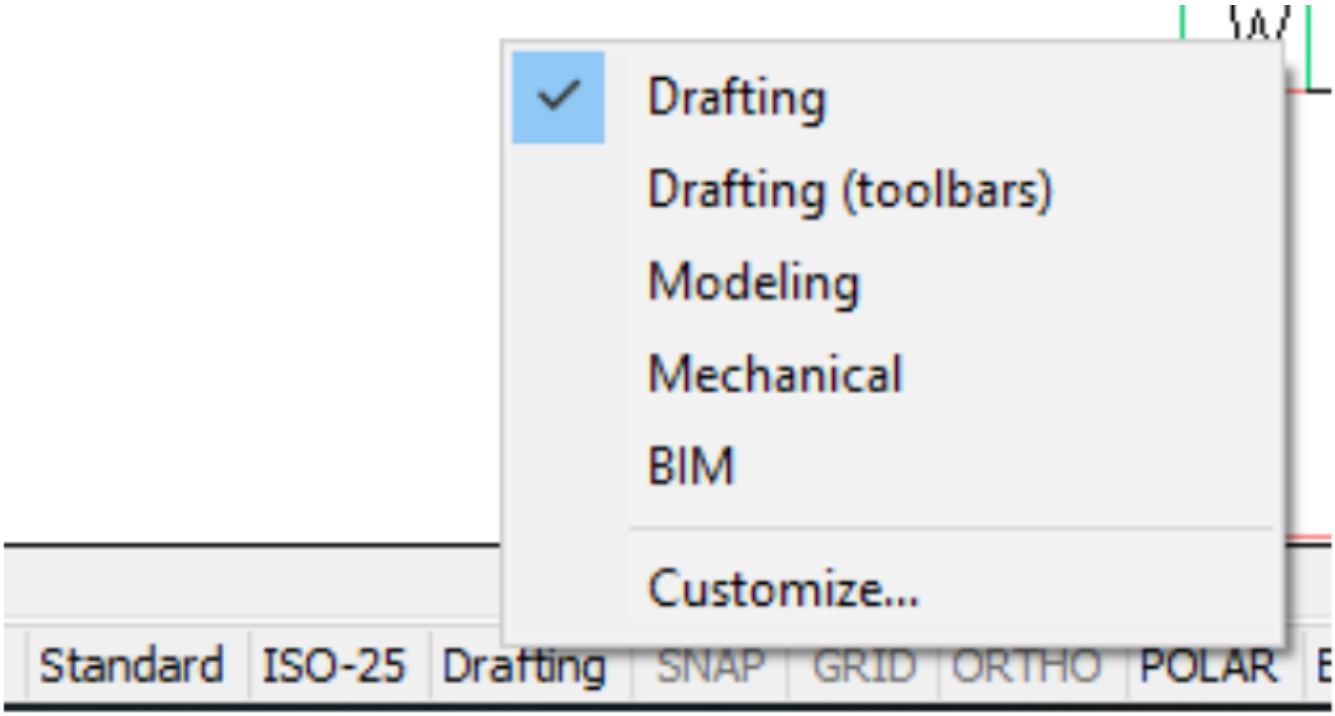


Figure 1.56



1.10.7 Exploring each Workspace

The profile you select determines which workspace is active when you launch BricsCAD. Regardless of which profile you select, you can easily change the current workspace. Simply right-click on the **Workspace** control in the status bar. Now, let's take a closer look at each of the pre-defined workspaces. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/PredefinedWorkspaces.PNG})

Drafting Workspaces

In the Drafting and Drafting (toolbars) workspaces, you have easy access to the most common tools for drafting and annotation. These tools are available in all the BricsCAD editions from Classic to Platinum. If you're exploring BricsCAD as a former AutoCAD LT® user, the Drafting environment should feel familiar to you. However, the Drafting and Drafting (toolbars) workspaces in BricsCAD include valuable tools you won't find in AutoCAD LT®. I'll describe some of them in future posts. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/D
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/DraftingToolbars.PNG})

Figure 1.57

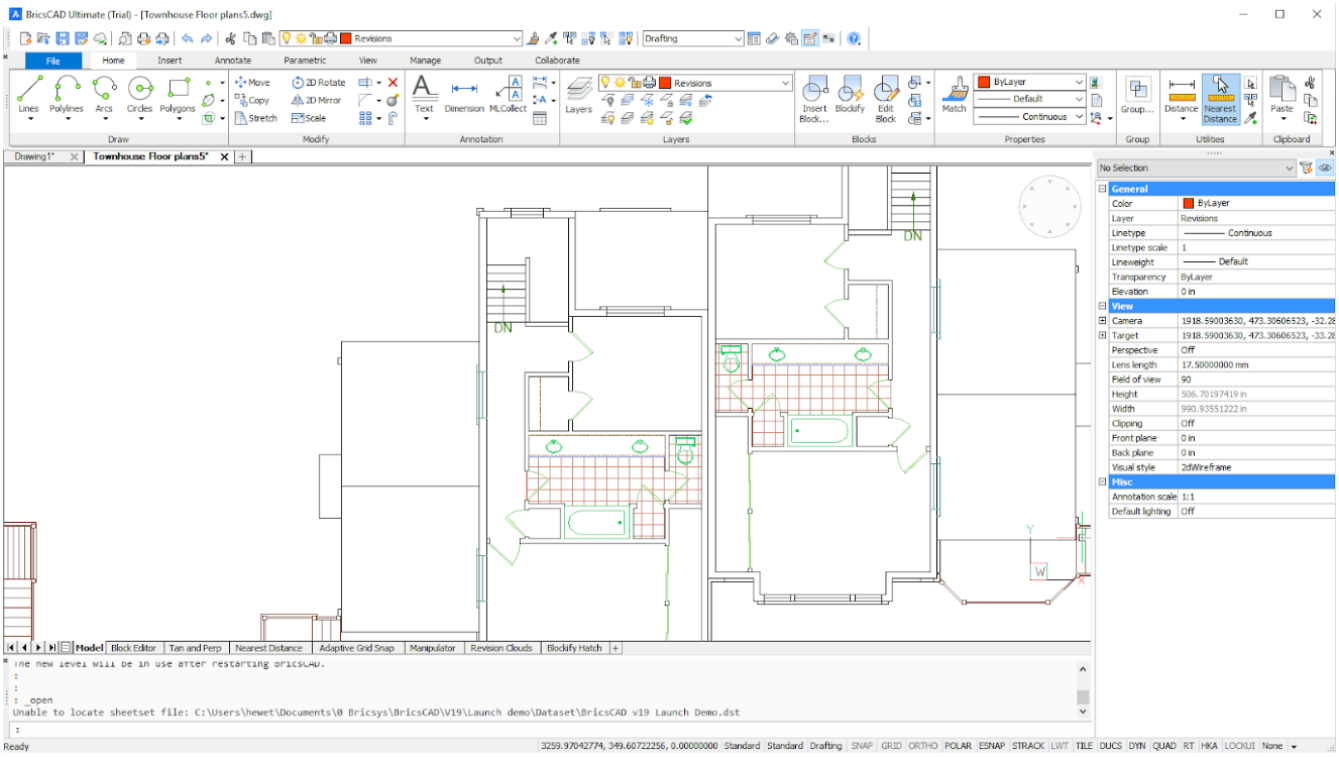
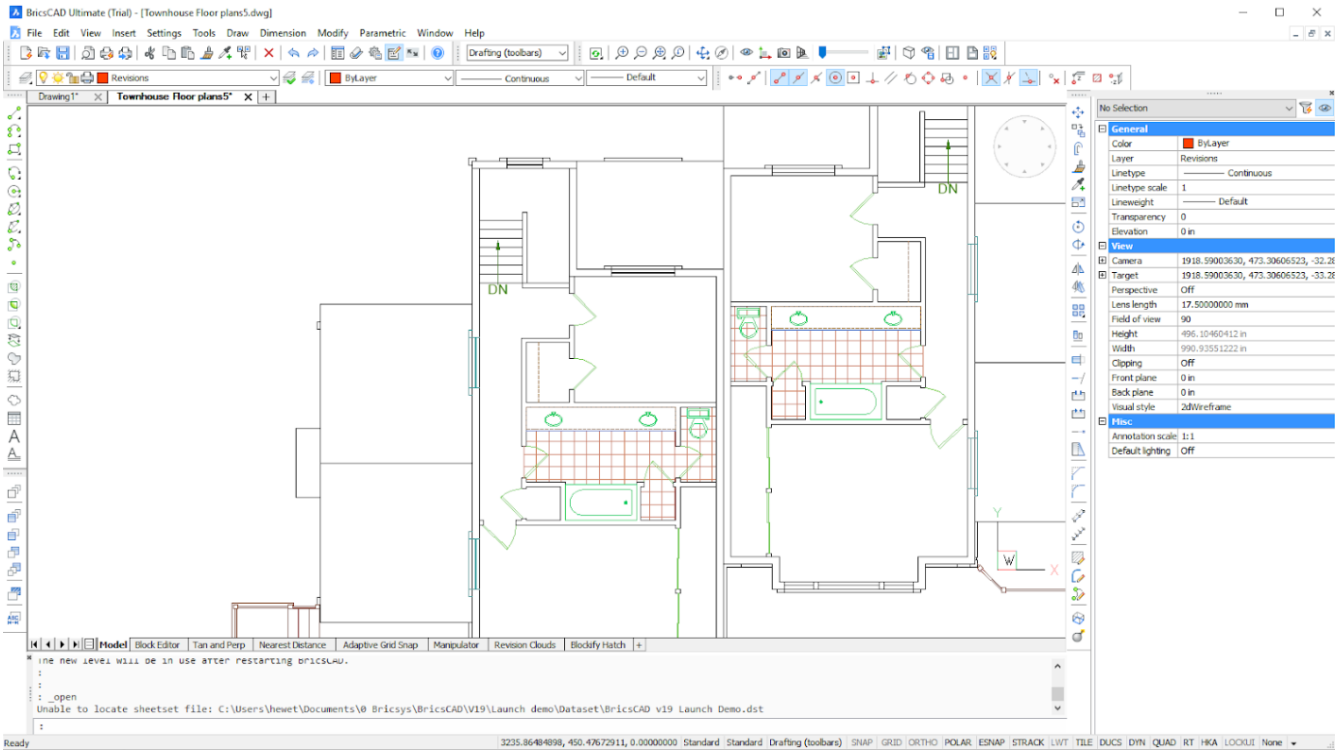


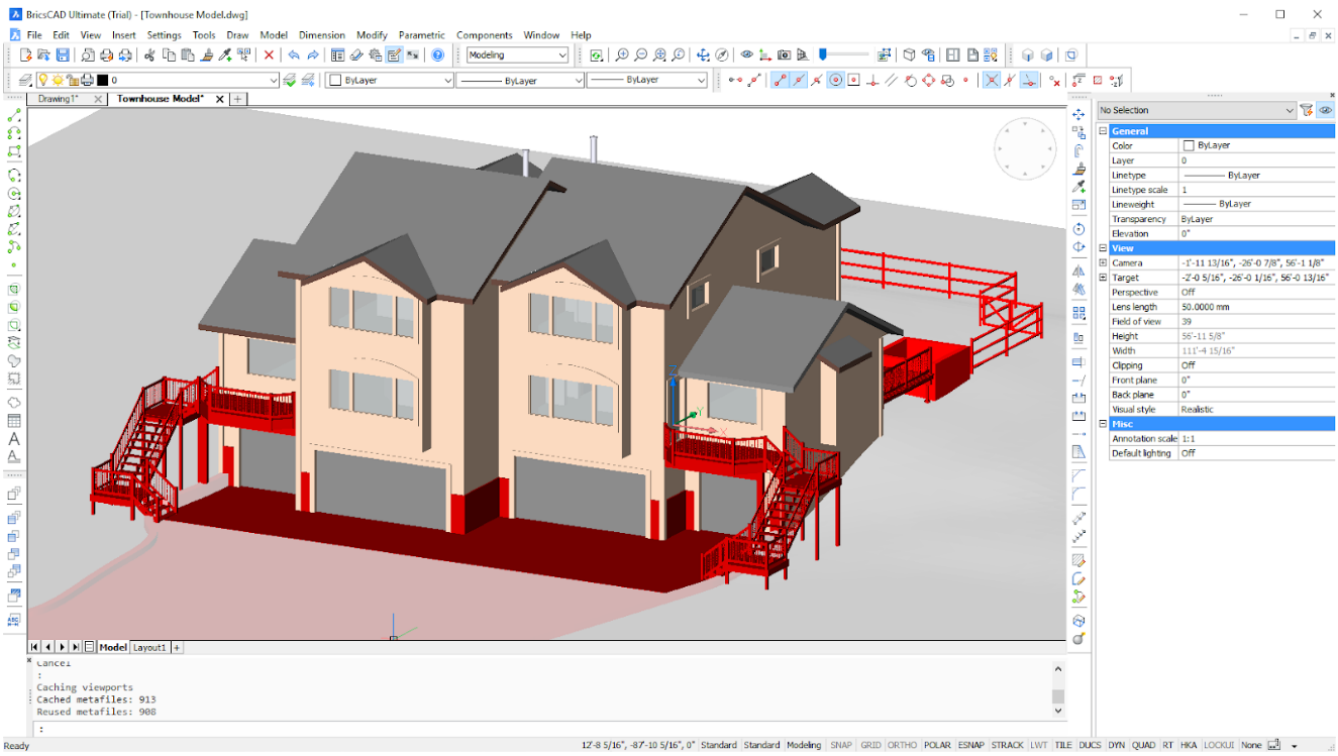
Figure 1.58



Modeling

The Modeling workspace offers easy access to tools for 3D modeling and visualization. These 3D tools are available in the Pro and Platinum editions. If you try to access them while your Trial level is set to Classic, you'll see most of the 3D tools are disabled. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/Mo

Figure 1.59



Mechanical

The Mechanical workspace is optimized for mechanical and sheet metal design. For mechanical design start by creating or importing a 3D solid. Then, easily modify your design using 3D direct modeling tools and constraints. Next, combine your parts with standard parts from the included libraries to create complex assemblies. The mechanical tools and libraries in the mechanical workspace are available in the BricsCAD Platinum edition. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/Mechanical.PNG}) For sheet metal design, create or import the desired shape of the solid using direct modeling operations. Then, convert them to sheet metal to prepare for production. Next, automatically unfold sheet metal parts with a single click. Rework your designs at any time without having to restart from scratch. Finally, export your design for CNC machining. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/SheetMetal.PNG})

Figure 1.60

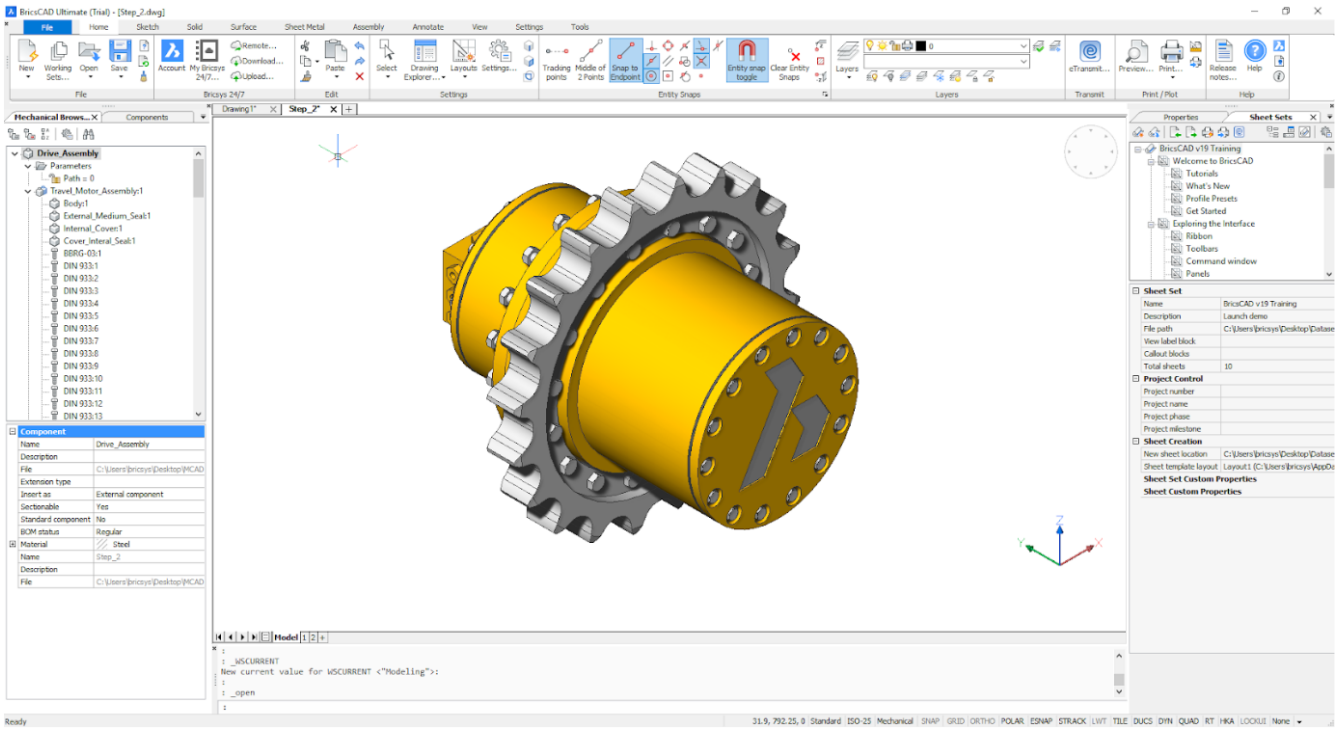
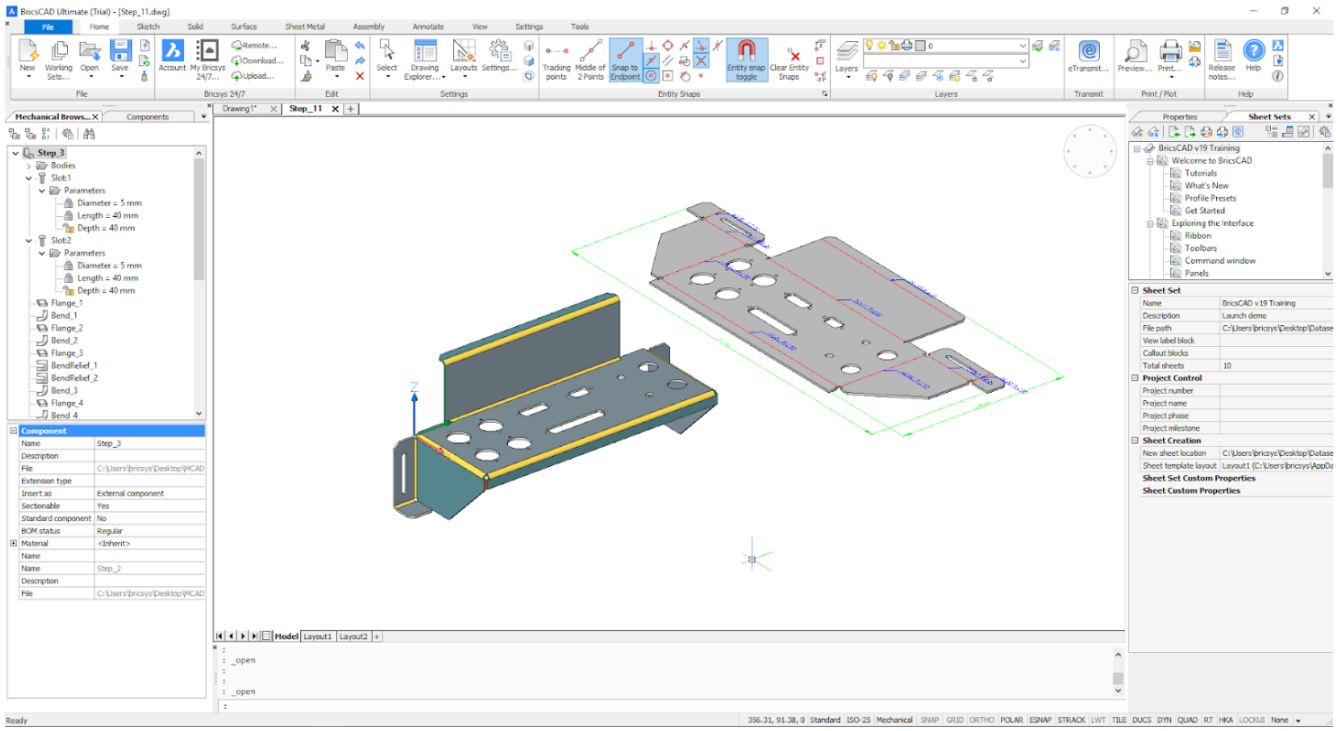


Figure 1.61



BIM

The BIM (Building Information Modeling) workspace includes tools and libraries to design your building from concept through documentation. First, use direct modeling functionality to imagine and explore your designs. Then, attach information such as materials and compositions to building elements. Finally, generate sheet sets with design documents directly from your building model. The BIM tools require BricsCAD Platinum edition with a BIM license (included in the 30-day trial). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/BI

Figure 1.62

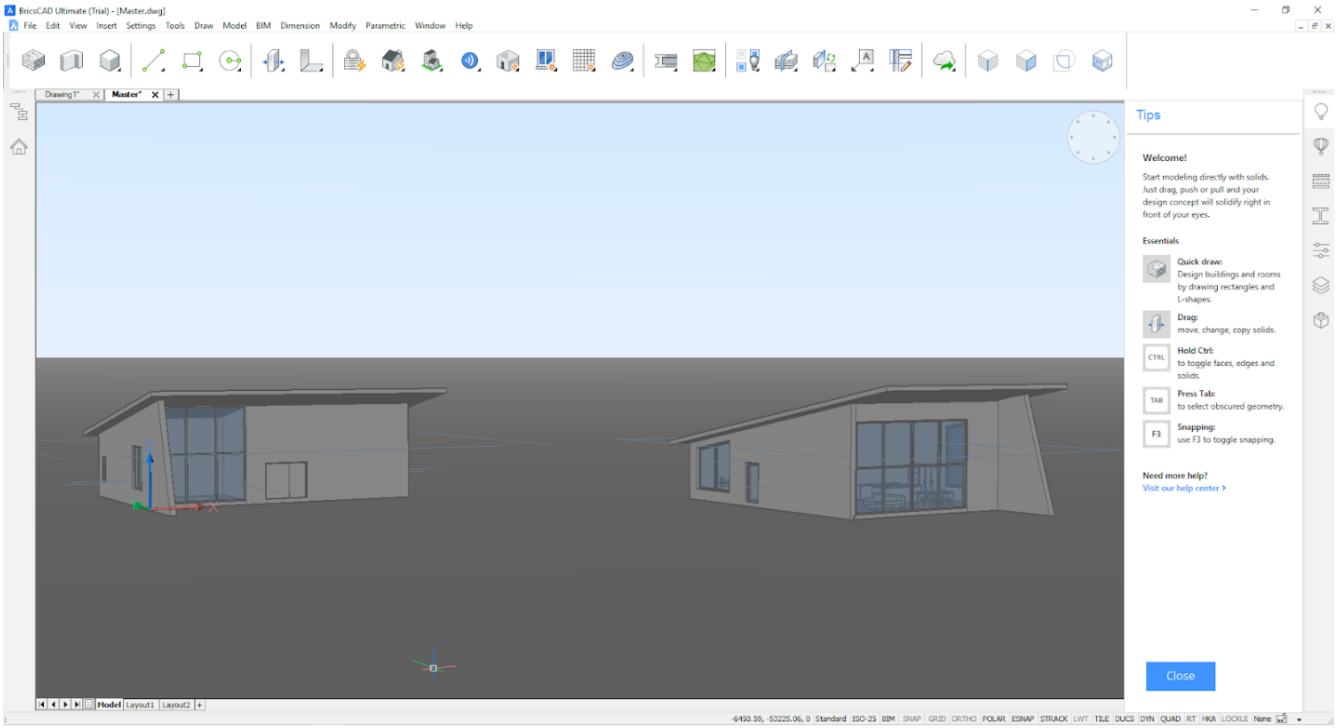
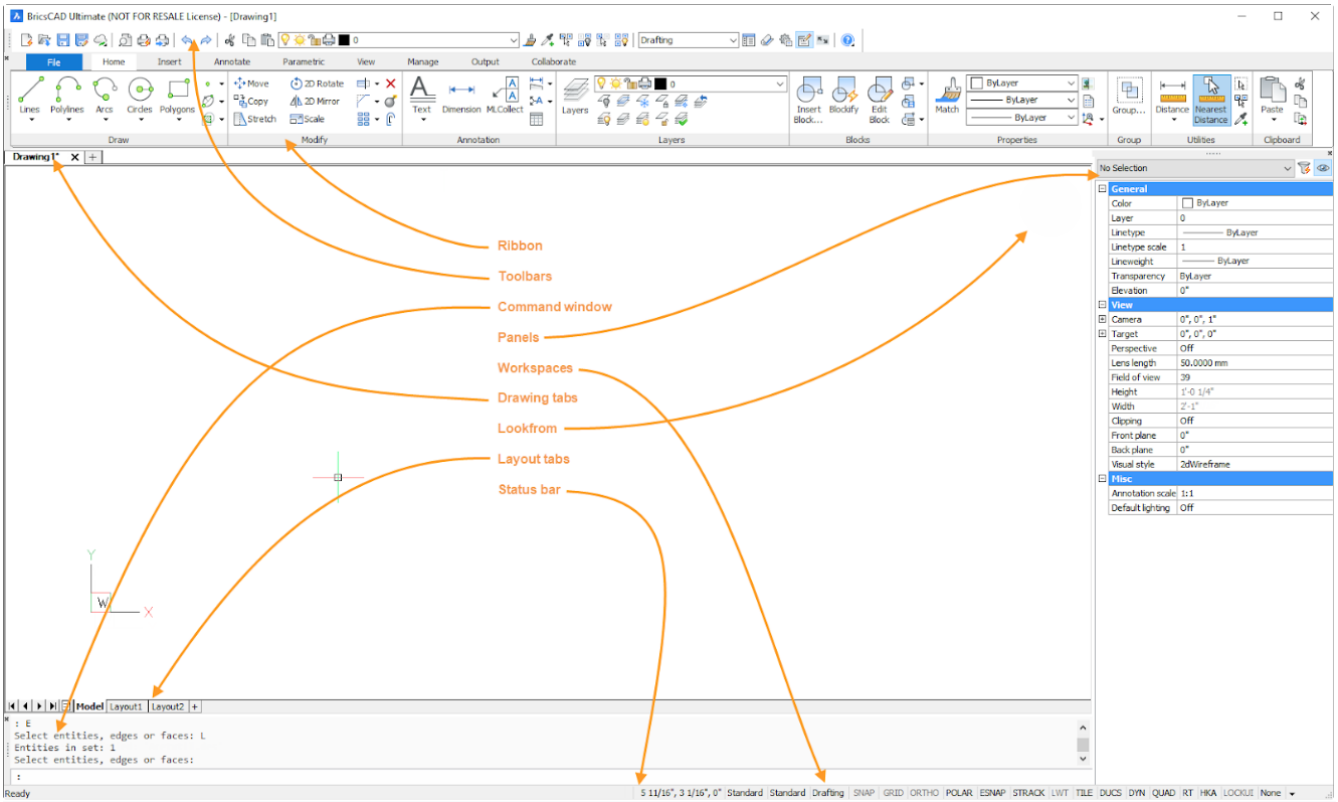

Tips
Welcome!
Start modeling directly with solids. Just drag, push or pull and your design concept will solidify right in front of your eyes.
Essentials
Quick draw:
Design buildings and rooms by drawing rectangles and L-shapes.
Drag:
move, change, copy solids.
Hold Ctrl:
to toggle faces, edges and solids.
Press Tab:
to select obscured geometry.
Snapping:
use F3 to toggle snapping.
Need more help?
Visit our help center
Close

Figure 1.63



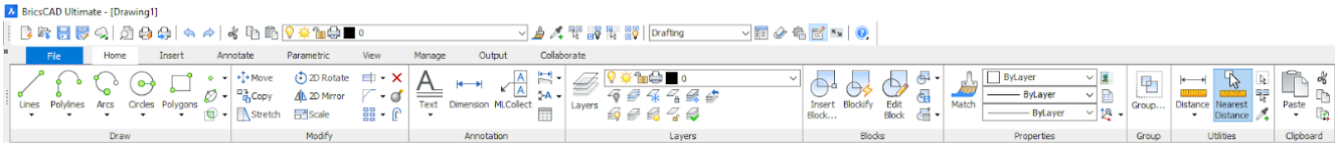
1.10.8 Exploring the BricsCAD Interface

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/Layout.PNG})

Menus, toolbars, and the ribbon

Many CAD users want nothing to do with the ribbon because it can take up significant drawing space. They prefer, instead, to access their tools from the command line or from menus and toolbars. Others, like myself, are willing to give up screen space in exchange for easy access to relevant tools on the ribbon. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/MenusToolbarsRibbons.PNG})

Figure 1.64



Command line

If you're a veteran- or power-user, you likely rely heavily on the Command line. In fact, you may prefer it over menus, toolbars, and the ribbon. You'll notice the Command line is docked at the bottom of the display by default. If you prefer, you can drag and dock it at the top or sides. And, of course, you can drag it to the drawing area or even another monitor as a floating palette. As a former AutoCAD user, you may notice the Command line doesn't have clickable options. Instead, BricsCAD automatically displays all the Command line options in an easy-to-access list displayed in the upper right corner of the drawing area. No need to right-click! If you prefer the option list in another location, you can drag it, even to another monitor, while you're in the command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/CommandLine.PNG}) A right-click menu enables you to control the behavior of the Command line. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19

Figure 1.65

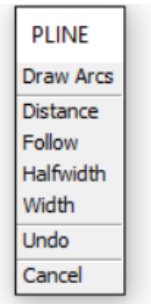
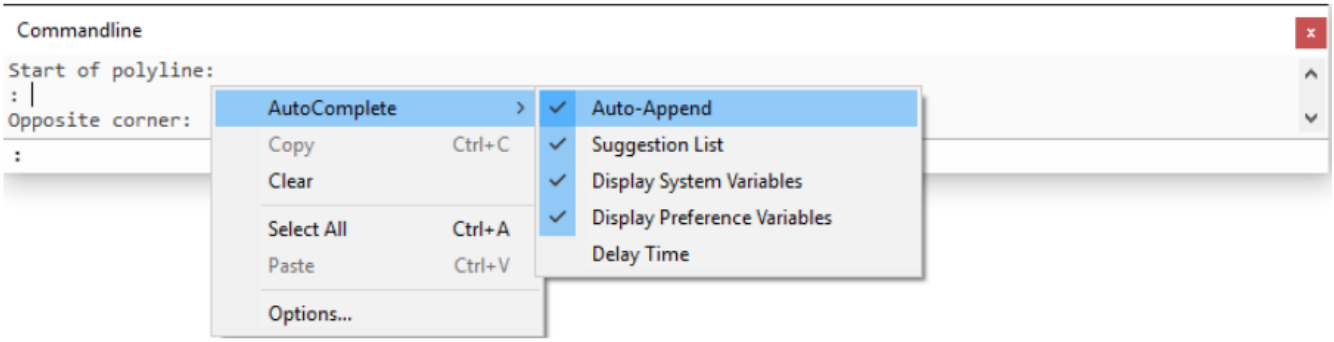




Figure 1.66



Dockable Panels (aka Palettes)

You may not think of the Command line as a dockable panel (palette), but it is. Other dockable panels include Properties, Sheet Sets and, of course, Tool Palettes. Dockable panels are special because they can be docked or floating and easily resized. And, unlike dialog boxes, they can remain open while you're using other commands. In BricsCAD, dockable panels have another special power that may surprise most AutoCAD users. They can be combined to create tabbed panels! Simply open your favorite dockable panels and then drag one panel to the center of another one. When the blue box appears, let it drop. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingS
Voila! Multiple panels, stacked on top of each other, consuming minimal space! (\vref {fig:C:/Dev/Task-2448/images/staticbric
If you're ambitious, or maybe crazy, you can stack all the dockable panels! (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

Figure 1.67

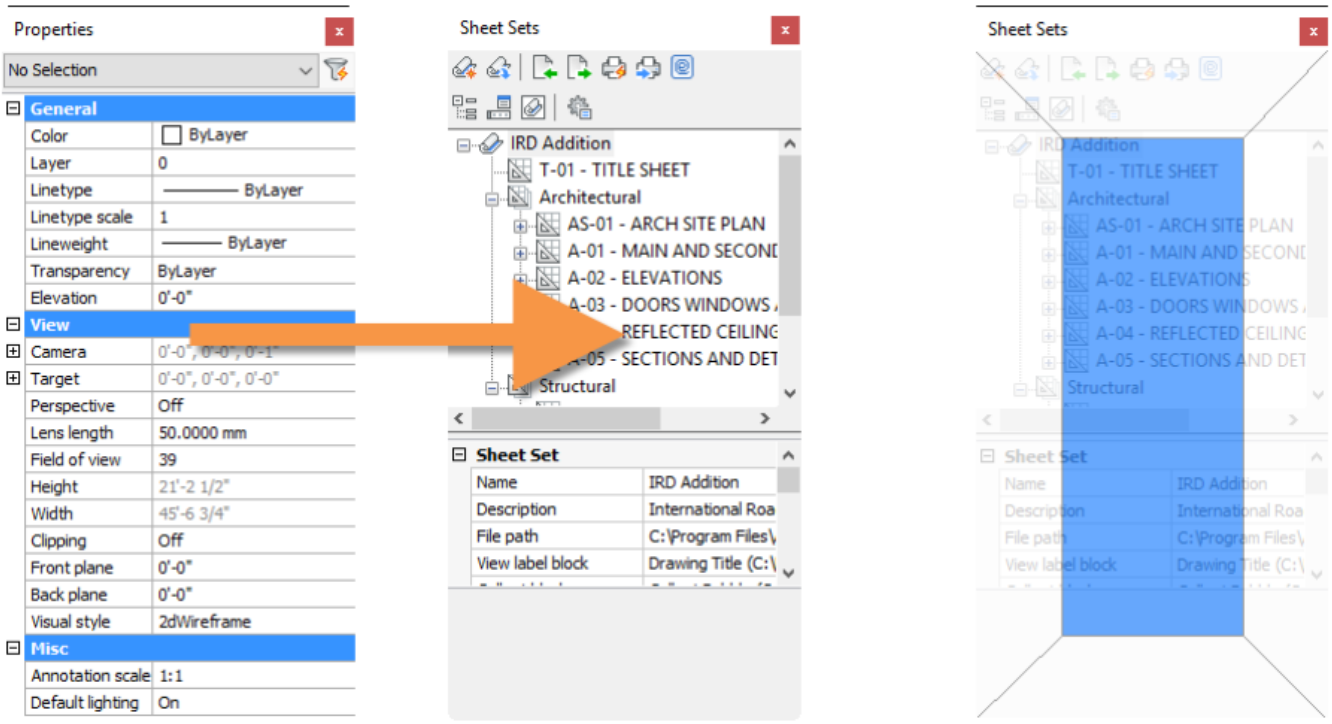

Properties
No Selection
General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Lineweight
ByLayer
Transparency
ByLayer
Elevation
0'-0"
View
Camera
Target
Perspective
Off
Lens length
50.0000 mm
Field of view
39
Height
21'-2 1/2"
Width
45'-6 3/4"
Clipping
Off
Front plane
0'-0"
Back plane
0'-0"
Visual style
2dWireframe
Misc
Annotation scale
1:1
Default lighting
On
Sheet Sets
IRD Addition
T-01 - TITLE SHEET
Architectural
AS-01 - ARCH SITE PLAN
A-01 - MAIN AND SECOND
A-02 - ELEVATIONS
A-03 - DOORS WINDOWS
REFLECTED CEILING
A-05 - SECTIONS AND DET
Structural
Sheet Set
Name
IRD Addition
Description
International Roa
File path
C:\Program Files\
View label block
Drawing Title (C:\
Sheet Sets

Figure 1.68

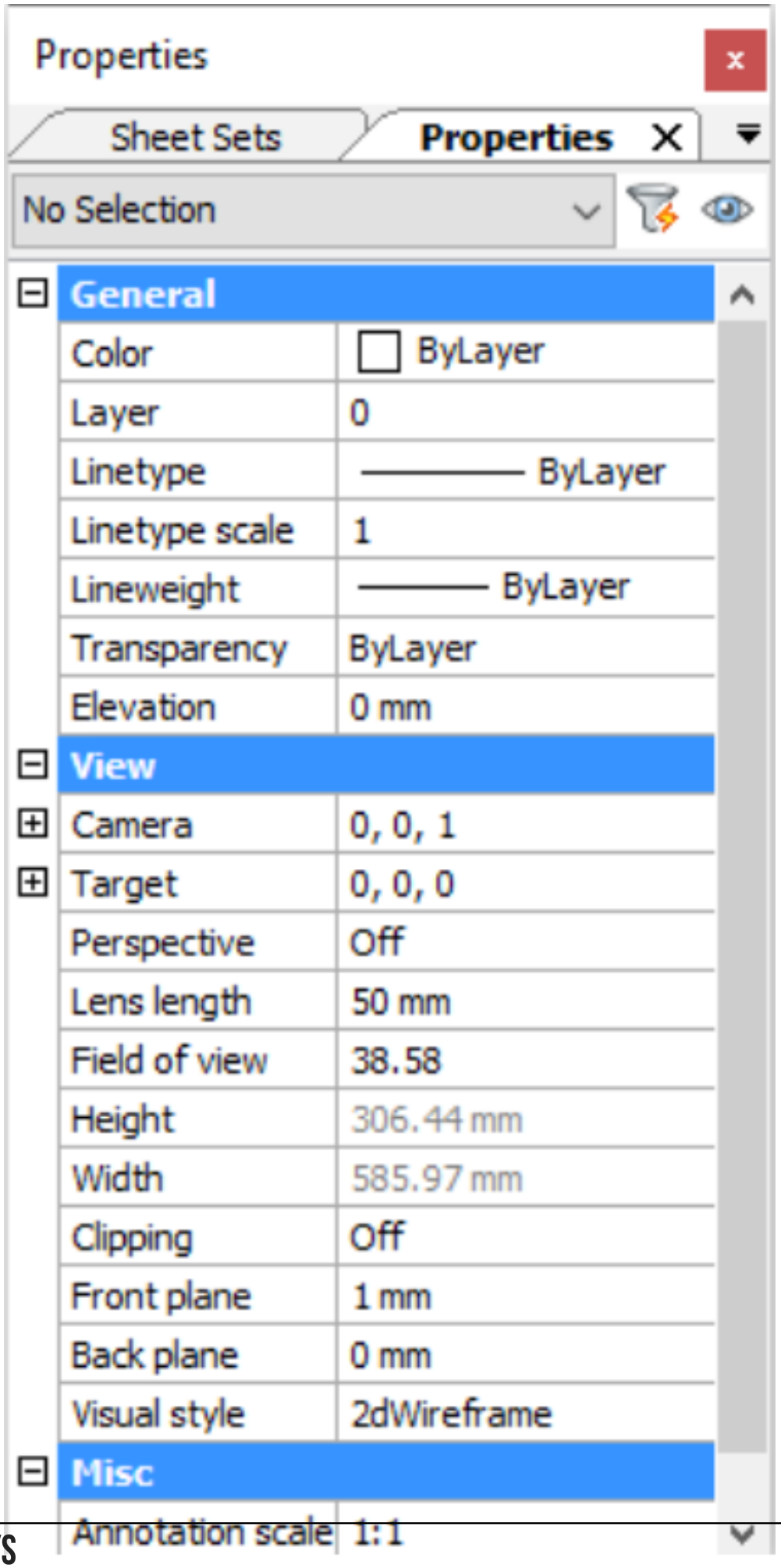
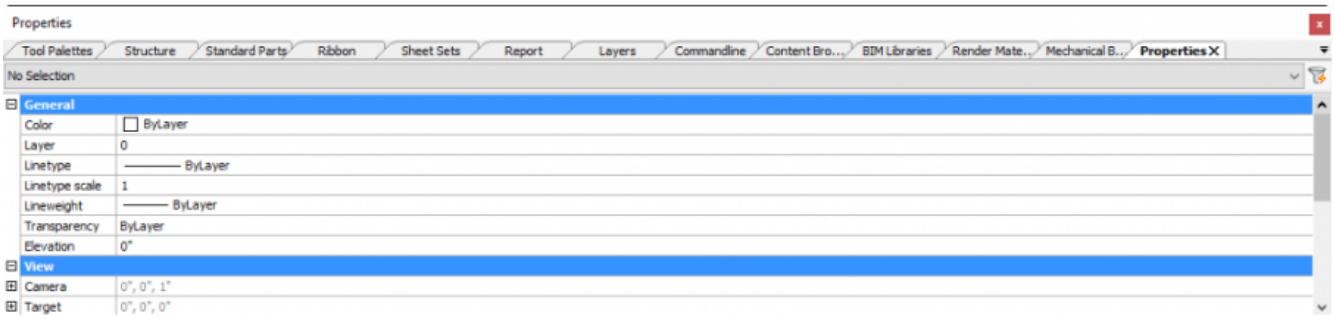

Properties
Sheet Sets
Properties
No Selection
General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Lineweight
ByLayer
Transparency
ByLayer
Elevation
0 mm
View
Camera
0, 0, 1
Target
0, 0, 0
Perspective
Off
Lens length
50 mm
Field of view
38.58
Height
306.44 mm
Width
585.97 mm
Clipping
Off
Front plane
1 mm
Back plane
0 mm
Visual style
2dWireframe
Misc
Annotation scale
1:1

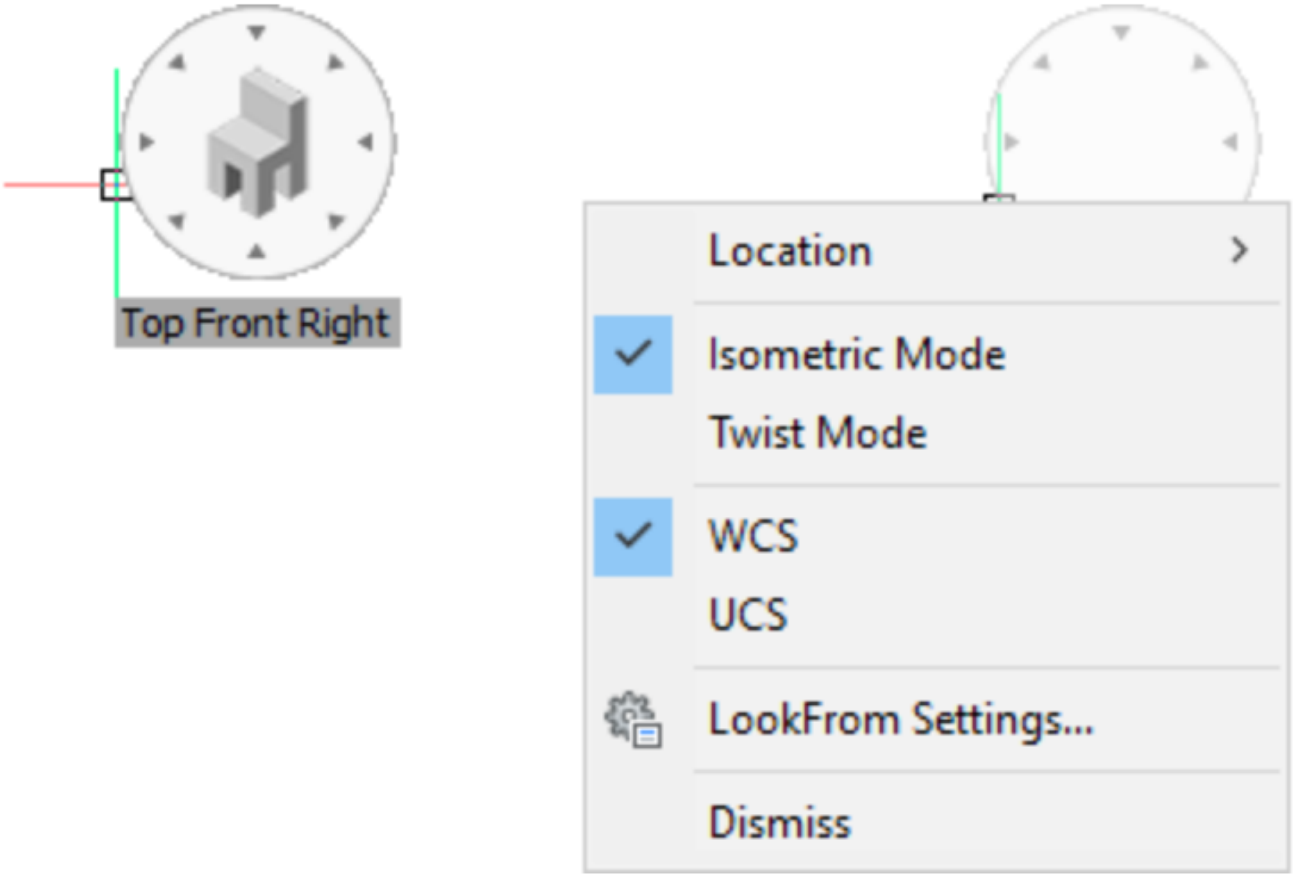
Figure 1.69


Properties
Tool Palettes
Structure
Standard Parts
Ribbon
Sheet Sets
Report
Layers
Commandline
Content Bro...
BIM Libraries
Render Mate...
Mechanical B...
Properties
No Selection
General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Lineweight
ByLayer
Transparency
ByLayer
Elevation
0"
View
Camera
0", 0", 1"
Target
0", 0", 0"

Lookfrom

The name, LookFrom, might not be familiar to you former AutoCAD users, but the functionality probably is. It's the navigation control in the upper right corner of the drawing window that lets you look at your model from different sides. Pass your cursor over the LookFrom tool and select the side from which you want to view your model. A right-click menu offers relevant controls. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingSta

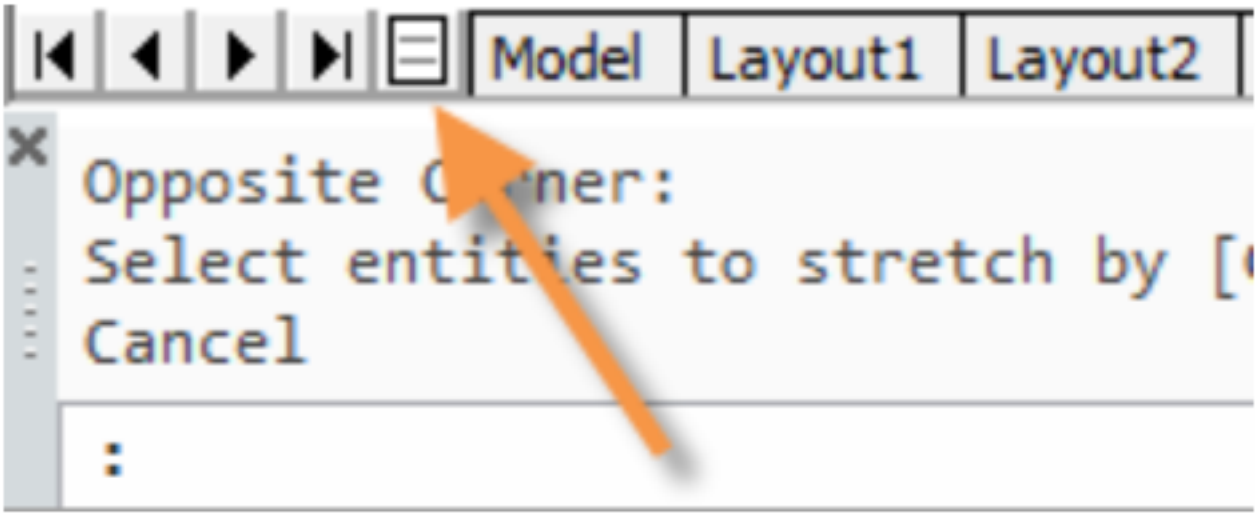
Figure 1.70



Layouts

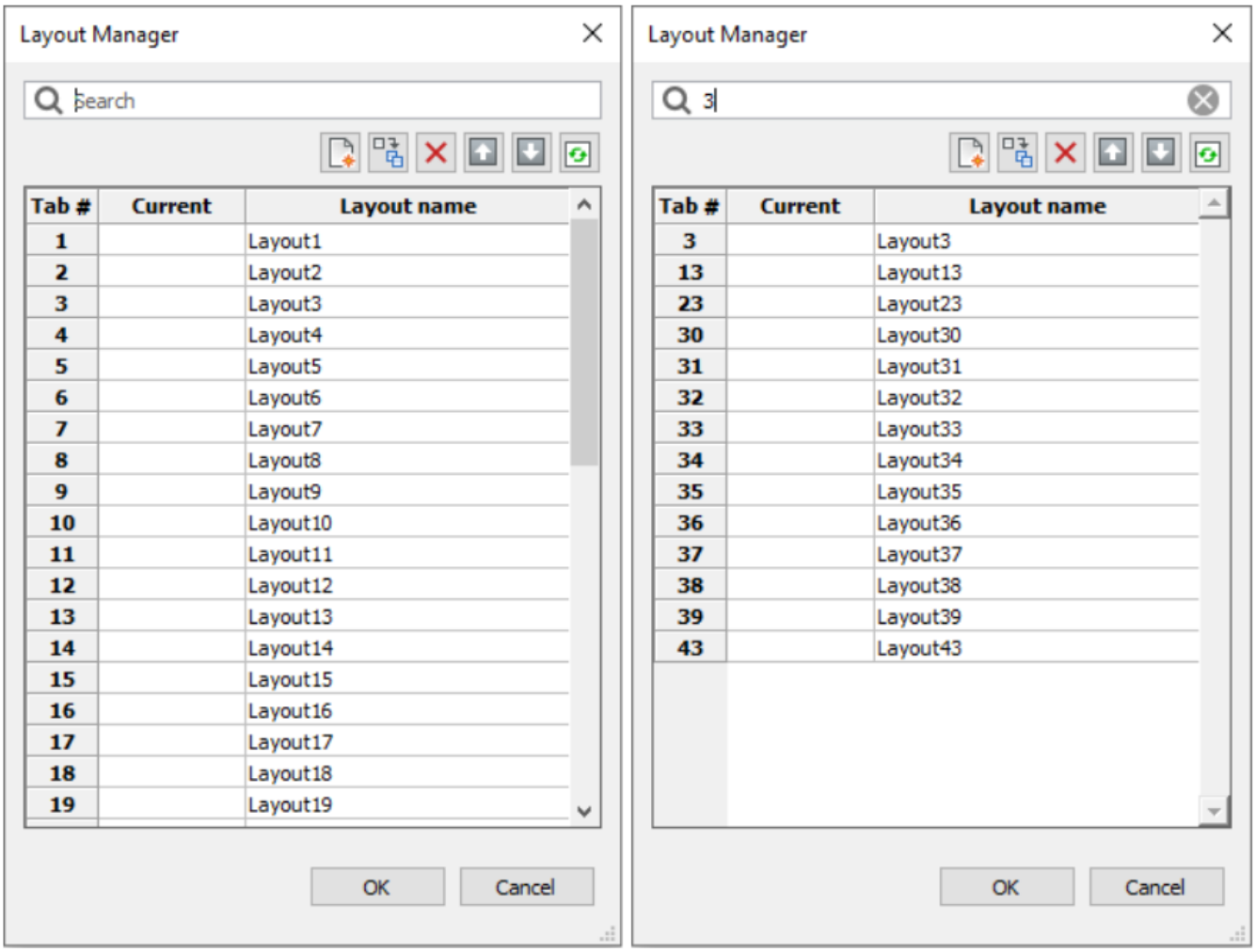
I promised to point out the “most noticeable” differences as I travel through the land of BricsCAD. In general, drawing layouts look and act as you may expect coming from AutoCAD. I did, however, notice this little icon next to the Model tab. While this is barely noticeable, it’s certainly worth a mention! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/Layouts.PNG}) Selecting that icon opens the Layout Manager where you can view, add, copy, remove, and reorder layouts in a single location! Easily select and manage multiple layouts. And, you can even search for character strings to easily find relevant layouts. A handy tool for working on drawings with many layouts! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Ge

Figure 1.71



Tutorial: Exploring the BricsCAD User Interface Click here (1.2.1) to watch

Figure 1.72


Layout Manager
Search
Tab # Current Layout name
1 Layout1
2 Layout2
3 Layout3
4 Layout4
5 Layout5
6 Layout6
7 Layout7
8 Layout8
9 Layout9
10 Layout10
11 Layout11
12 Layout12
13 Layout13
14 Layout14
15 Layout15
16 Layout16
17 Layout17
18 Layout18
19 Layout19
OK
Cancel
Layout Manager
3
Tab # Current Layout name
3 Layout3
13 Layout13
23 Layout23
30 Layout30
31 Layout31
32 Layout32
33 Layout33
34 Layout34
35 Layout35
36 Layout36
37 Layout37
38 Layout38
39 Layout39
43 Layout43
OK
Cancel

1.10.9 Command Access

Using the Command line

The Command line is, historically, the fastest and most efficient method for most users to launch commands and set system variables. Command names, aliases, and system variables are, in most cases, the same in BricsCAD and AutoCAD. If, as a former AutoCAD or AutoCAD LT user, you primarily use the Command line to launch commands, your transition to BricsCAD will be virtually seamless. So, jump in and get started! If you enter a command or system variable that isn't found, check out this document. It has valuable information for AutoCAD users transitioning to BricsCAD. At the end of the doc, you'll find appendices for mapping commands and system variables. Like AutoCAD, BricsCAD has more than 1700 commands and variables. That's a lot to try and remember if you're relatively new to either of these applications. So, although Command line access is fast and readily available, I'll focus on the UI as I document my BricsCAD Journey. I won't, however, address every UI access method. Instead, I'll focus on what's significantly different from AutoCAD.

Using menus, toolbars and the ribbon

The appearance and location of tools on the toolbars and ribbon varies slightly between AutoCAD and BricsCAD. But, not enough to warrant detailed blog posts. If you're accustomed to working in a default AutoCAD or AutoCAD LT environment, I'm confident you can quickly find your favorite tools and easily adapt to the slight differences in BricsCAD. If you're used to working in a custom environment, you'll be happy to know the BricsCAD UI is fully customizable. I'll talk more about that in a future post. But not until you've had a chance to experience the Quad!

Using the Quad

What's the Quad? At first glance, you may think the Quad is simply a rollover tooltip because it displays basic entity properties like color and layer. But the Quad is so much more! It's intelligent and contextual, offering relevant tools and information when you need them right at the cursor! The first time I watched someone use the Quad, it blew my mind. He was editing so fast I couldn't see him selecting objects or launching commands. It seemed like the software was reading his mind! Like magic!

Editing with the Quad The Quad offers different information and editing tools based on your behavior. If, for example, you access the Quad, by hovering over or selecting an entity, it displays basic information about the entity, which you can edit. It also includes the most recently used *relevant* tool for editing that type of entity. You can select the tool, which is right by the cursor. Or, simply right-click on the entity to launch the command. You don't even have to select the entity or the tool! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/Qua
If the current tool isn't what you want, hover the cursor over the tool to expand the Quad. Additional recently used *relevant* tools are displayed. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/
If the tool you want still isn't accessible, pass your cursor over any of the tabs at the bottom of the Quad to expand the command groups and select a *relevant* tool. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCA
Did you notice my repeated emphasis on *relevant*? BricsCAD software includes hundreds of commands but only the ones most relevant to your current context are displayed in the Quad. The relevant tools for editing a polyline, for example, are quite different from those for editing text. So, the type of entity you're editing determines which tools are displayed in the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/Quad

Figure 1.73

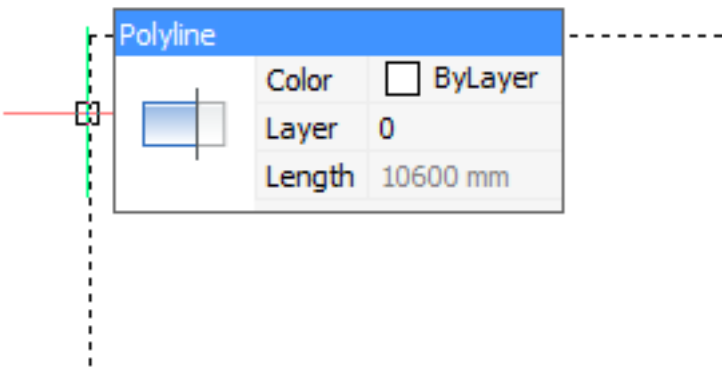


Figure 1.74

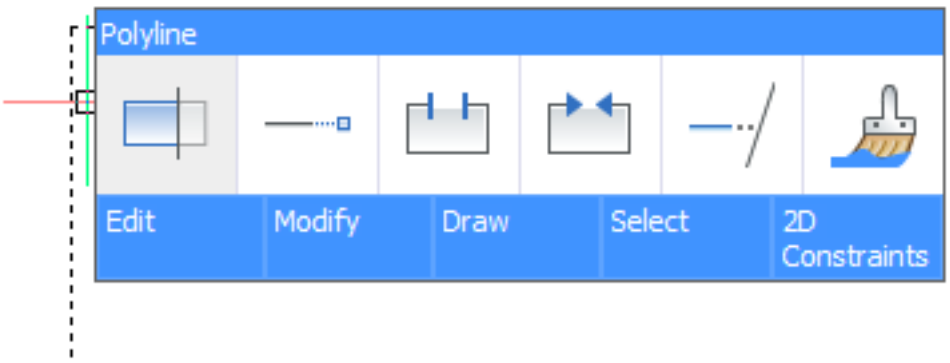


Figure 1.75

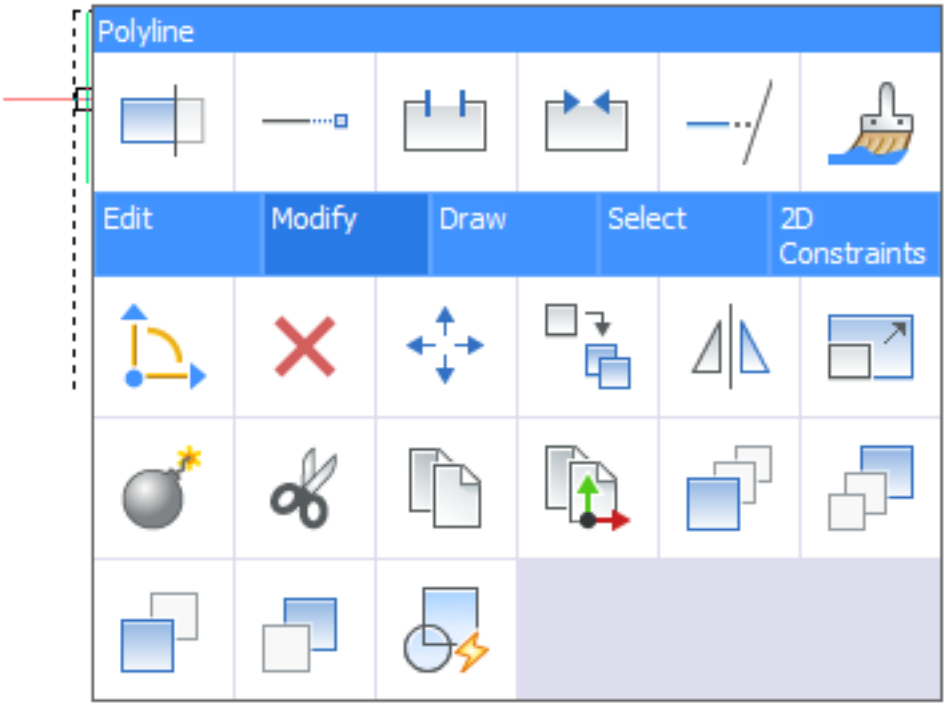
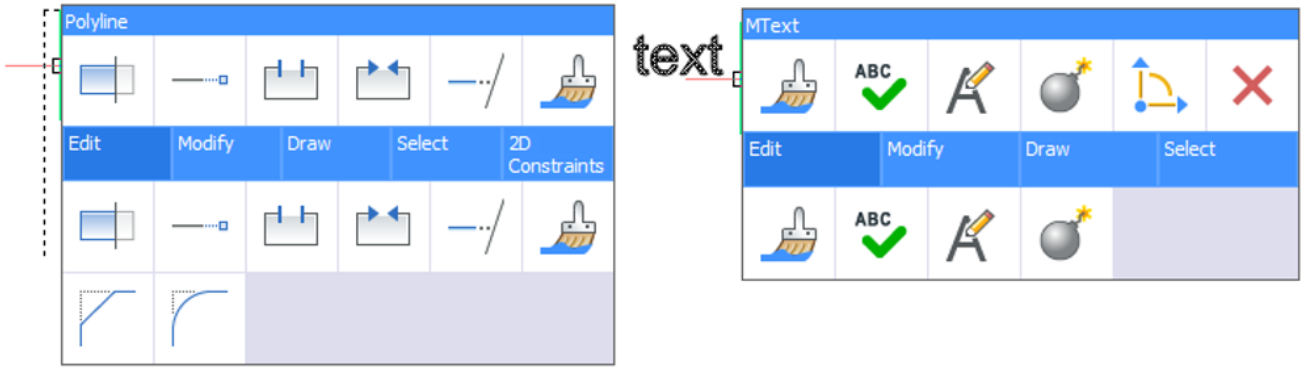
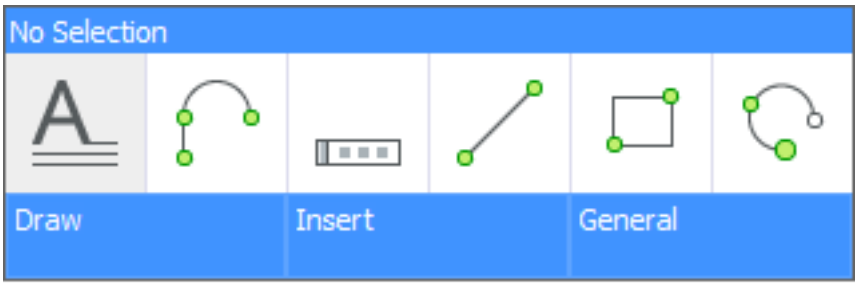


Figure 1.76



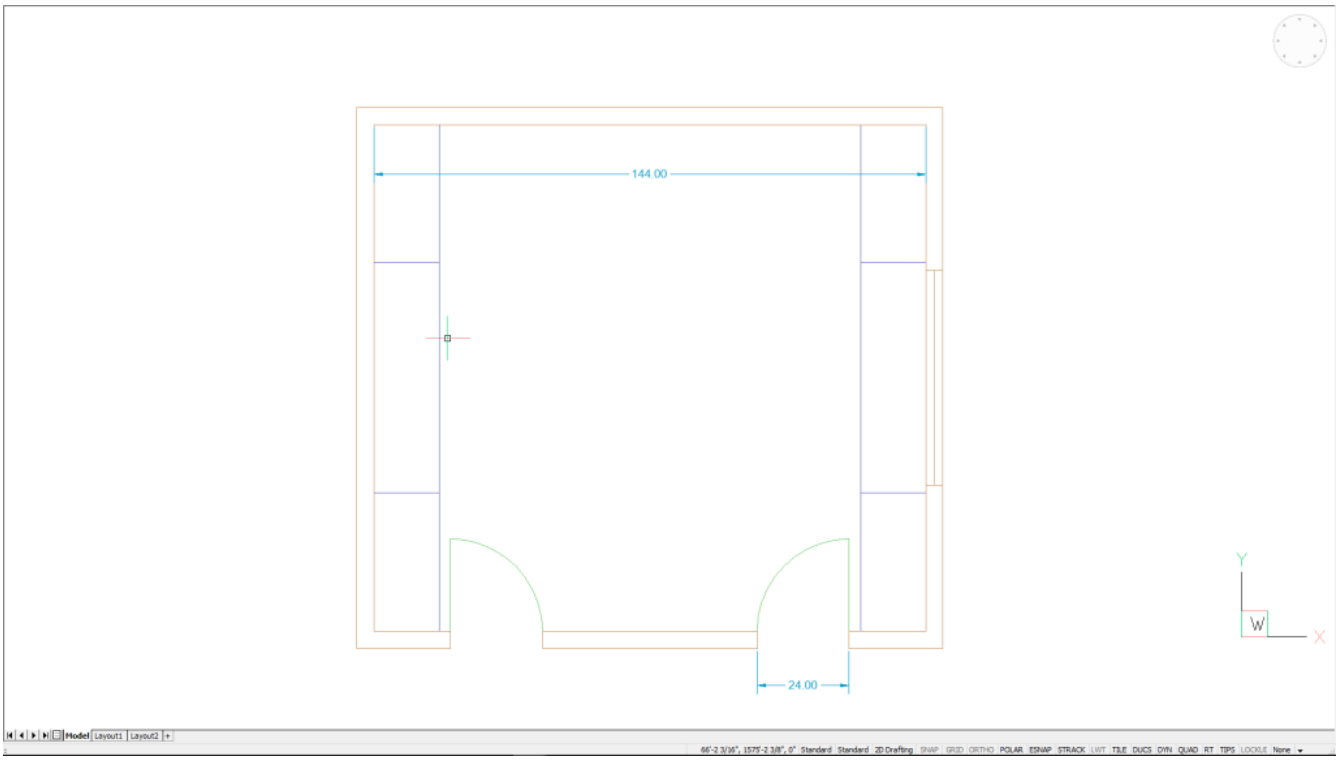
Drawing with the Quad The Quad isn't limited to editing existing entities. You can, for example, draw and insert new entities using the Quad. Simply right-click in a blank part of the drawing area with no entities selected. The Quad offers easy access to draw and insert tools as well as general ones such as Print and Settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/DrawingWithQuad.png})

Figure 1.77



Why the Quad? The speed with which you can access commands using the Quad is a clear benefit. The Quad offers a robust set of drawing and editing tools right at the cursor, with minimal clicks. With all the necessary tools appearing as you need them, is there any reason to access toolbars? Menus? The ribbon? What about the command line? Curiosity got the best of me. So, I turned off the menu and Command line and turned on CleanScreen. Starting with a blank drawing, I drew my office. I created layers, drew a rectangle and arc. I exploded, offset, copied, mirrored, moved, trimmed, extended, dimensioned, and modified dimension properties. And, I did it all with the Quad! I only used the keyboard to enter values and deselect entities (ESC key). Nearly everything I know about the Quad, I learned while writing this blog post. I could have drawn my office plan using the familiar Command line and keystrokes I've used for more than 30 years. But, even as a brand-new BricsCAD user, I did it faster with the Quad! I wish you could have watched me as I was drawing my floorplan. My dogs were here to observe but they just didn't share my enthusiasm. The Quad is so Qool!! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/OfficePlan.png})

Figure 1.78



Let's make a deal I'll tell you how to turn off the Quad if you promise to commit one lunch hour to drawing your office plan using nothing but the Quad? I'll even share a few bonus tips that I learned while working with the Quad. Tip 1: Contextual menus, similar to AutoCAD, are available if you select an object and then right-click and hold for a second before releasing. If you release too quick, it displays the Quad. I'm only sharing this to help you with your transition. I doubt you'll need the right-click menu after you're comfortable with the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/ContextualMenu.png}) Tip 2: When running with CLEANSCREEN on and the Command window off, look in the lower left corner of the drawing area. A prompt, command history and even suggestion list if you type a command, are subtly displayed without taking any screen space! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/CommandHistory.png}) Tip 3: You can turn the Quad on or off using the status bar control. Right-click on it for additional controls. But, remember our deal! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/GettingStarted/StatusBar.pr

Figure 1.79

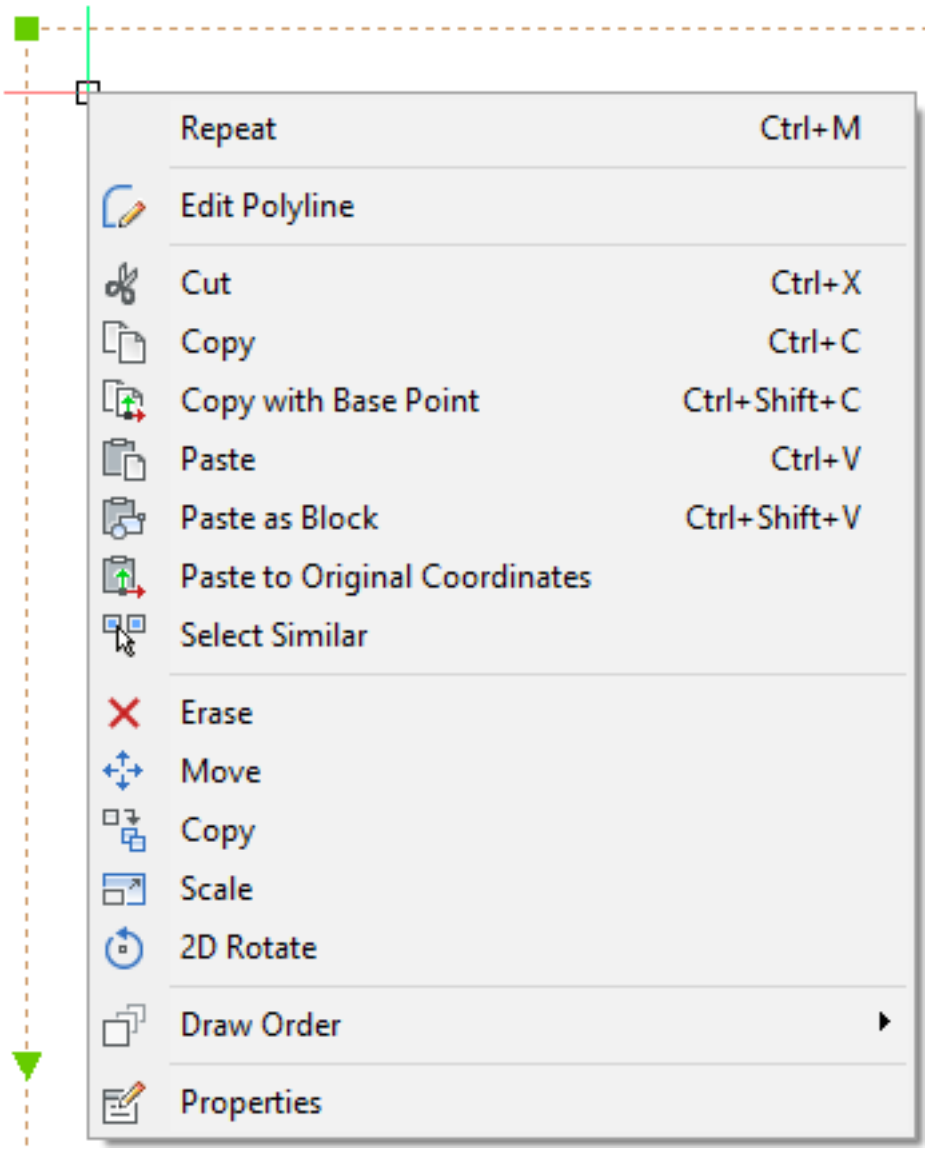


Figure 1.80

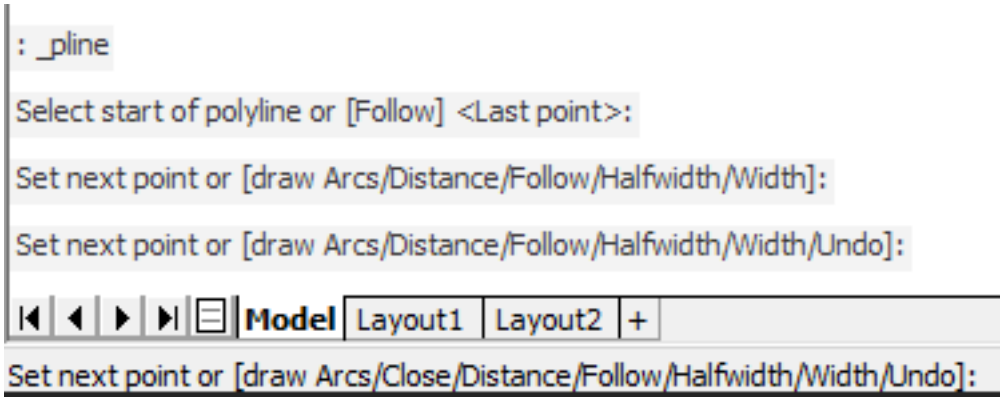


Figure 1.81



1.11 Installation & Licensing

1.11.1 Installation

BricsCAD comes as a single download file. Each new major version can be installed next to a previous major version and can be run in trial mode first. BricsCAD can be installed on :

- Microsoft Windows 10, 8, 7 and Vista on both 32 and 64 bits.
- Linux: Ubuntu 16.04 LTS or newer, or on other distributions with a c/c++ runtime that is at least as new. BricsCAD (Linux) V19 is compiled with gcc 5.3 on Ubuntu 16.04 LTS. As a result the minimum required c++ runtime (libstdc++) is 5.3.
- Mac OSX 10.8 or higher.

Related Topics

Silent Installation (1.11.2)

1.11.2 Silent Installation (Windows only)

Silent installations are installations that run without a user interface. A normal (non-silent) installation receives the necessary input from the user in the form of responses to dialog boxes. However, a silent installation does not prompt the end user for input. The user interface of an MSI installer (Microsoft Installer) typically queries the target system and displays an installation wizard (Setup Wizard) which enables the user to change various options that will affect the installation. The same MSI can also perform a so-called 'quiet mode' or 'silent mode' installation whereby the User Interface of the Setup Wizard gets suppressed. This is done by running the msiexec.exe command-line utility with the /qn option (= quite no interface) and specifying on the command line all the information that the wizard would normally gather. Therefore, any actions that occur within the user interface sequence will not be performed during a 'silent' installation.

To silently perform a BricsCAD installation

1. Do one of the following:
   - Open a Command Window, then change the directory (cd) to the location of the BricsCAD installer .msi file.
   - In the Windows Explorer dialog, browse to the location, then hold down the Shift key, right click and select **Open command window here** in the context menu.
2. Type **msiexec /i**,then do one of the following:
   - type **"<BricsCAD installer filename>"** (e.g. "BricsCAD-Vxx.x.xx-x-en_US(x64).msi").
   - drag the BricsCAD installer file from the Windows Explorer dialog onto the command window.
3. Type **/qn** at the command prompt.
4. (option) To suppress the adding of a Desktop Shortcut: add **ADDDESKTOPSHORTCUT=""**.
5. (option) To suppress the display of the Release Notes when the installation completes: add **SHOWRELEASENOTES=""**.
6. (option) To install BricsCAD in a specific folder, different from the default folder: add **APPLICATIONFOLDER="your_app_f**
7. (option) To specify a folder for the license file if any of the properties **BRXLICENSEKEY**,**BRXLICENSEFILE** or **BRXLICENSESERVER** are defined (default is**APPLICATIONFOLDER\BricsCAD.lic**: add **BRXLICENSEDESTFILE="path_to_lic_file"**.
8. (option) To activate the license and store it in the folder specified by**BRXLICENSEDESTFILE**: add **BRXLICENSEKEY="your_license_key"**.
9. (option) To copy the license file to the folder specified by **BRXLICENSEDESTFILE**: add **BRXLICENSEFILE="your_license_fi**
10. (option) To create a license file in the folder specified by**BRXLICENSEDESTFILE**,containing the server host specification: add **BRXLICENSESERVER="host"**, or **BRXLICENSESERVER="port@host"** if your license server is using a different port than the default port, being 5053.
11. (option on 32 BIT only) To not install VBA: add **INSTALLVBA="0"** or **INSTALLVBA=""**. The complete entry at the command prompt might be: **msiexec /i "BricsCAD-Vxx.x.xx-x-en_US(x64).msi" /qn ADDDESKTOPSHORTCUT="" SHOWRELEASENOTES=""**.
12. Press Enter to launch the installation.

NOTES

- For more options, type: **msiexec /?** at the command prompt**.**
- On the Microsoft website, you can find more informationabout the **msiexec** command.

Related topics

Installation (1.11.1)

1.11.3 Licensing

Commands: LICENSEMANAGER (6.20.22), LICPROPERTIES (6.20.24), LICPROPERTIESBIM (6.20.25), LICPROPERTIESCOMMUNICATOR (6.20.26), LICPROPERTIESMECHANICAL (6.20.27) BricsCAD is available with three types of commercial licenses combined to two platforms and three feature levels:

- Commercial licenses: Single, Volume, Network
- Platforms: Windows, Mac, Linux
- Feature levels: Classic, Pro, Platinum

See also the differences between platforms and between levels (1.10.2). BricsCAD can also run in trial mode (1.11.9) for evaluation.

Tutorial: How to push or pull solids (add the related tutorial here) Click here to watch

NOTE	The procedures described in this chapter are also valid for BricsCAD Sheet Metal and BricsCAD Communicator.

View your licenses

All your valid licenses can be viewed on the Bricsys website.

1. Log in to the Bricsys website. Make sure you log in with the Bricsys account containing your licenses. Contact your Bricsys account manager if needed.
2. In the upper right corner, click on your name. Your account page opens.
3. In the left menu choose Licenses and support contracts. All your active licenses are displayed.
4. (Optional) Click a license to see its details.

Manage your licenses

Depending on the type and status, licenses can be activated, modified and deactivated. Use the activation (1.11.5) to:

- Activate a trial license.
- Activate a Single or Volume license.
- Use a Network license (beforehand activated on a network license server).

Use the modification (1.11.12) to:

- Upgrade a license (e.g. from Single Classic to Single Platinum).
- Replace a license (e.g. a Single Classic by a Network Pro).

Use the deactivation (1.11.13) to:

- Migrate a license to another computer.
- Deactivate a license.
- Be able to reactivate a license after installing a new hard disk, changing partitions or upgrading the operating system.

1.11.4 Proxy Settings

Some computers connect to a network or the internet through a proxy server. Therefore the proxy settings should be configured before activating or deactivating a license.

To configure the proxy settings

1. In the Activate Bricsys or the License Information dialog, click the **Proxy Settings...** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/LicenseManagerActivate.png}) The Proxy dialog opens.
2. In the **Proxy** dialog, fill out the necessary fields. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/E
3. Click the OK button to continue.

NOTE	The proxy settings configuration remains active for the current BricsCAD session only. If you need to reactivate or deactivate a license in future session, you will have to reconfigure the proxy settings.

Figure 1.82

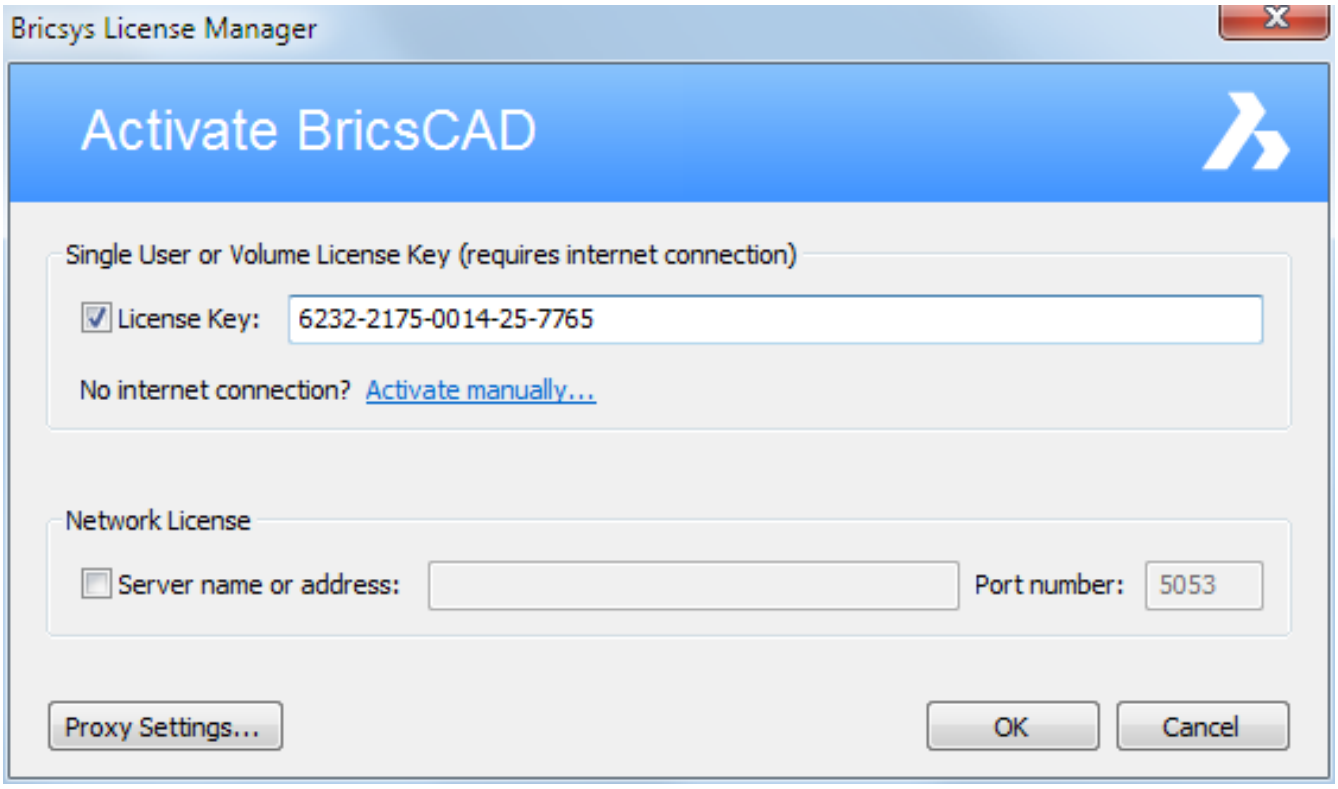
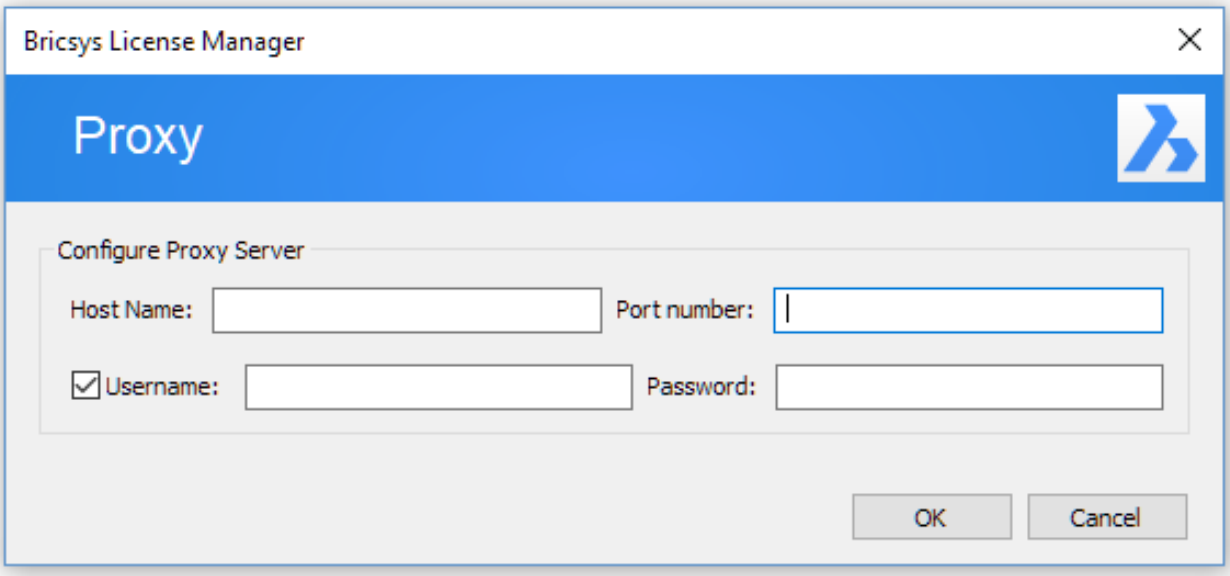


Figure 1.83



To configure the proxy settings permanently for Network License use

Using a Network license, BricsCAD needs to connect to the license server each time a session is opened. Therefore the proxy settings should be configured permanently. Proceed as follows on the computer where BricsCAD is running:

1. Create the HTTP_PROXY system variable: HTTP_PROXY=[LicenseServer]:[PortNumber]
2. (Optional) I f the proxy server supports basic authentication, create the HTTP_PROXY_CREDENTIALS variable: HTTP_PROXY_CREDENTIALS=[UserName]:[Password]

1.11.5 Activation

The BricsCAD license activation always requires an Internet connection. However, if your computer is not connected to the Internet, the license can be activated manually via another computer (which is connected to the Internet).

The first time BricsCAD is started

Use this procedure to:

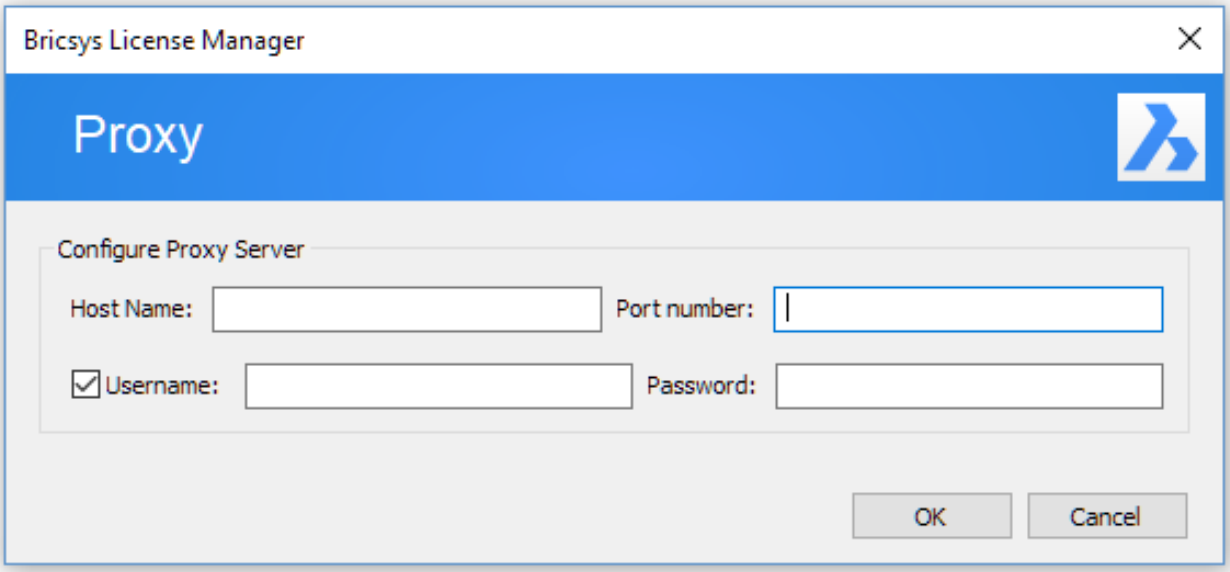
- Activate a trial license.
- Activate a Single or Volume license.
- Use a Network license (beforehand activated on a license server).

1. The Activate BricsCAD dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Inst

2. If your computer connects to the Internet through a proxy server, click the **Configure proxy...** link to configure the proxy settings (1.11.4).

3. Do one of the following:
   - Click the Activate Trial button to activate a trial (1.11.9).
   - Click the Activate Now... button to activate a Single or Volume license (1.11.10), or to use a Network license (1.11.11).

Figure 1.84



To activate a license when starting BricsCAD in trial mode

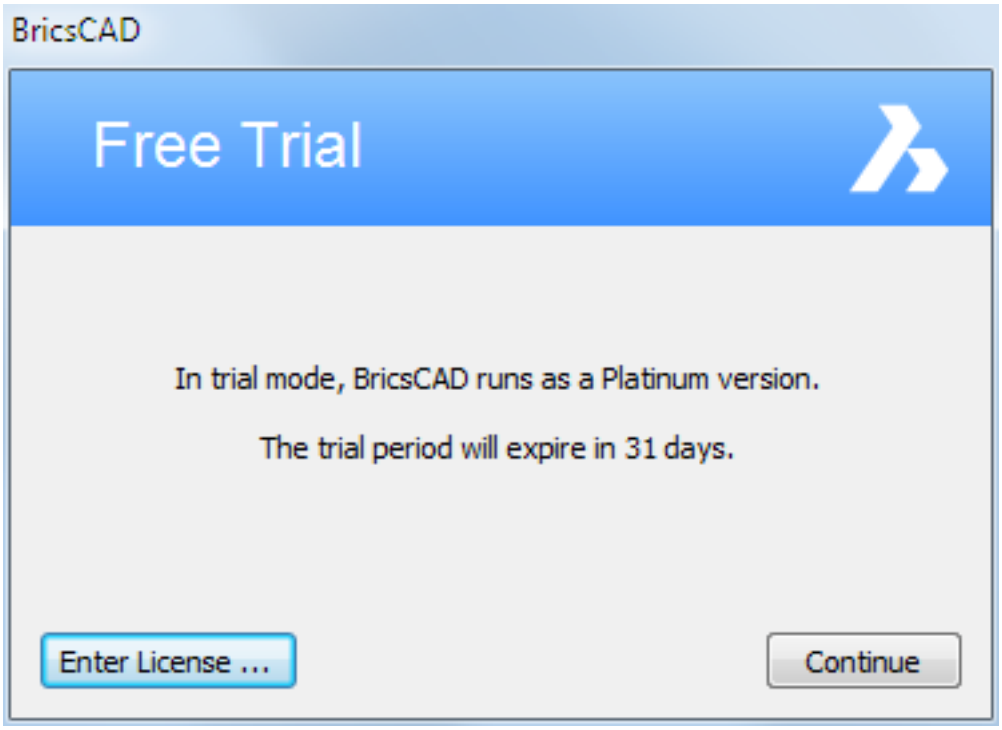
Use this procedure to:

- Activate a Single or Volume license.
- Use a Network license (beforehand activated on a license server).

1. The Free Trial dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationS

2. Click the Enter License... button to activate a Single or Volume license (1.11.10), or to use a Network license (1.11.11).

Figure 1.85



To activate a license replacing or upgrading an existing license

Use this procedure to:

- Activate a Single or Volume license.
- Upgrade a license (e.g. from Single Classic to Single Platinum). The old key is no longer valid then.
- Replace a license (e.g. a Single Classic by a Network Pro). The old key is deactivated automatically and is available to be activated on another computer.

1. In the **Help** menu, choose **License Manager...** . The **Bricsys License Manager** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/BricsysLicenseManager.png})

2. In the **Bricsys License Manager** dialog, click the **Details...** button next to the application you want to manage the license. The **Licensing Information** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS

3. Click the **Modify...** button to activate another Single or Volume license (1.11.10), or to use another a Network license (1.11.11).

Figure 1.86

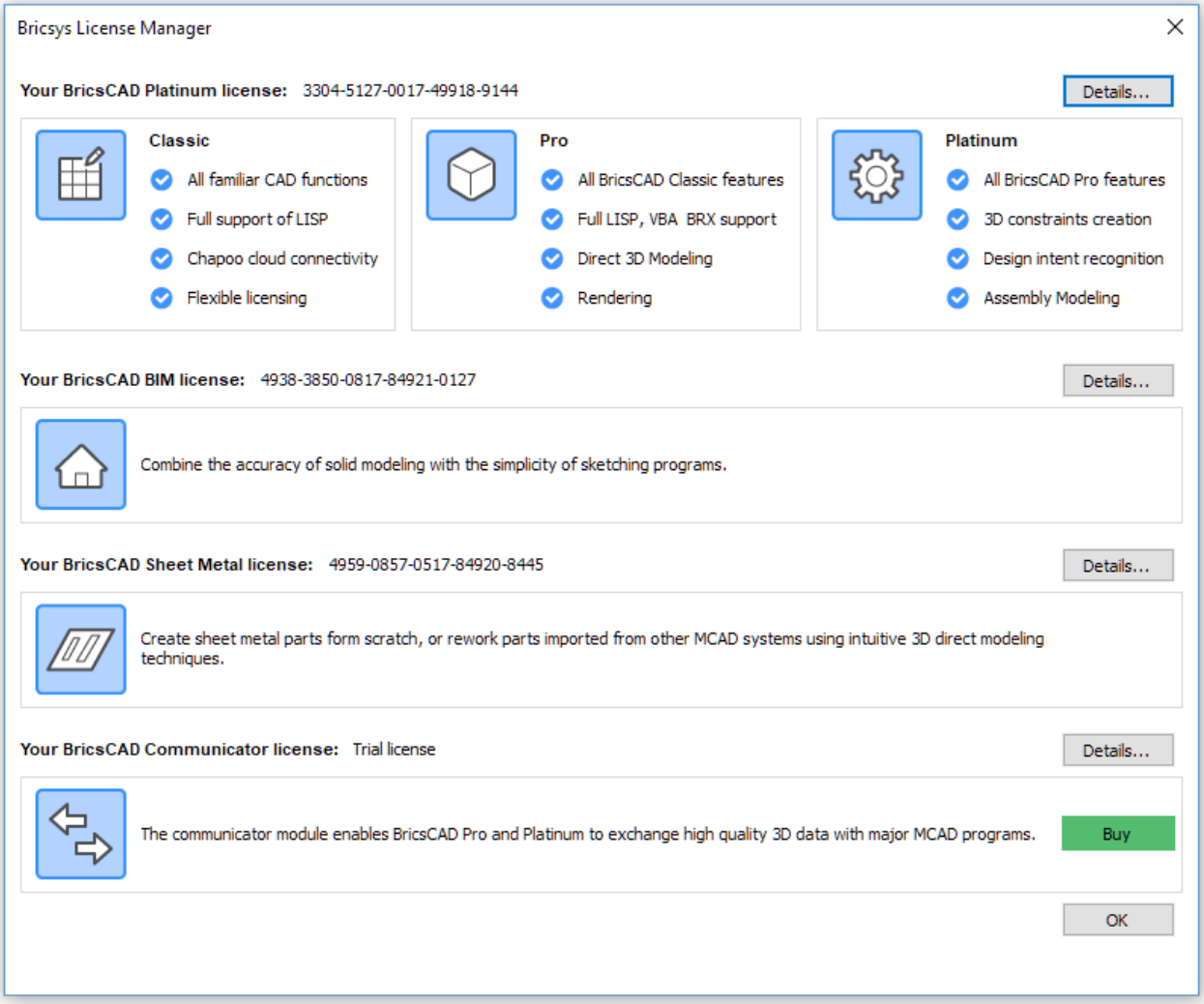
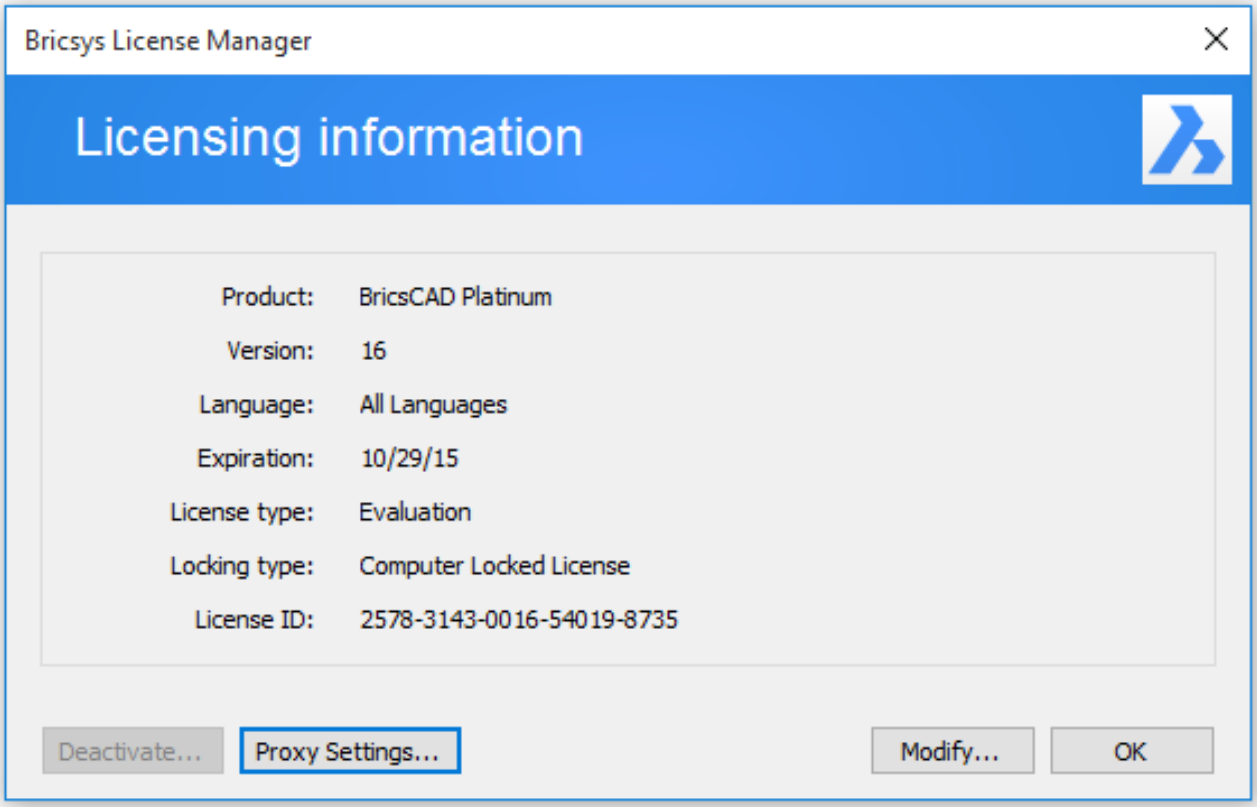

Bricsys License Manager
Your BricsCAD Platinum license: 3304-5127-0017-49918-9144
Details...
Classic
All familiar CAD functions
Full support of LISP
Chapoo cloud connectivity
Flexible licensing
Pro
All BricsCAD Classic features
Full LISP, VBA BRX support
Direct 3D Modeling
Rendering
Platinum
All BricsCAD Pro features
3D constraints creation
Design intent recognition
Assembly Modeling
Your BricsCAD BIM license: 4938-3850-0817-84921-0127
Details...
Combine the accuracy of solid modeling with the simplicity of sketching programs.
Your BricsCAD Sheet Metal license: 4959-0857-0517-84920-8445
Details...
Create sheet metal parts form scratch, or rework parts imported from other MCAD systems using intuitive 3D direct modeling techniques.
Your BricsCAD Communicator license: Trial license
Details...
The communicator module enables BricsCAD Pro and Platinum to exchange high quality 3D data with major MCAD programs.
Buy
OK

Figure 1.87



Related topics

Configure the proxy settings (1.11.4) Activate a trial license (1.11.9) Activate a Single or Volume license (1.11.10) Use a Network license (1.11.11)

1.11.6 Activate a trial manually

The BricsCAD activation always requires an Internet connection. However, if your computer is not connected to the Internet, activation can be done manually via another computer (which is connected to the Internet).

To activate a trial manually

1. Launch BricsCAD.
2. In the Activate BricsCAD dialog box, click the Activate Manually... link. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys
   The Manual Activation dialog box opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/
3. Copy the full content of the Computer Hostid field.
4. With the Internet connected computer, go the Bricsys website.
5. In the Support menu, choose My Account (login required). The My Account page opens.
6. In the left menu, choose **Licenses and support contracts**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
7. Click on **Activate trial license manually...** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD
8. In the **Host ID** field, paste the computer host ID of the computer you want to install BricsCAD on.
9. Click the **Activate** button. A license file is generated and made available for download.
10. Click the **Download License File** button. The license file **BricsCADV14.lic** is downloaded to your default download folder.
11. Copy the license file to the computer you want to install BricsCAD on.
12. In the Manual Activation dialog box, click the browse button (). Open the folder where you copied the **BricsCADV14.lic** license file to and select the file. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
13. Click the OK button.
13. In the Free Trial dialog box, click the **Continue** button. BricsCAD is launched.

Figure 1.88

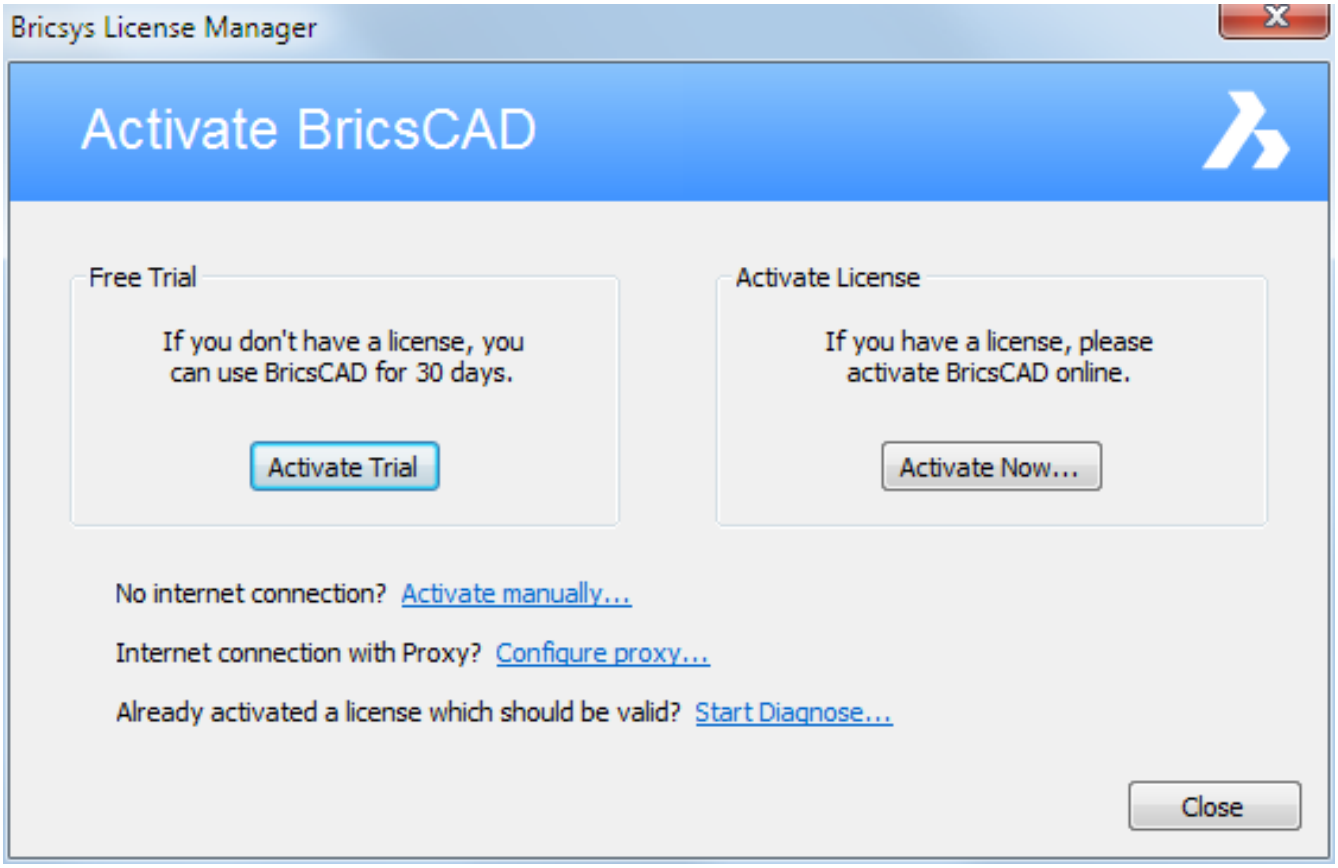


Figure 1.89

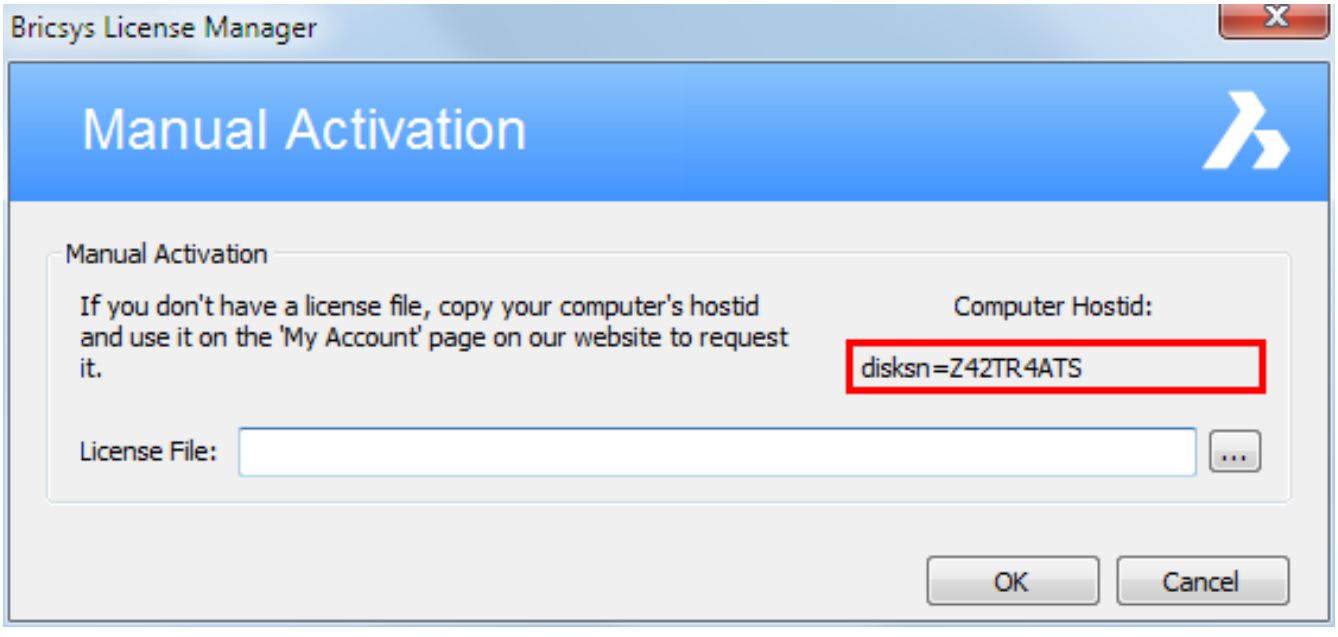


Figure 1.90

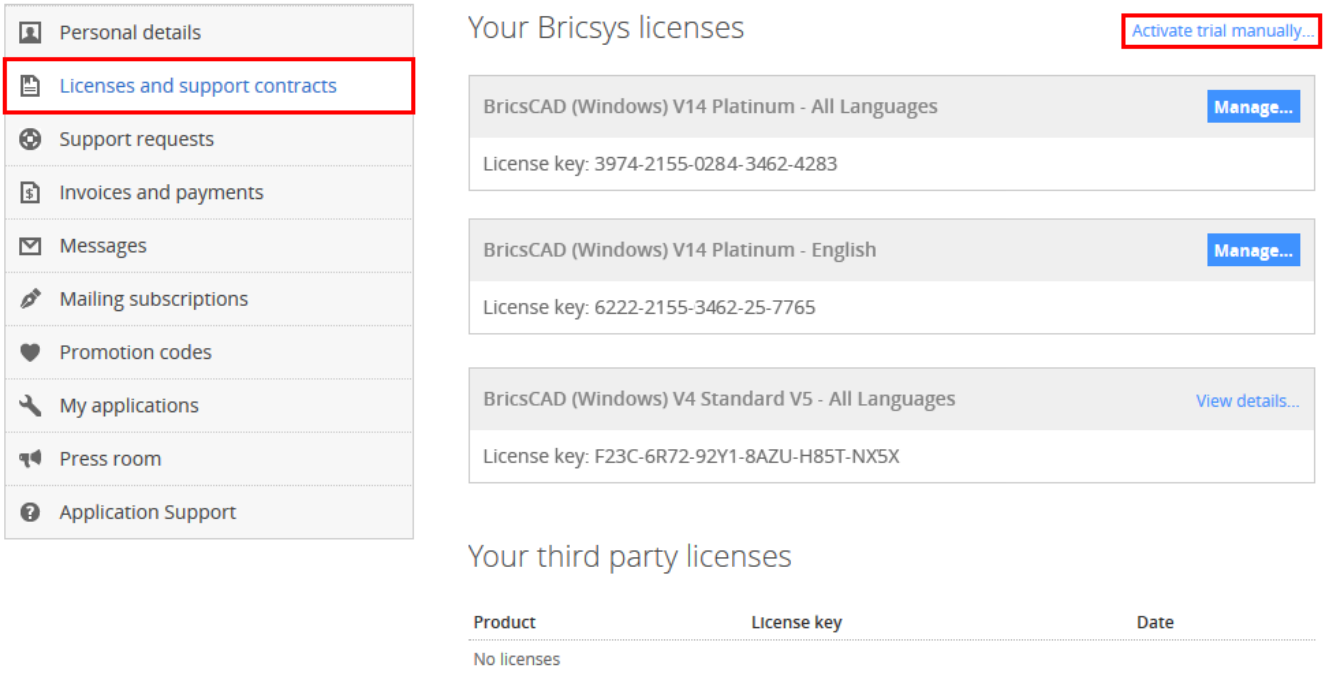


Figure 1.91

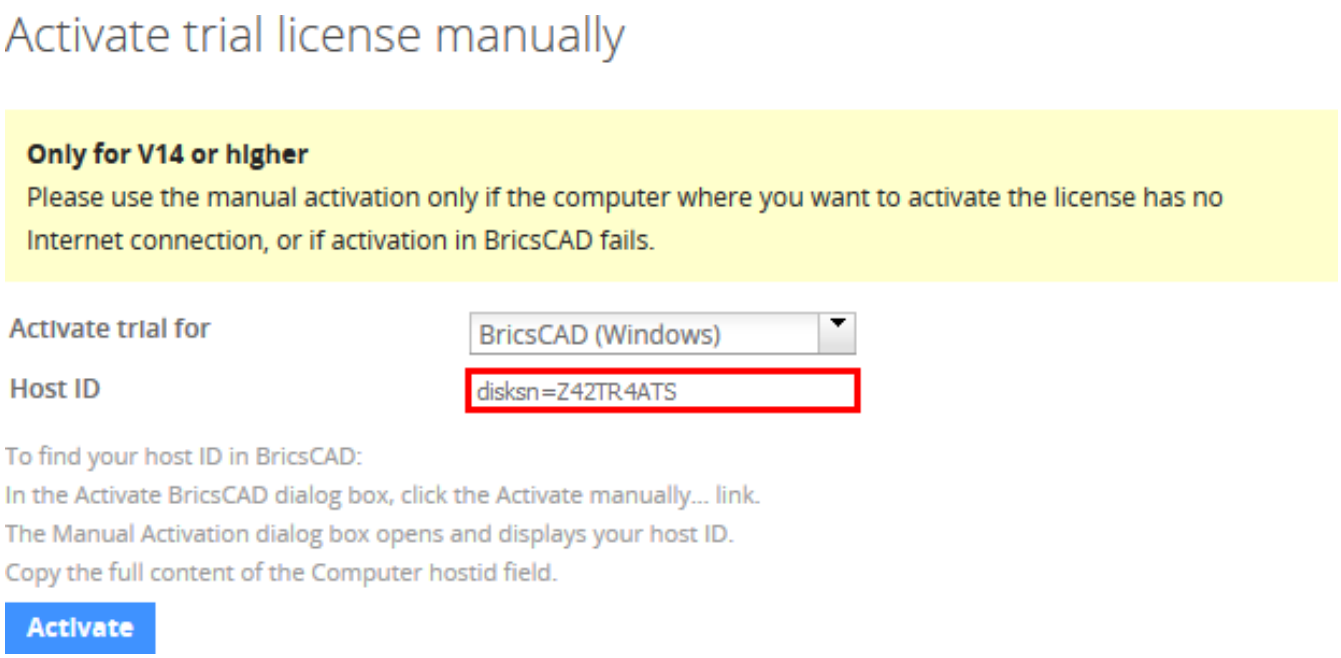
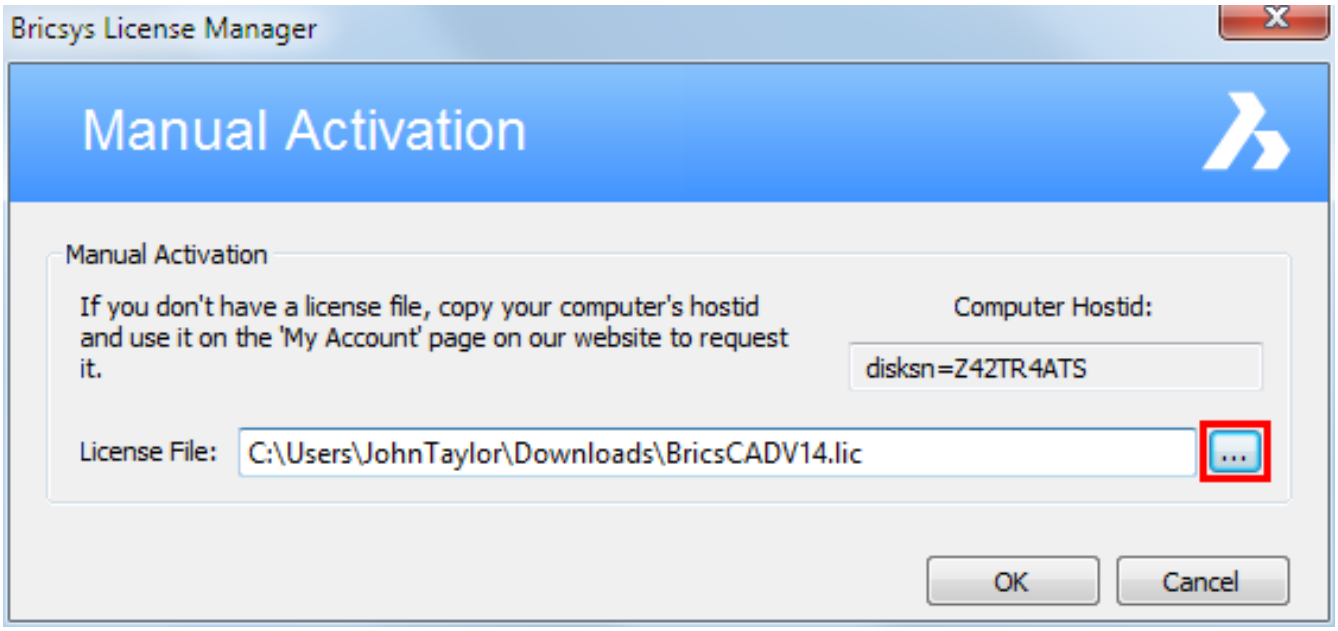


Figure 1.92


Bricsys License Manager
Manual Activation
Manual Activation
If you don't have a license file, copy your computer's hostid and use it on the 'My Account' page on our website to request it.
Computer Hostid:
disksn=Z42TR4ATS
License File:
C:\Users\JohnTaylor\Downloads\BricsCADV14.lic
...
OK
Cancel

1.11.7 Activate a Single or Volume license manually

The BricsCAD license activation always requires an Internet connection. However, if your computer is not connected to the Internet, the license can be activated manually via another computer (which is connected to the Internet).

To activate a Single or Volume license manually

1. Do one of the following:
   - The first time BricsCAD is started - go to next step.
   - When starting BricsCAD in trial mode - in the Free Trial dialog, click the **Enter License...** button.
   - When replacing or upgrading an existing license - in the Help menu, choose **BricsCAD License...** . The Licensing Information dialog box opens. Click the **Modify...**button.
2. The Activate BricsCAD dialog box opens. Click the **Activate Manually...**link. The Manual Activation dialog box opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/LicenseManager
3. Copy the full content of the Computer Hostid field.
4. With the Internet connected computer, log in to the Bricsys website. Make sure you log in with the Bricsys account containing your licenses.
5. Once you are logged in, click on your name in the menu above right. A page opens, with information linked to your account.
6. In the left menu choose Licenses and support contracts. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
7. Click the **Manage...** button of the license you want to activate.
8. In the **Host ID** field, paste the computer host ID of the computer you want to install BricsCAD on. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/PasteHostID.png})
9. Click the **Activate** button. A license file is generated and made available for download.
10. Click the **Download License File** button. The license file **BricsCADVxx.lic** is downloaded to your default download folder.
11. Copy the license file to the computer you want to install BricsCAD on.
12. In the Manual Activation dialog box, click the browse button (). Open the folder where you copied the **BricsCADVxx.lic** license file to and select the file. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18
13. Click the **OK** button. BricsCAD is launched.

Note: Manually activated licenses cannot be deactivated.

Figure 1.93

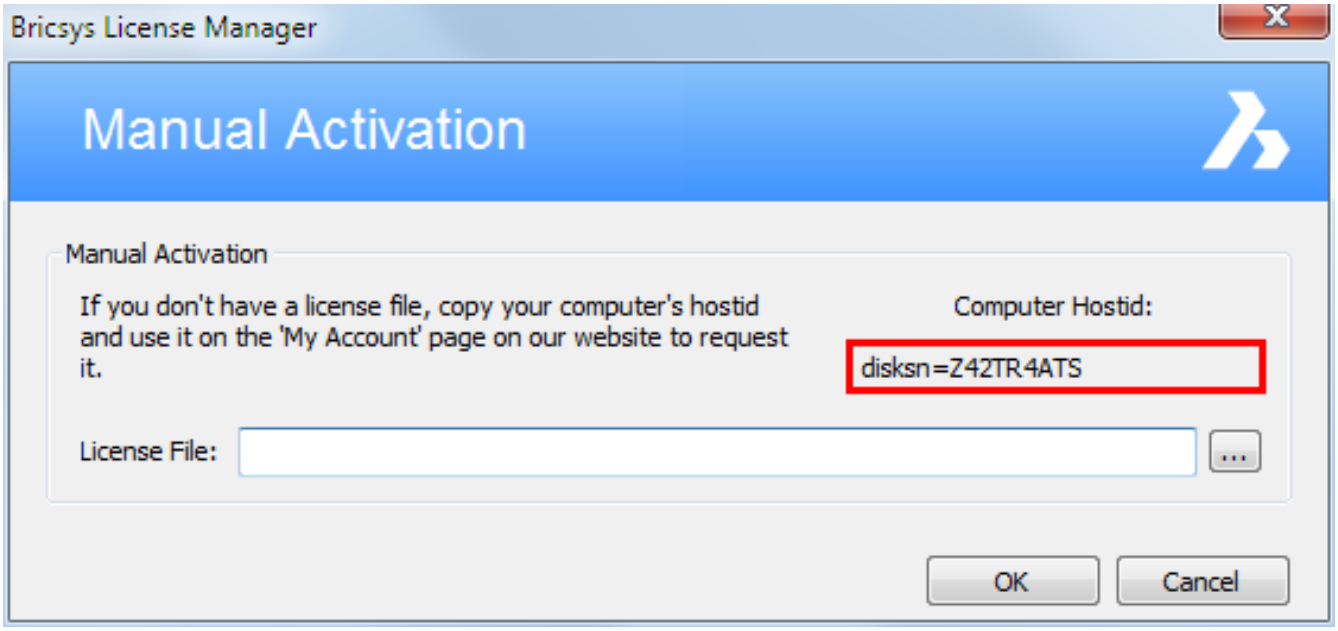


Figure 1.94

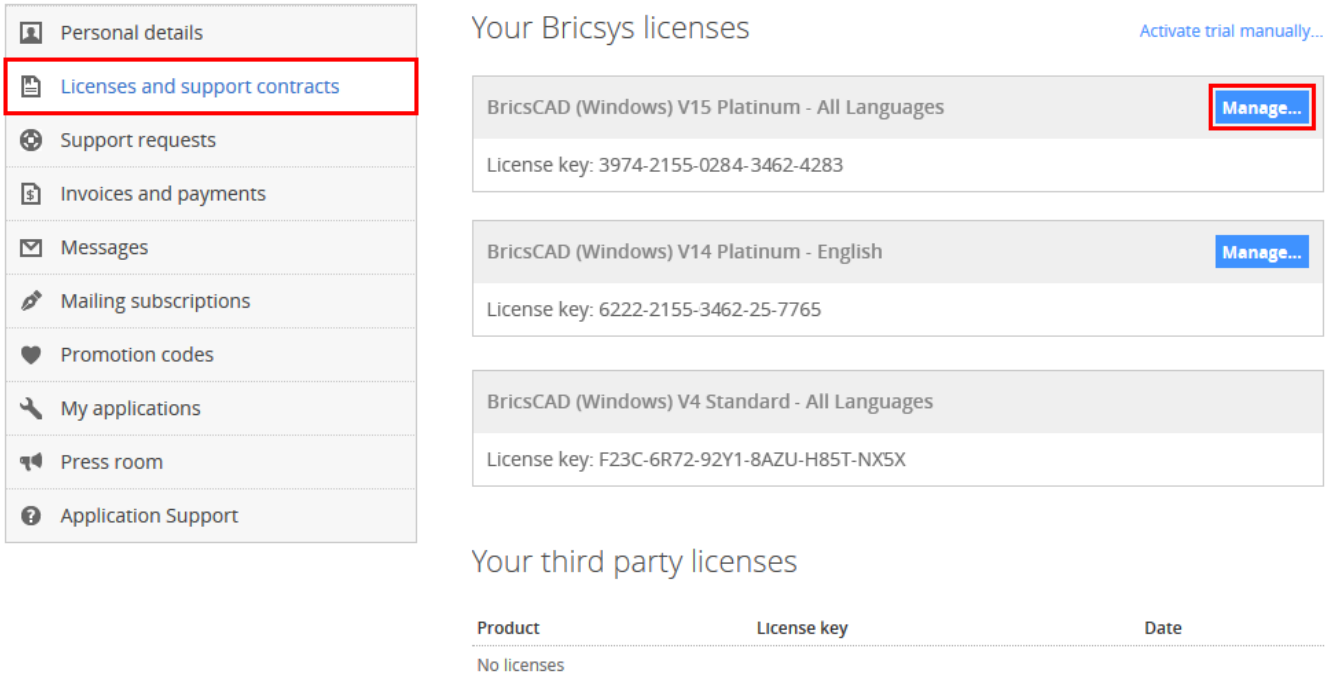


Figure 1.95

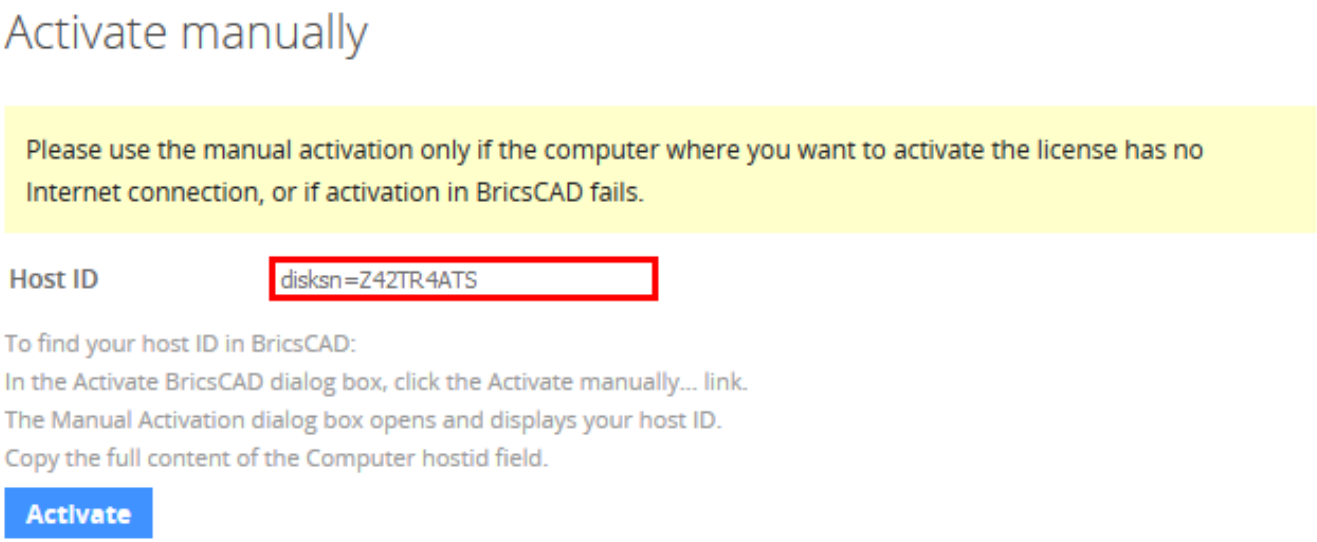
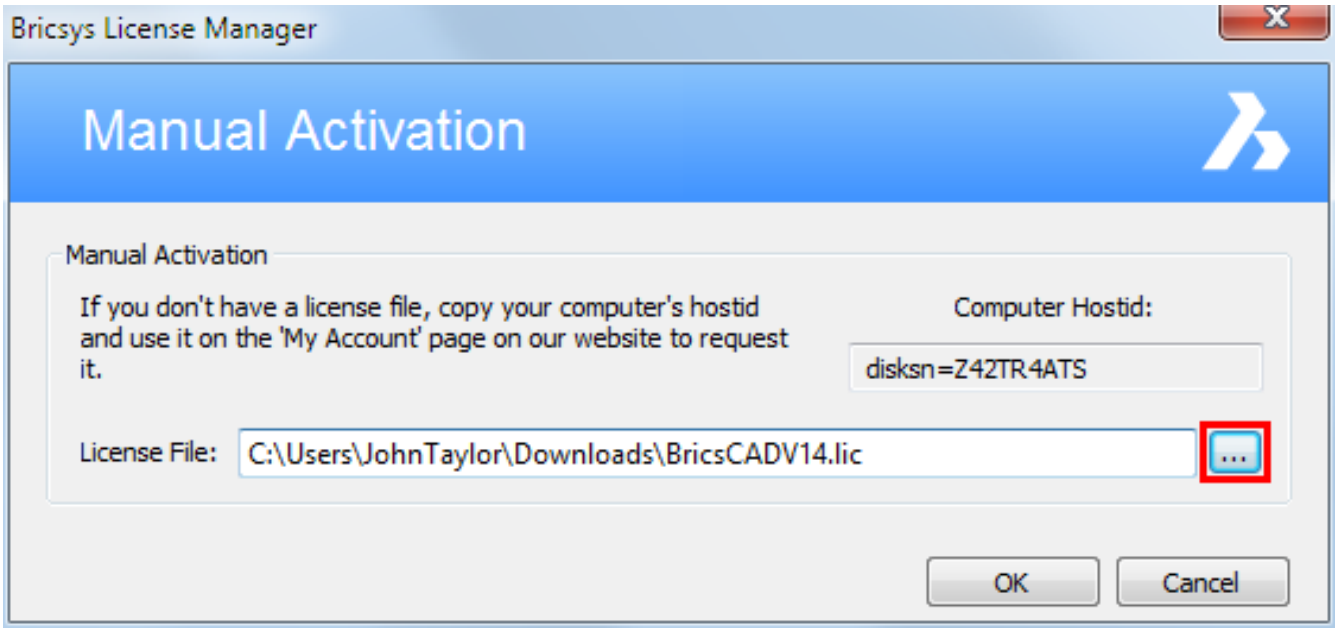


Figure 1.96



1.11.8 Activate a Network license manually

The BricsCAD license activation always requires an Internet connection. However, if your computer is not connected to the Internet, the license can be activated manually via another computer (which is connected to the Internet).

To activate a Network license manually

1. In the **Start** menu, click theLicense Activationshortcut. The**Activate Network Licensedialog box** opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Installation&Settings/ActivateNetworkLicense
2. Copy the full content of theComputer **Host ID**field.
3. With the Internet connected computer, go to theBricsys website.
4. In theSupportmenu, choose**My Account**(login required). Make sure you have the Bricsys account of your company which contains the Volume license. Contact your Bricsys account manager if needed. TheMy Accountpage opens.
5. In the left menu, choose**Licenses and support contracts**.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Installation&Settings/MyAccountLicensesActi

6. Click the**Activate**button of the license you want to activate.
7. In the Host ID field, paste the computer host ID of the computer you want to install BricsCAD on.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Installation&Settings/PasteHostID.png})

7. Click the**Activate**button. A license file is generated and made available for download.
8. Click the**Download License File**button. The license fileBricsCAD.licis downloaded to your default download folder.
9. Copy the license file to the server, in the Bricsys Network License Manager installation folder **(by default :C:\Program Files (x86)\Bricsys\Bricsys Network License Manager)**.

NOTE Manually activated licenses cannot be deactivated.

Figure 1.97

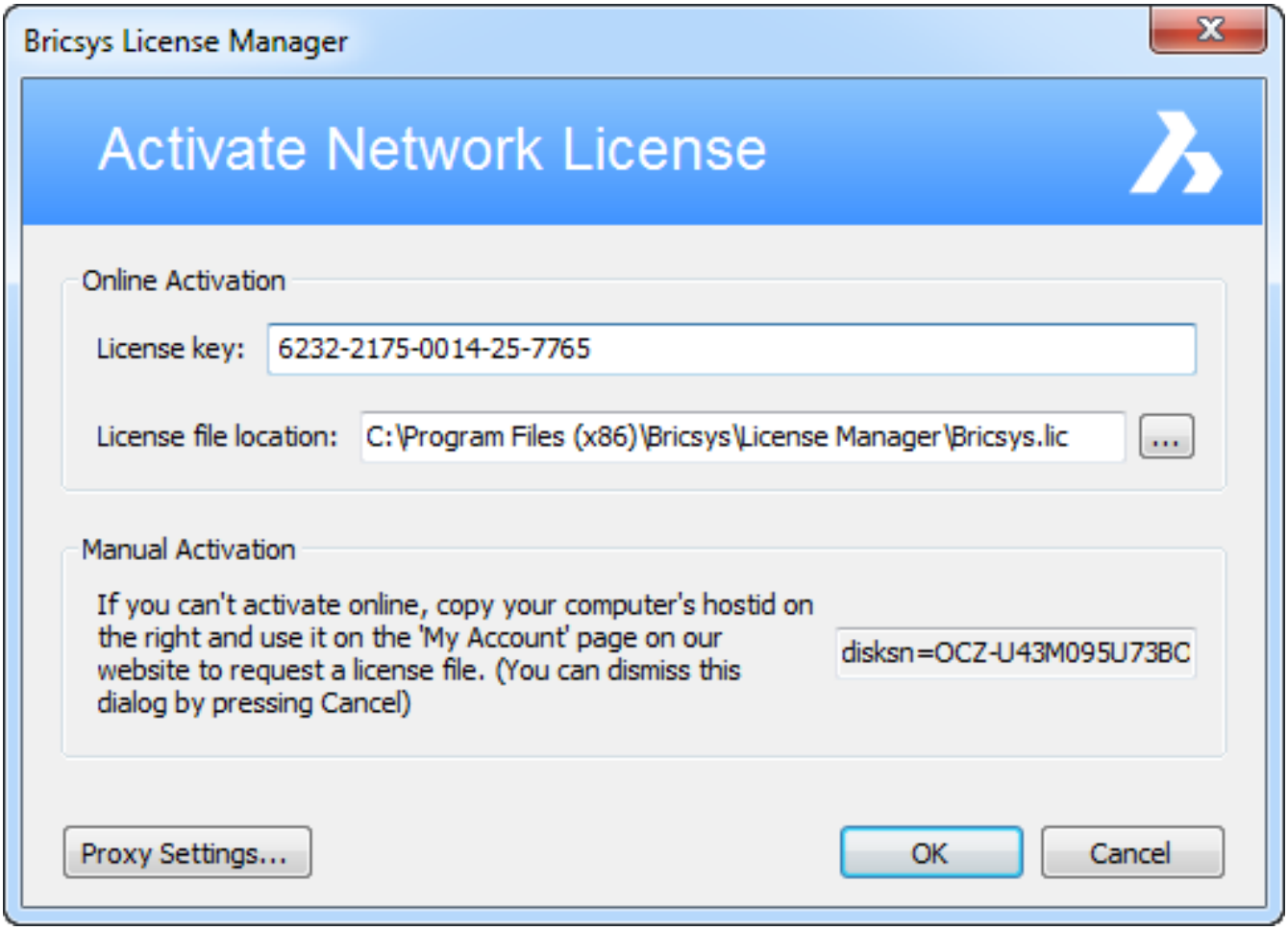


Figure 1.98

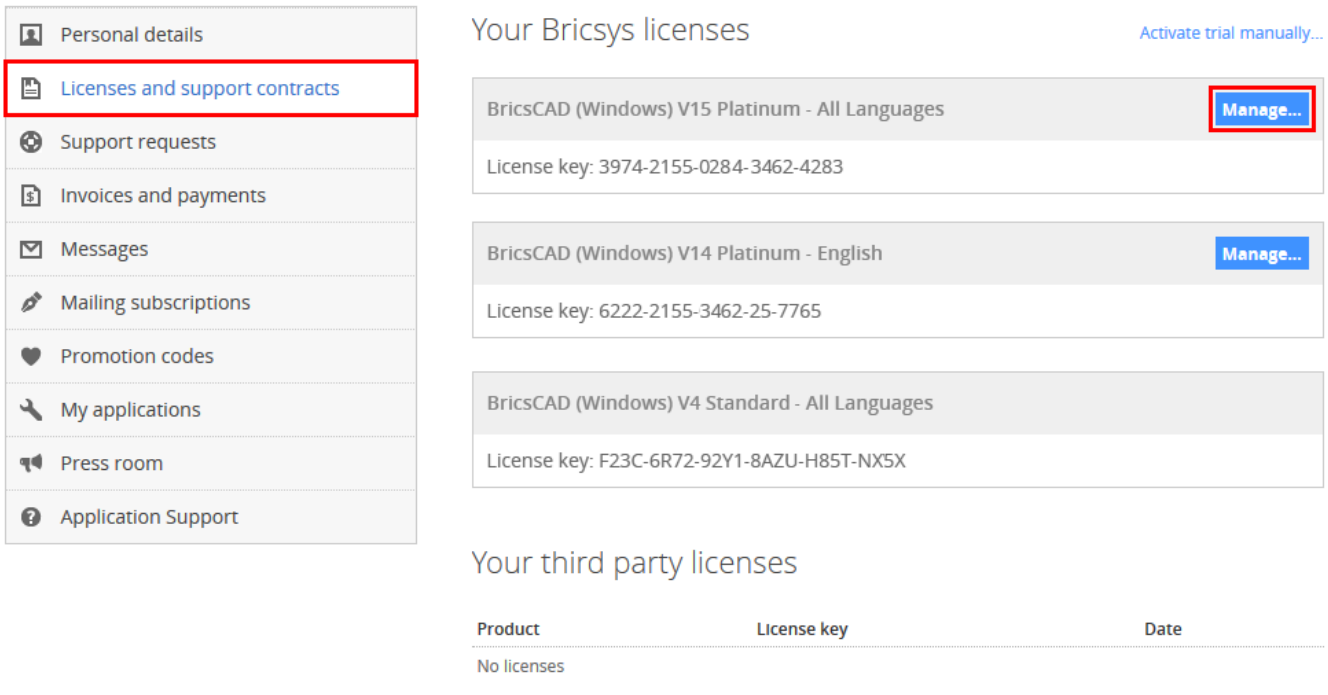
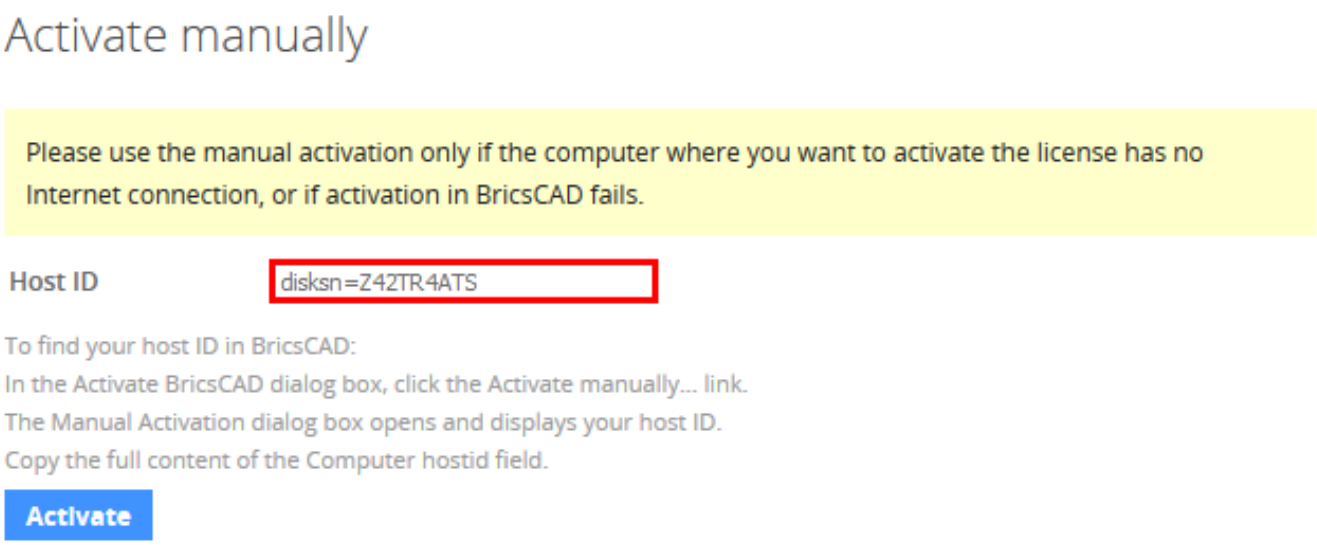


Figure 1.99


Activate manually
Please use the manual activation only if the computer where you want to activate the license has no Internet connection, or if activation in BricsCAD fails.
Host ID
disksn=Z42TR4ATS
To find your host ID in BricsCAD:
In the Activate BricsCAD dialog box, click the Activate manually... link.
The Manual Activation dialog box opens and displays your host ID.
Copy the full content of the Computer hostid field.
Activate

1.11.9 Trial mode

BricsCAD can run in trial mode for a 30 days period. After this period, a valid commercial license needs to be activated.

To activate BricsCAD in trial mode

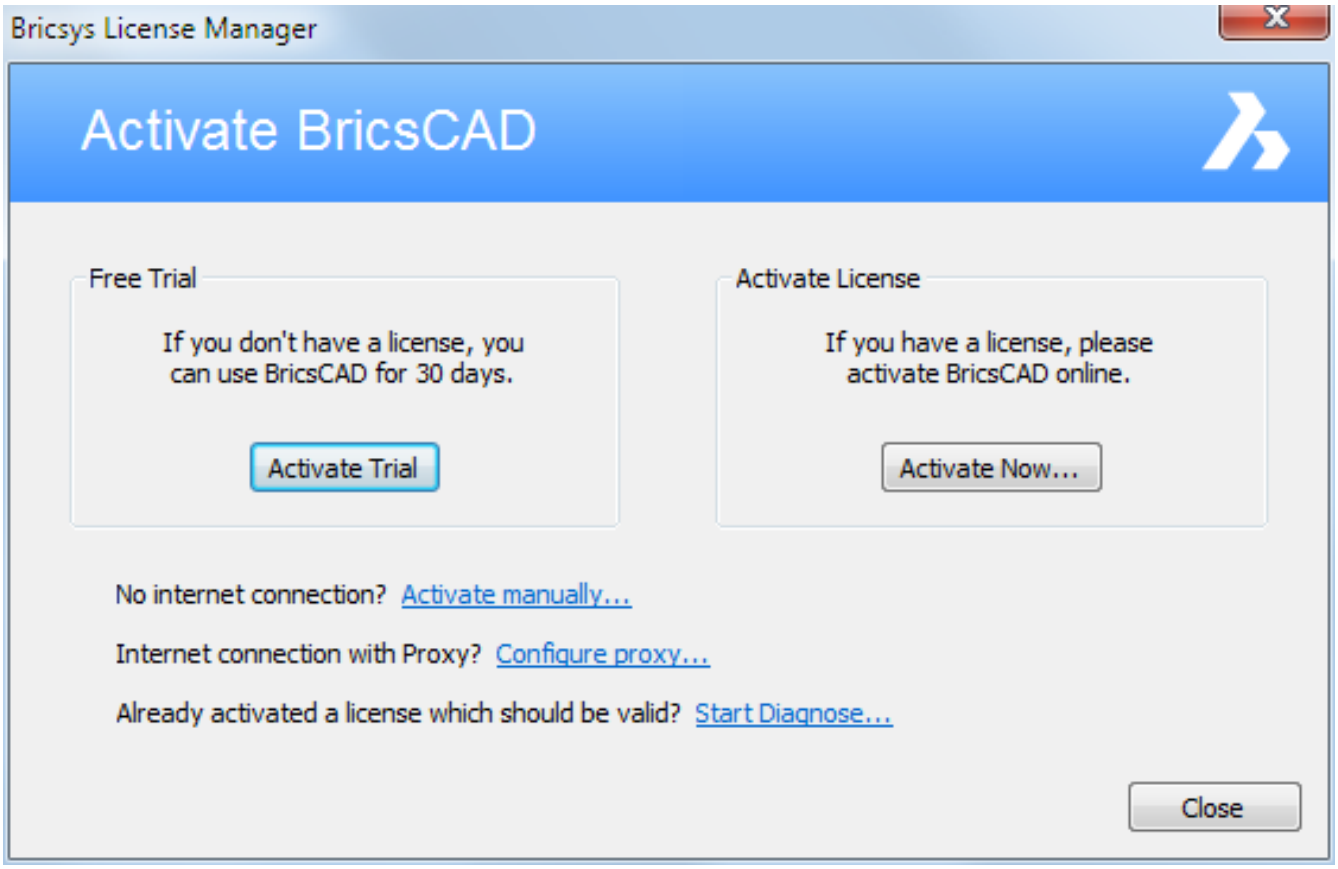
If your computer is not connected to the Internet, activation can be done manually via another computer (which is connected to the Internet). See Activate a trial manually (1.11.6).

1. Launch the BricsCAD executable, e.g. double click the BricsCAD desktop icon. The Bricsys License Manager - Activate BricsCAD dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/sour

2. If your computer connects to the Internet through a proxy server, click the **Configure proxy...** link to configure the proxy settings (1.11.4).

3. Click the Activate Trial button.

Figure 1.100

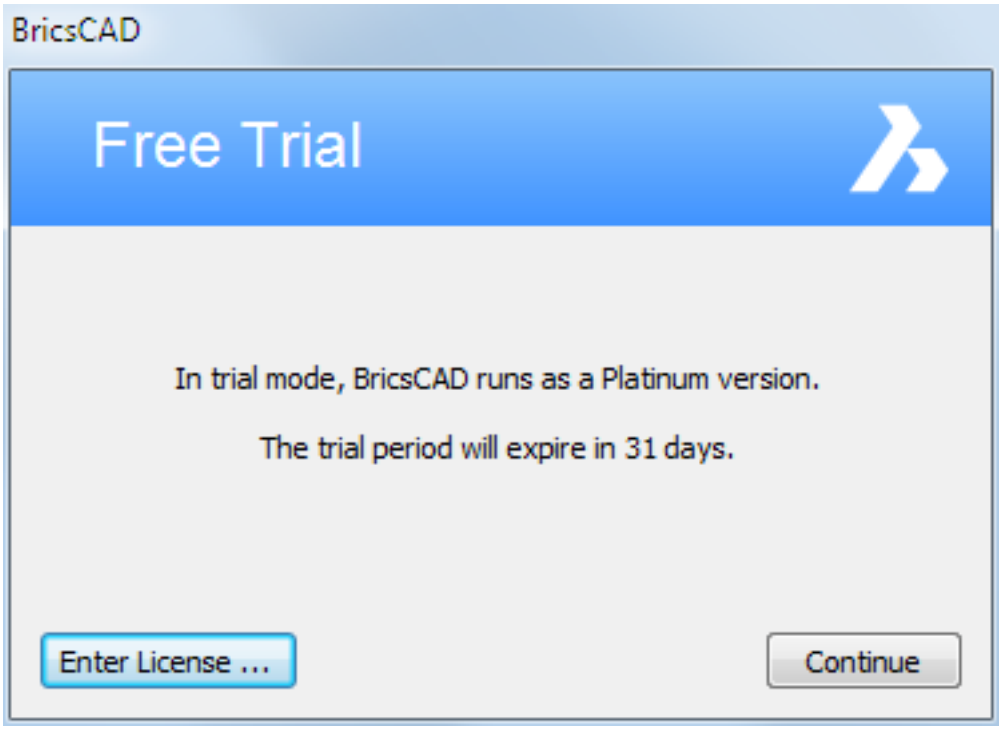


Running BricsCAD in trial mode

Each time you start BricsCAD in trial mode the BricsCAD - Free Trial dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbr

1. Click the **Continue** button to open BricsCAD in trial mode.
2. Click the **Enter License...** button to activate a Single or Volume license (1.11.10)or to use a Network license (1.11.11).

Figure 1.101



Feature levels

In trial mode BricsCAD runs as full Platinum version without any feature restrictions. The RunAsLevel preference setting allows to run BricsCAD in a lower license level mode, e.g. a Platinum version can be run as a Pro or a Classic, a Pro can be run as a Classic. This facilitates evaluation of the features available for each license level.

1. In BricsCAD, in the Settings dialog box, go to the RunAsLevel preference or, on the command line, type ”RunAsLevel”.
2. Enter the value for the level you want to try: 0 = Classic, 1 = Pro, 2 = Platinum (default).
3. Restart BricsCAD to apply the changes.

Tutorial: Run your BricsCAD trial as Classic, Pro or Platinum Click here (1.1.1) to watch

Related topics

Configure the proxy settings (1.11.4) Activate a Single or Volume license (1.11.10) Use a Network license (1.11.11)

1.11.10 Single or Volume license

To activate a Single or Volume license

If your computer is not connected to the Internet, activation can be done manually via another computer (which is connected to the Internet). See Activate a Single or Volume license manually (1.11.7).

1. Do one of the following:
   - The first time BricsCAD is started - In the Activate BricsCAD dialog, click the **Activate Now... button.**
   - When starting BricsCAD in trial mode - In the Free Trial dialog, click the **Enter License...** button.
   - When replacing or upgrading an existing license - In the Help menu, choose **BricsCAD License...** . The Licensing Information dialog opens. Click the **Modify...** button. The Activate BricsCAD dialog opens.
2. Check the **Single User or Volume License Key** option.
3. Enter or copy/paste your license key in the **License Key** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
4. If your computer connects to the Internet through a proxy server, click the **Proxy Settings...** button to configure the proxy settings (1.11.4).
5. Click the **OK** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/Suc

Figure 1.102

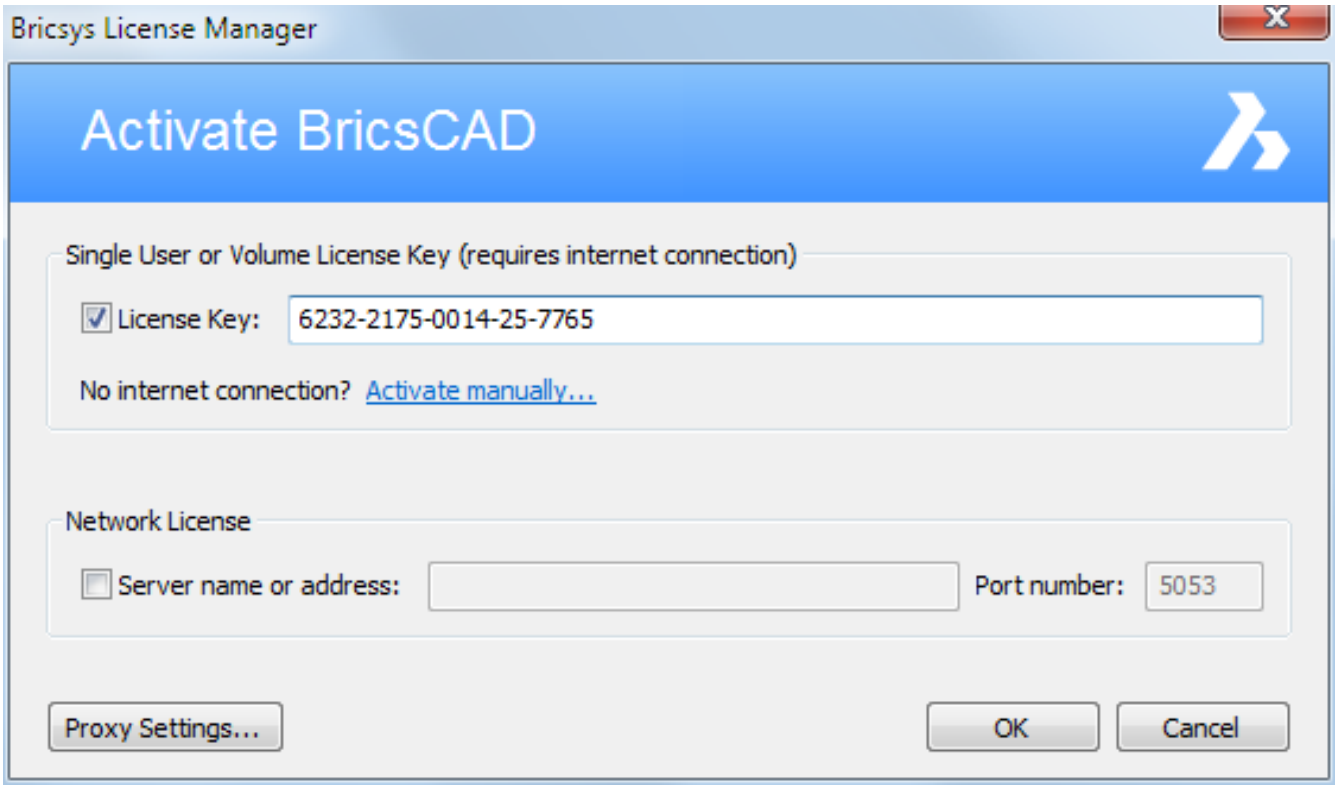
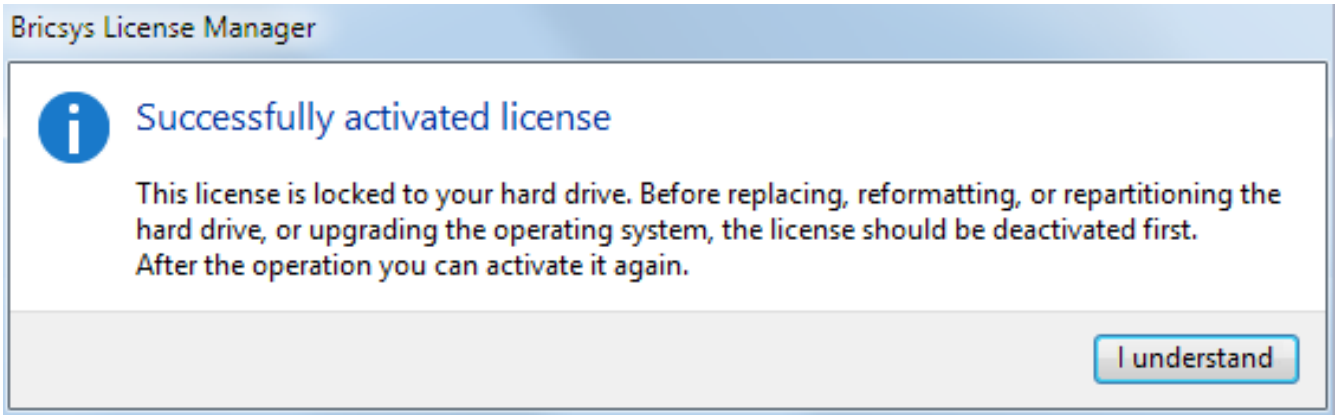


Figure 1.103



Related topics

Configure the proxy settings (1.11.4)

1.11.11 Network license

The network administrator first needs to set up a network license server (1.11.14). The license server must be started and the Network license must be activated on the server.

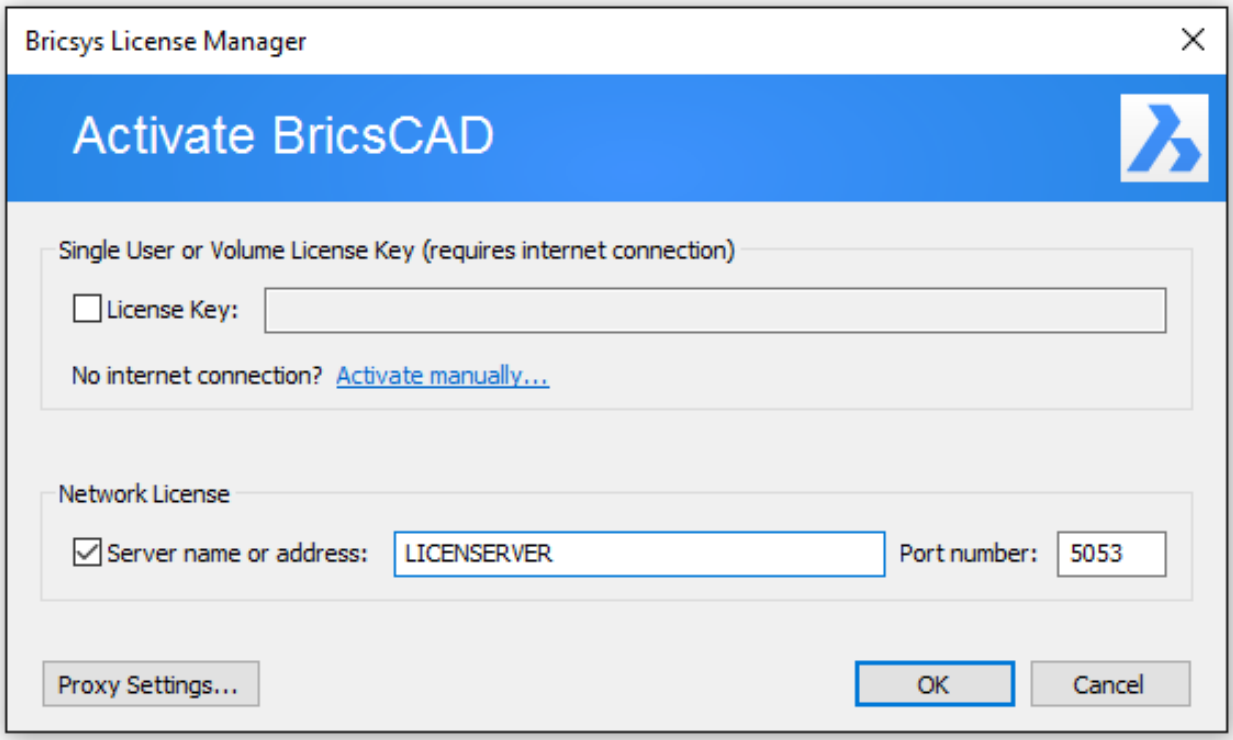
To use a network license on a client computer

Once the Network license is activated on the license server, the Network license can be used on client computers.

1. Do one of the following:
   - The first time BricsCAD is started - In the Activate BricsCAD dialog, click the **Activate Now...** button.
   - When starting BricsCAD in trial mode - In the Free Trial dialog, click the **Enter License...** button.
   - When replacing or upgrading an existing license - In the Help menu, choose **BricsCAD License...** . The Licensing Information dialog opens. Click the **Modify...** button

   The Activate BricsCAD dialog opens.
2. Check the **Network License** option.
3. Type the host name or IP address of the license server in the **Server name or address** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/NetworkLicense.png})
4. Optional) Type the port number in the **Port Number** field.
5. If your computer connects to the license server through a proxy server, click the **Proxy Settings...** button to configure the proxy settings for a network license (1.11.4).
6. Click the **OK** button.

Figure 1.104



To roam a network license on a client computer

A network license can be roamed in order to use BricsCAD without connection to the license server (work outside of the office, at home, ...). When a network license is used on the client computer, the Roaming... button is displayed in the In the Licensing Information dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/sou

1. Click the **Roaming...** button. The Roaming dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
2. Enter the number of days you want to checkout the license for roaming, then click the **Checkout** button
3. (Optional) Check the check box if you want to automatically refresh the roaming period whenever your computer is connected to the server.
4. Re-start BricsCAD to apply the changes.

Tutorial: How to roam with a network license Click here (1.1.4) to watch

Figure 1.105

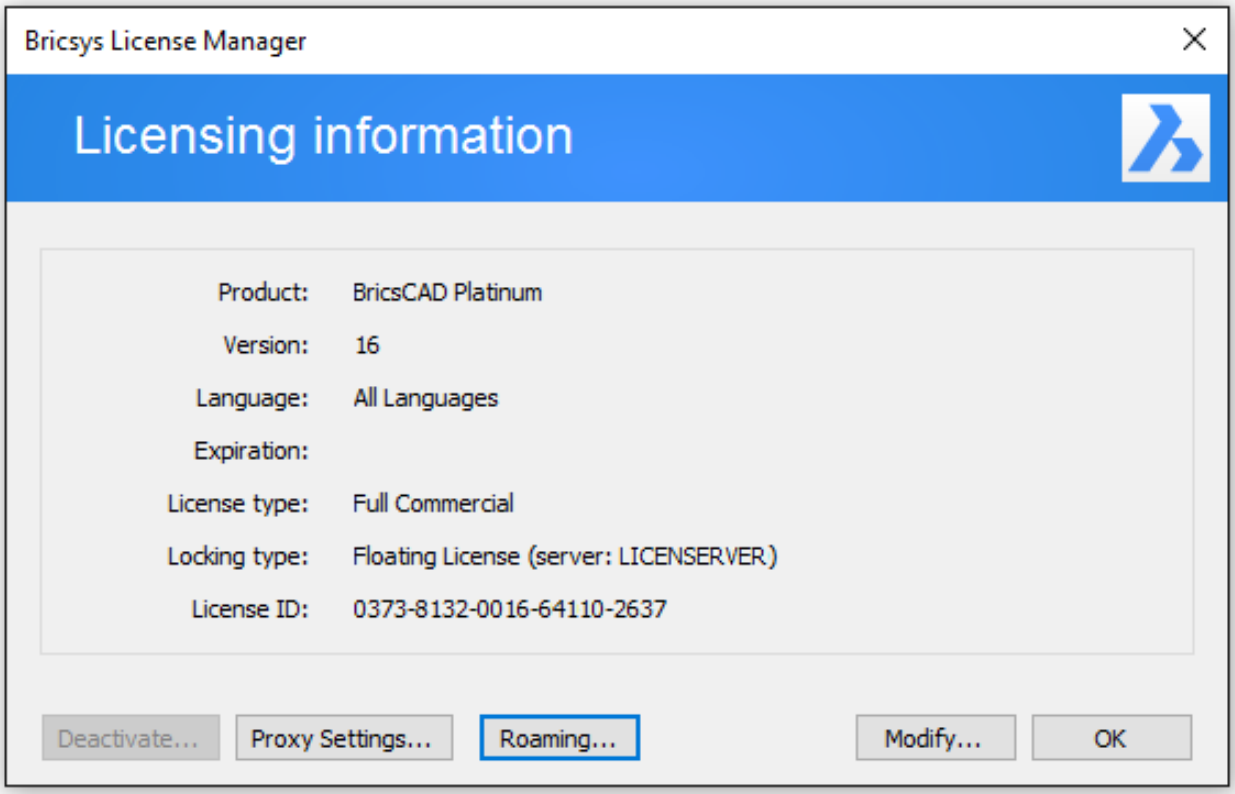
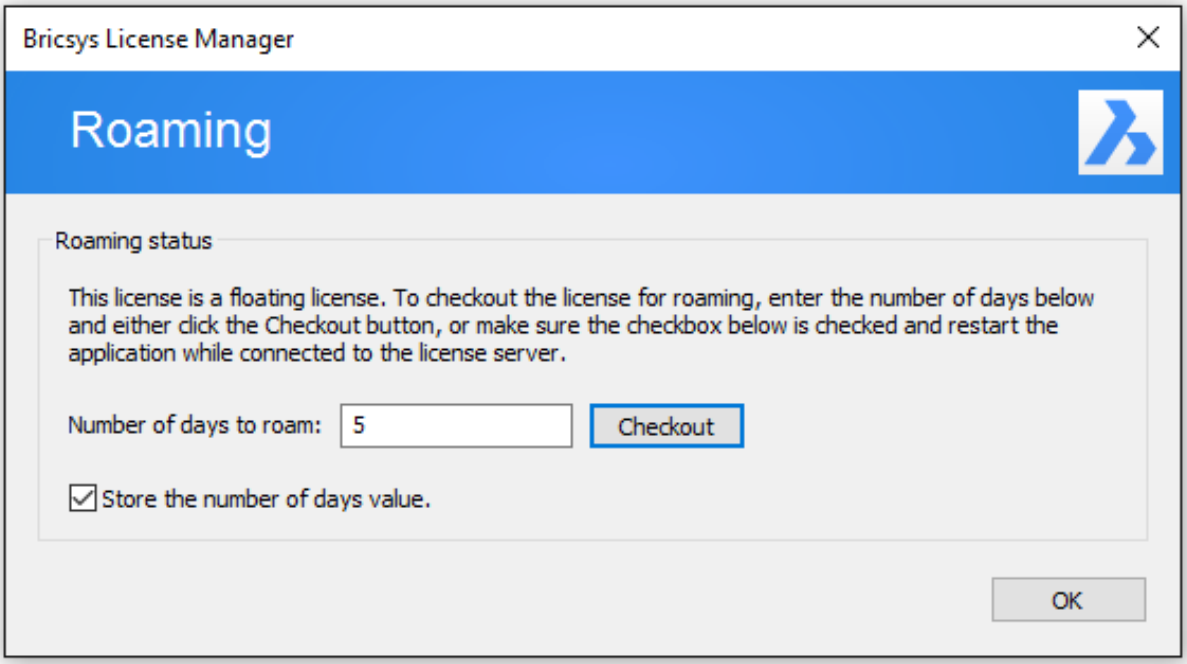


Figure 1.106

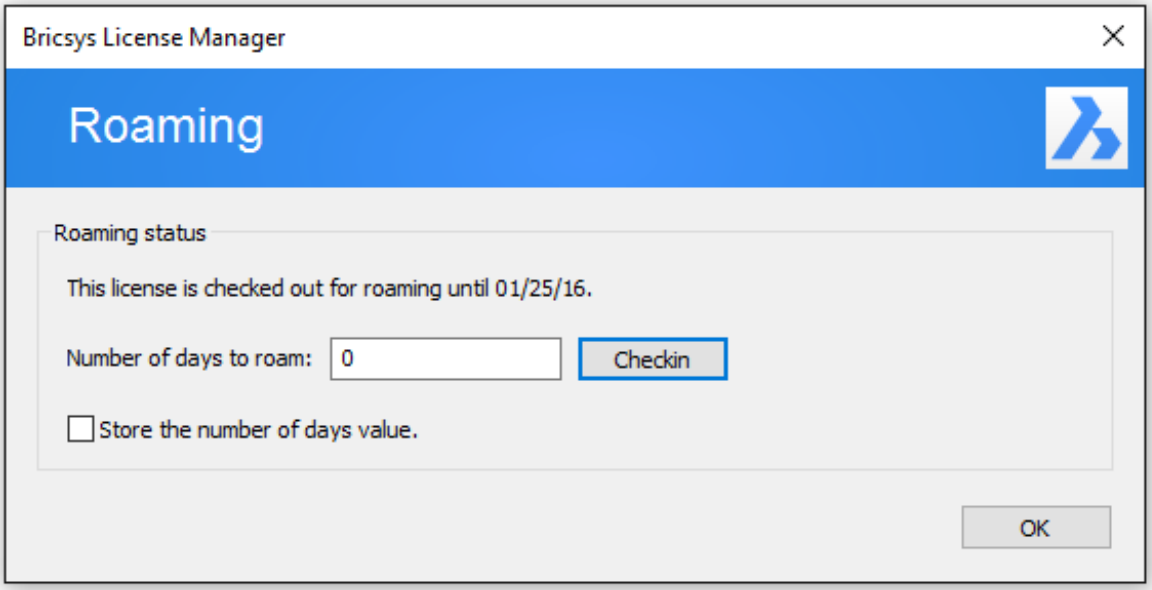


To return a roamed license to the server

If you are connected to the server, a roamed license can be returned to the server.

1. In the Licensing Information dialog, click the **Roaming...** button. The Roaming dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/RoamingCheckin.png})
2. Click the **Checkin** button
3. Re-start BricsCAD to apply the changes.

Figure 1.107



Related topics

Configure the proxy settings (1.11.4) Network license server setup (1.11.14) Roaming licenses (server side) (1.11.15)

1.11.12 Modification

The BricsCAD license modification always requires an Internet connection.

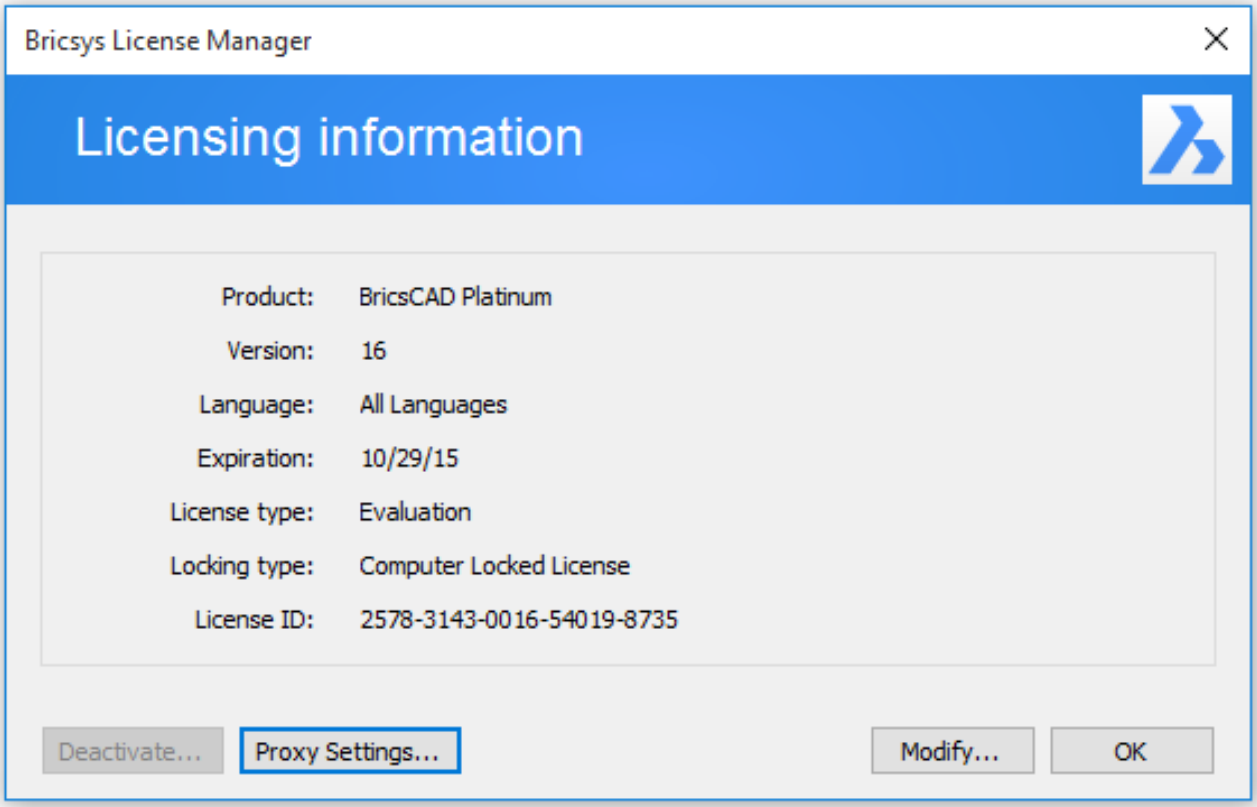
To modify a BricsCAD license

Use this procedure to:

- Upgrade a license (e.g. from Single Classic to Single Platinum). The old key is no longer valid then.
- Replace a license (e.g. a Single Classic by a Network Pro). The old key is deactivated automatically and is available to be activated on another computer.

1. In the Help menu, choose **BricsCAD License...** . The Licensing information dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/BricsCADLicenseDialog.png})
2. Click the **Modify...** button to activate another Single or Volume license (1.11.10), or to use another a Network license (1.11.11).

Figure 1.108



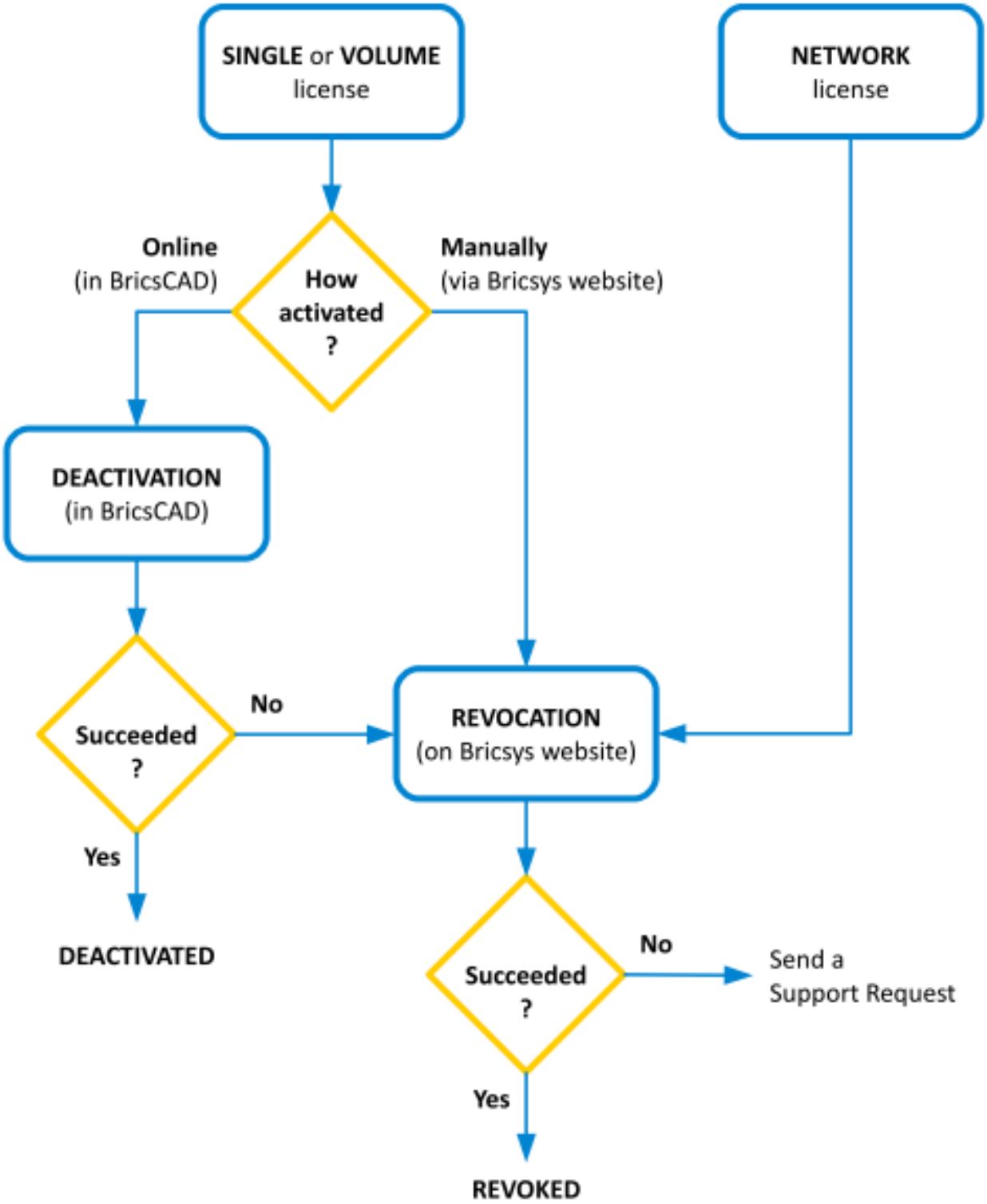
Related topics

Configure the proxy settings (1.11.4)

1.11.13 Deactivation

The BricsCAD license deactivation always requires an Internet connection. Licenses cannot be deactivated manually via another computer (which is connected to the Internet). Single and Volume licenses manually activated, Network and trial licenses cannot be deactivated. The chart below explains how a BricsCAD license can be deactivated.
If the deactivation is not possible, the license can be revoked. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU

Figure 1.109


SINGLE or VOLUME license
NETWORK license
Online (in BricsCAD)
How activated ?
Manually (via Bricsys website)
DEACTIVATION (in BricsCAD)
Succeeded ?
No
REVOCATION (on Bricsys website)
Yes
DEACTIVATED
Succeeded ?
No
Send a Support Request
Yes
REVOKED

To deactivate a license

Use this procedure to:

- Migrate a license to another computer.
- Remove BricsCAD permanently from a computer.
- Be able to reactivate BricsCAD after installing a new hard disk, changing partitions or upgrading the operating system.

1. In the Help menu, choose **License Manager...** . The Bricsys License Manager dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/LicenseManagerDialog.png})

2. Click **Manage License >** in the top right corner of the dialog. The Licensing information dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/LicensingInformationDialo

3. If your computer connects to the Internet through a proxy server, click the **Proxy Settings...** button to configure the proxy settings (1.11.4).

4. Click the **Deactivate...** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source

5. Click the **Yes** button on the confirmation dialog box to deactivate the license. The license is deactivated now and is available to be activated on another computer.

Figure 1.110

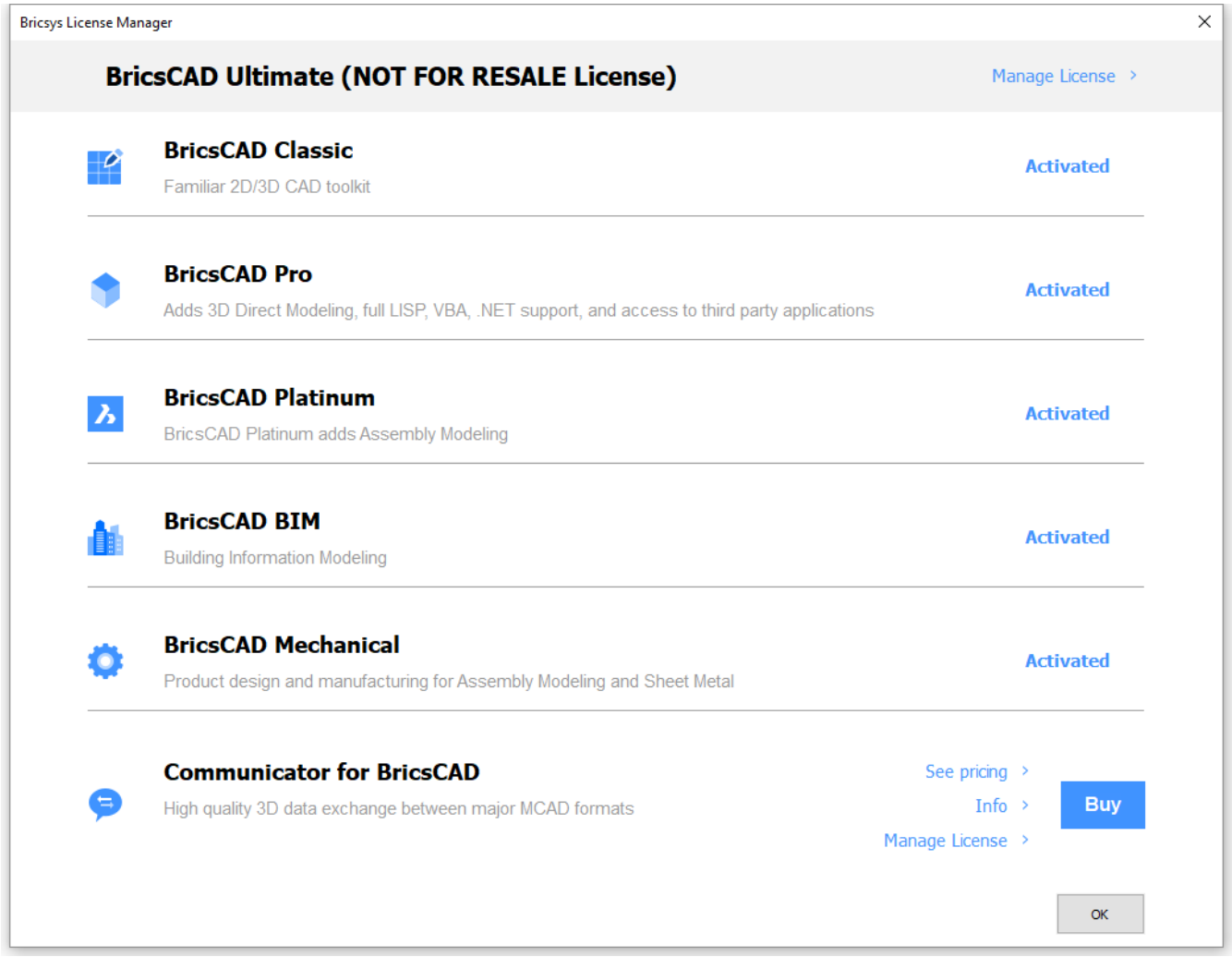


Figure 1.111

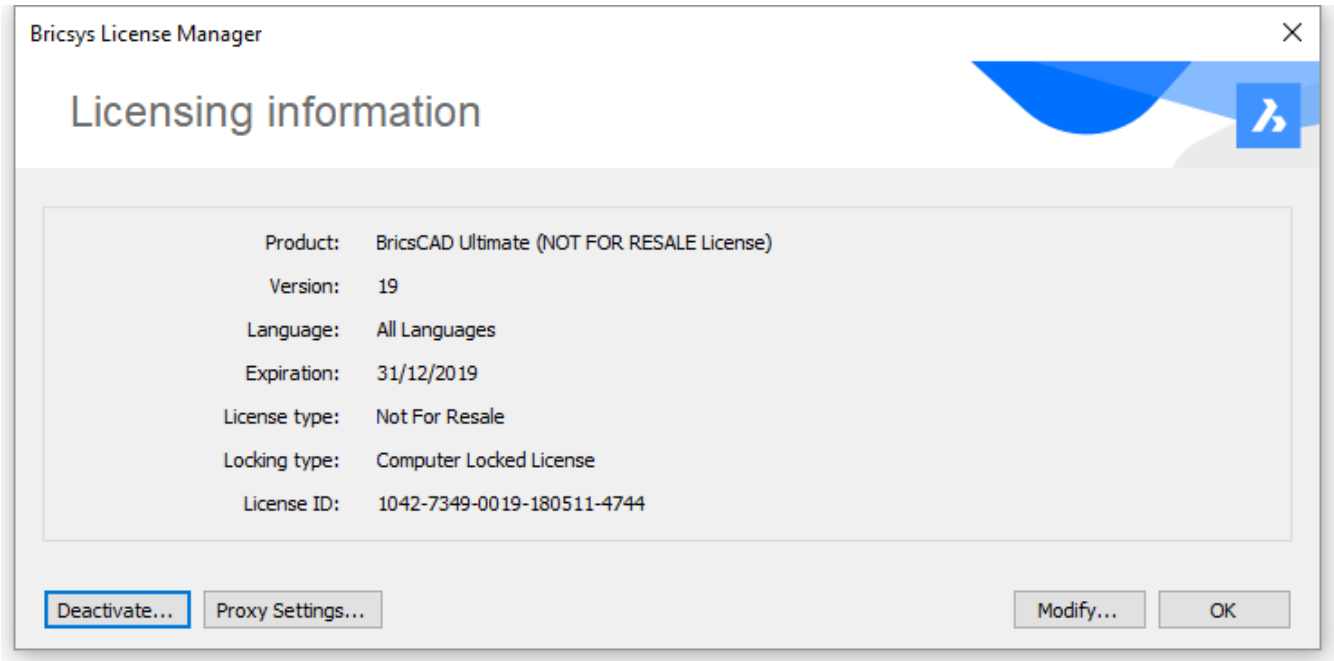
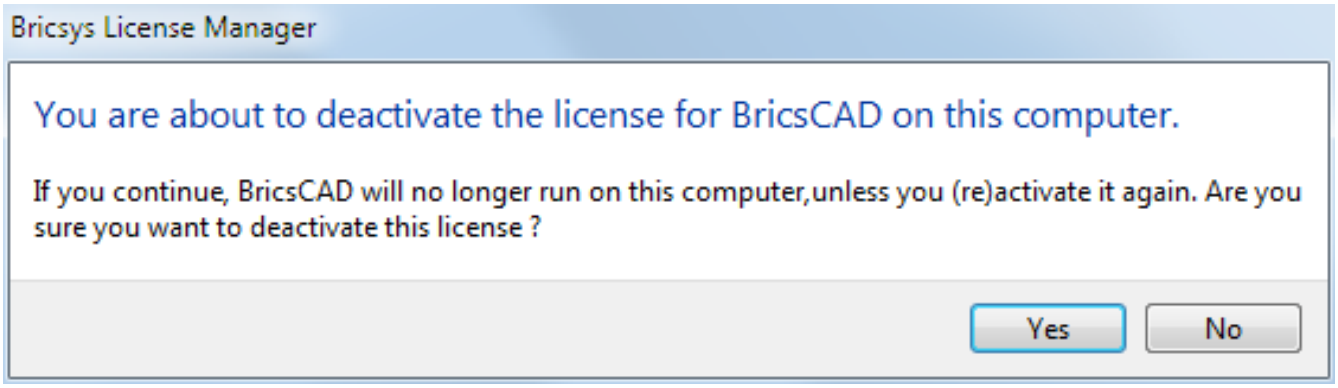


Figure 1.112



To revoke a license

In last resort, only if deactivation is not possible. Use this procedure to:

- Deactivate a Single or Volume license manually activated, or a Network license .
- Be able to reactivate BricsCAD after a hard disk crash.
- Be able to reactivate BricsCAD after installing a new hard disk, changing partitions or upgrading the operating system.
- Migrate a license to another computer.
- Remove BricsCAD permanently from a computer.

1. Log in to the Bricsys website. Make sure you log in with the Bricsys account containing your licenses.
2. Once you are logged in, click on your name in the menu above right. A page opens, with informations linked to your account.
3. In the left menu choose Licenses and support contract. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
All your active licenses are displayed.
4. Hover over the license key of the license you want to revoke and click on the pencil icon. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/InstallationSettings/LicenseDetailsIcon.png})
5. Click the **X** icon to revoke the the license. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/

Figure 1.113

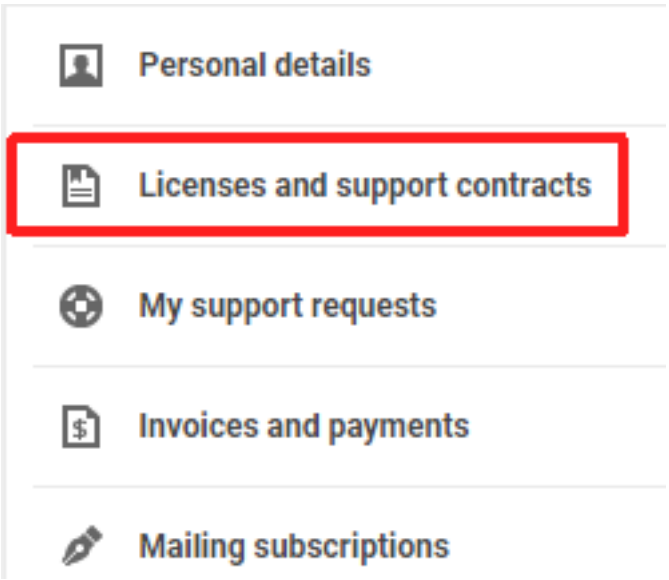


Figure 1.114

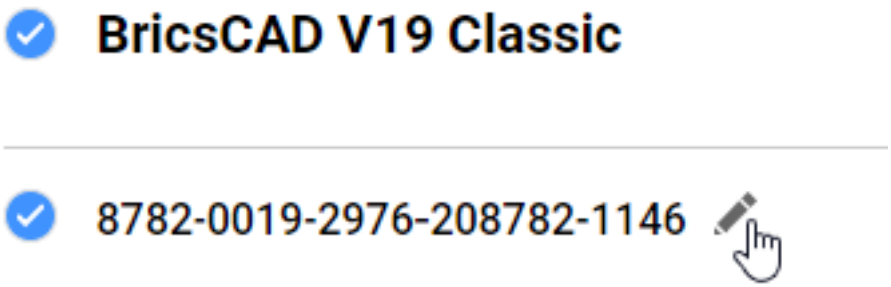


Figure 1.115



Related topics

Configure the proxy settings (1.11.4)

1.11.14 Network license server

To make a Network license available in BricsCAD on client computers, a network license server from Reprise Software needs to be installed first by the network administrator. The Bricsys Network License Manager is a separate program which will install the server. After starting the network license server, it can be completely configured by using a web interface. Just like BricsCAD needs a license file to run, the server also needs a Network license file that can be activated online or manually. A Network license can only be activated once and it is locked to the server. Once the Network license is activated and the server is started, it is possible to use the Network license in BricsCAD, on a client computer. **Note:** Reprise Software doesn't offer a network license server on Mac. However, it is possible to install a Windows or Linux network license server in order to serve licenses on Mac client computers.

Related topics

Windows network license server setup (1.11.15) Linux network license server setup (1.11.16) Server administration (1.11.17) Use a Network license (1.11.11) RLM License Administration Manual RLM

1.11.15 Windows network license server setup

Tutorial: BricsCAD Network License Setup Click here to watch

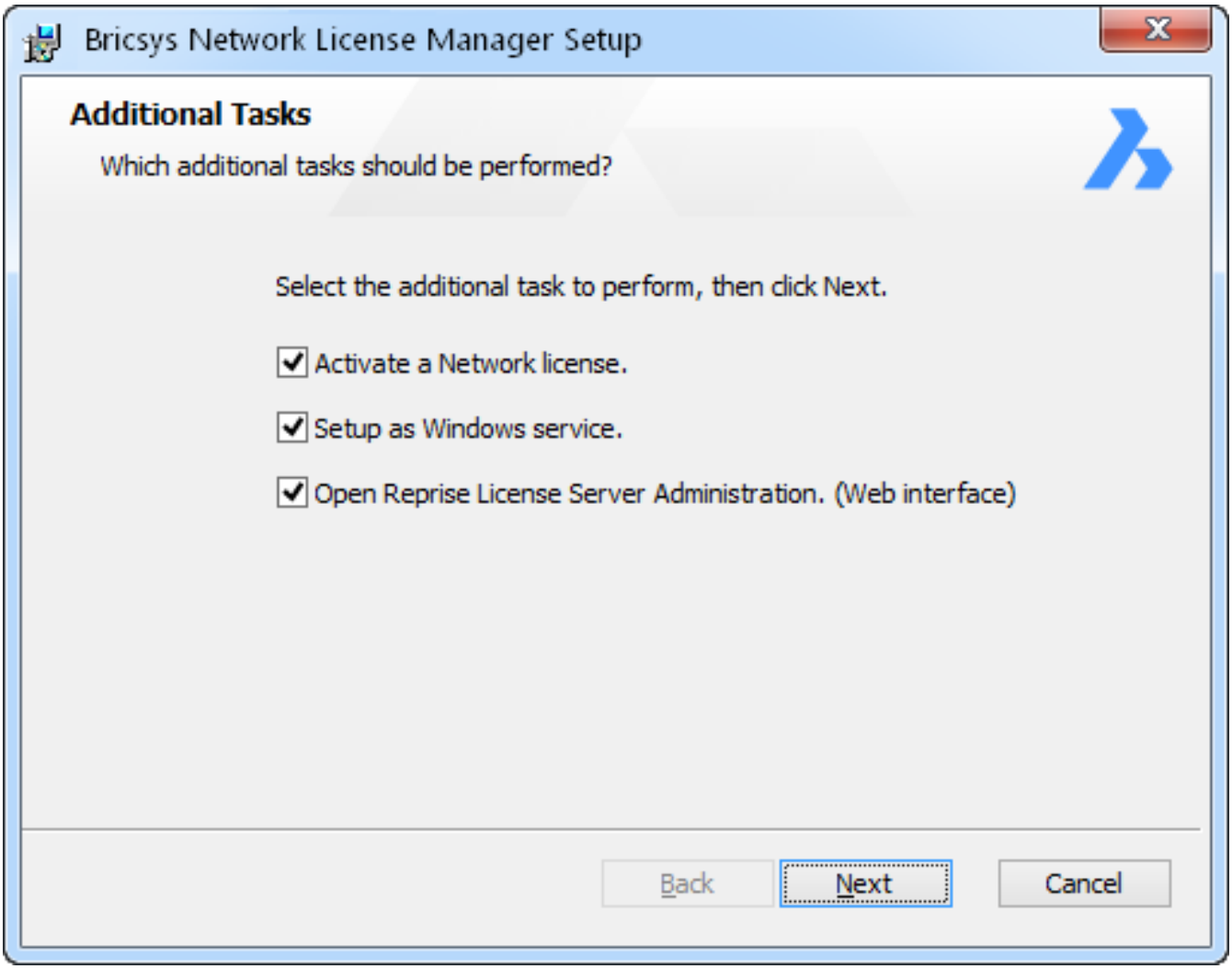
Install the Bricsys Network License Manager

Download and install the Bricsys Network License Manager on the computer that will be used as network license server. Download Bricsys Network License Manager (Windows) Double-click the installation MSI file in order to proceed with the installation:

1. Accept the terms of End User License Agreement.
2. Accept the Destination folder.
3. Click the Install button.

The Additional Tasks dialog displays 3 options. If checked, these options will be performed after the installation. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/AdditionalTasks.png})

Figure 1.116


Bricsys Network License Manager Setup
Additional Tasks
Which additional tasks should be performed?
Select the additional task to perform, then click Next.
Activate a Network license.
Setup as Windows service.
Open Reprise License Server Administration. (Web interface)
Back
Next
Cancel

Activate a Network license After the installation, opens the Bricsys Network License Manager , asking you to activate a Network license. You can later open the Bricsys Network License Manager by clicking the License activation shortcut in the Start menu.

Setup as Windows service Each time you start Windows, the license server will be automatically started as a service.

Open Reprise License Server Administration (Web interface) After the installation, open the Reprise License Server Administration web interface where you can manage all about the network license. You can later open the Reprise License Server Administration by clicking the License Server Administration shortcut in the Start menu. **Note:** By default, these 3 options are disabled when installing via the command line. See Installation from the command line (1.11.15).

New installation

In the Additional Tasks dialog, all options should be checked.

To install a new network license on an existing license server

1. In the Start menu, click the License activation shortcut to open the Bricsys License Manager .
2. In the Bricsys License Manager , enter the new network license, then click the OK button.
3. In the Start menu, click the License server administration shortcut to open the Reprise License Server Administration .
4. In the Reprise License Server Administration , click the Reread/restart servers option in the left menu.
5. Click the REREAD/RESTART button.

To upgrade the Bricsys Network License Manager

Download and install the Bricsys Network License Manager on the computer is used as network license server. In the Additional Tasks dialog, all options should be unchecked. The installation restarts the license server.

Installation from the command line

The Network License Manager can be installed from the command line. By default, options Activate a Network license , Setup as Windows service and Open Reprise License Server Administration (Web interface) are disabled when installing via the command line. To enable these options, use the respective parameters ACTIVATELICENSE , INSTALLSERVICE and OPENWEBINTERFACE . For example, **msiexec /qb /i Bricsys-NetworkLicenseManager.msi ACTIVATELICENSE=1 INSTALLSERVICE=1** will just show the activation dialog and install as a service without showing the web interface.

NOTES

- For more options, type: **msiexec /?** at the command prompt**.**
- On the Microsoft website, you can find more informationabout the **msiexec** command.

Activate the Network license

The Network license must be activated on the license server. If your computer is not connected to the Internet, activation can be done manually via another computer (which is connected to the Internet). See Activate a Network license manually (1.11.8).

1. In the Start menu, click the **License Activation** shortcut. The Activate Network License dialog box opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/ActivateNetworkLicense
2. In the License key field, enter your Network license.

Figure 1.117

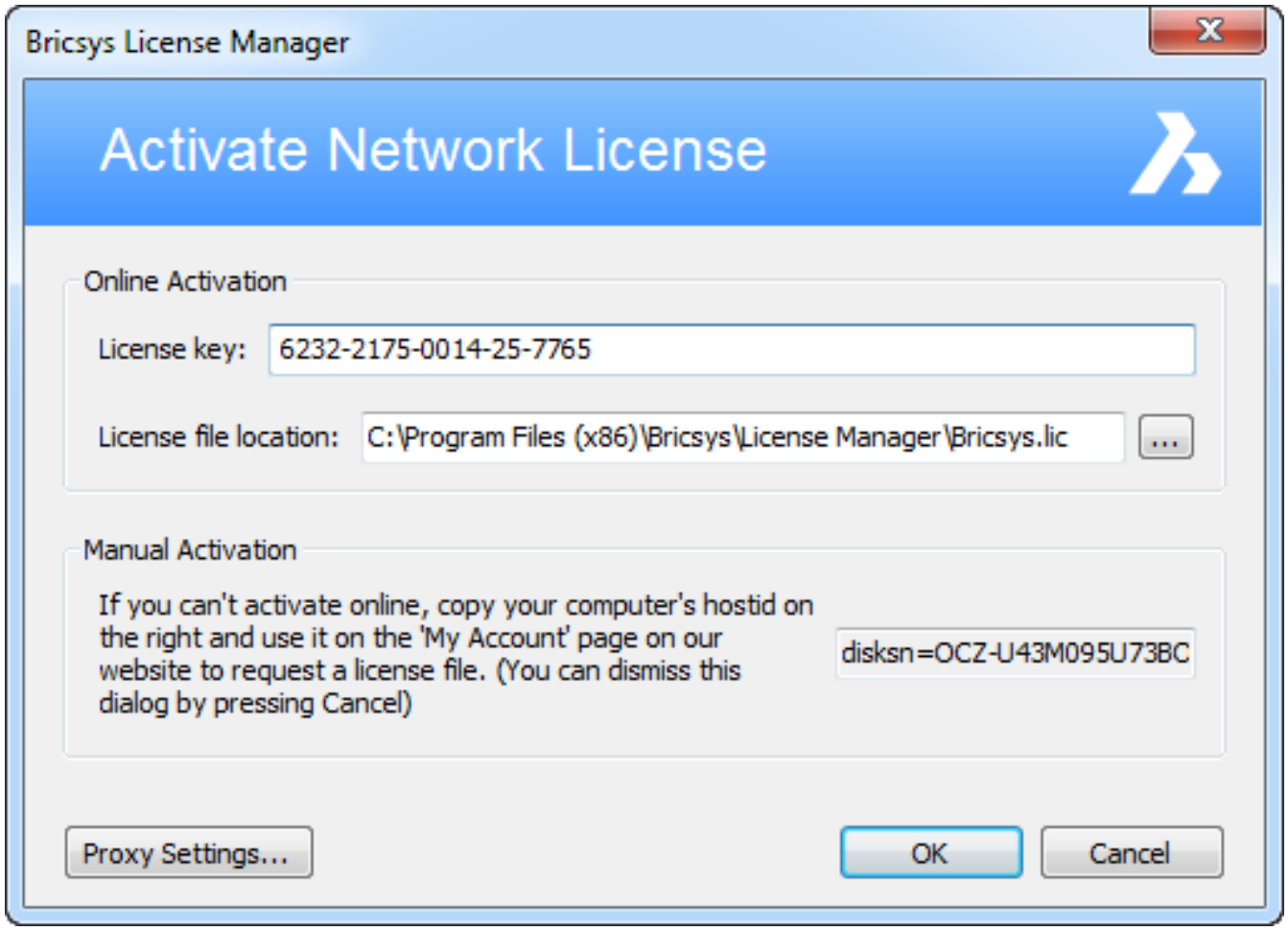
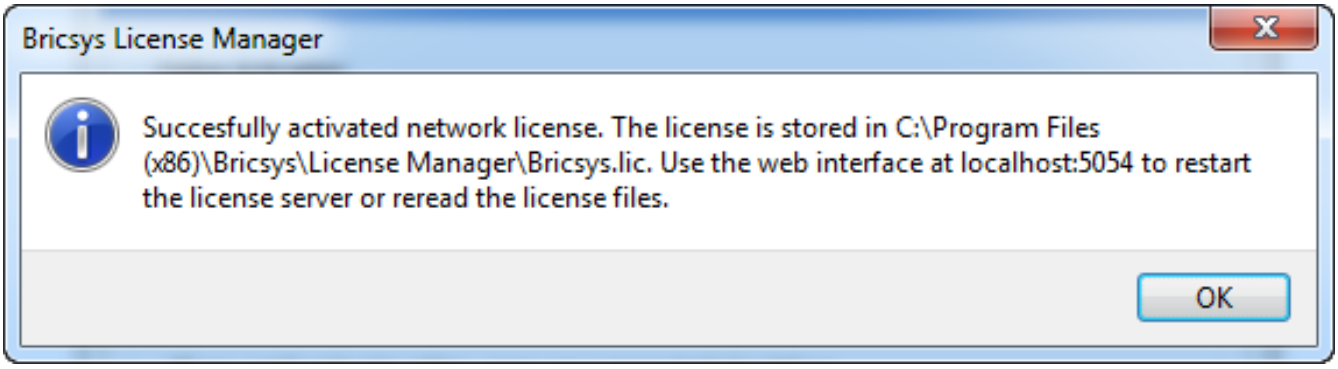

Bricsys License Manager
Activate Network License
Online Activation
License key:
6232-2175-0014-25-7765
License file location:
C:\Program Files (x86)\Bricsys\License Manager\Bricsys.lic
...
Manual Activation
If you can't activate online, copy your computer's hostid on the right and use it on the 'My Account' page on our website to request a license file. (You can dismiss this dialog by pressing Cancel)
disksn=OCZ-U43M095U73BC
Proxy Settings...
OK
Cancel

Figure 1.118



3. In the License file location field, the location of the license file must be the Bricsys Network License Manager installation folder.
4. Click the **OK** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/Net

Note: Network licenses cannot be deactivated.

Start the Network license server

The Network license server is automatically started/restarted after the installation. However, you can start it manually if necessary. In the Start menu, click the **License Server** shortcut. The server is now started and can be now configured via the Network License Server Administration (1.11.17). Once the Network license is activated and the server is started, the server is ready to serve licenses. It is now possible to use the Network license (1.11.11) in BricsCAD, on a client computer.

Roaming

With a Network license, you have the ability to allow a floating license to roam to a system which will subsequently be disconnected from the network. The resulting license can be used for the number of days specified when the license was set to roam, and is checked back in automatically at the end of this time. In addition, you can return the roamed license back to the license pool early if this is desired. The license file(s) on the network license server contain(s) a parameter **max_roam_count**. This is set to the total number of seats on a network license by default. See also Roaming (1.11.11) on the client side.

Related topics

Configure the proxy settings (1.11.4) Linux Network license server setup (1.11.16) Server administration (1.11.17) Use a Network license (1.11.11) Roaming licenses (client side) (1.11.11) RLM License Administration Manual RLM FAQ

1.11.16 Linux network license server setup

Tutorial: BricsCAD License Server Setup Click here (1.1.3) to watch

Install the Bricsys Network License Manager

Download and install the Bricsys Network License Manager on the computer that will be used as network license server. Download Bricsys Network License Manager (Linux) Extract Bricsys-NetworkLicenseManager.tgz into any folder (command line: tar xvzf Bricsys-NetworkLicenseManager.tgz).

1. Start the application rlm in that folder (command line in that folder: ./rlm).
2. Open a browser at **http://localhost:5054**.
3. On that web page, click the System Info button on the left. Copy an ethernet hostid and use this to activate your license on your My Account page on the Bricsys website.
4. When you download a license file from the Bricsys website, store it in the folder where you extracted Bricsys-NetworkLicenseManager.tgz.
5. Click the Reread/Restart Servers button.
6. Click the Status button to see the server status, on the next page click the bricsys button in the table to view Bricsys licenses.
7. Click the RLM Manual... button to learn how to configure the license server in detail. Among other things, it is explained on that page how to automatically start rlm when the server boots.

Once the Network license is activated and the server is started, the server is ready to serve licenses. It is now possible to use the Network license (1.11.11) in BricsCAD, on a client computer.

Related topics

Configure the proxy settings (1.11.4) Windows Network license server setup (1.11.15) Server administration (1.11.17) Use a Network license (1.11.11) RLM License Administration Manual RLM FAQ

1.11.17 Network license server administration

When the Network license server (also named RLM License server) is running, it can be configured using the Reprise License Server Administration web interface.

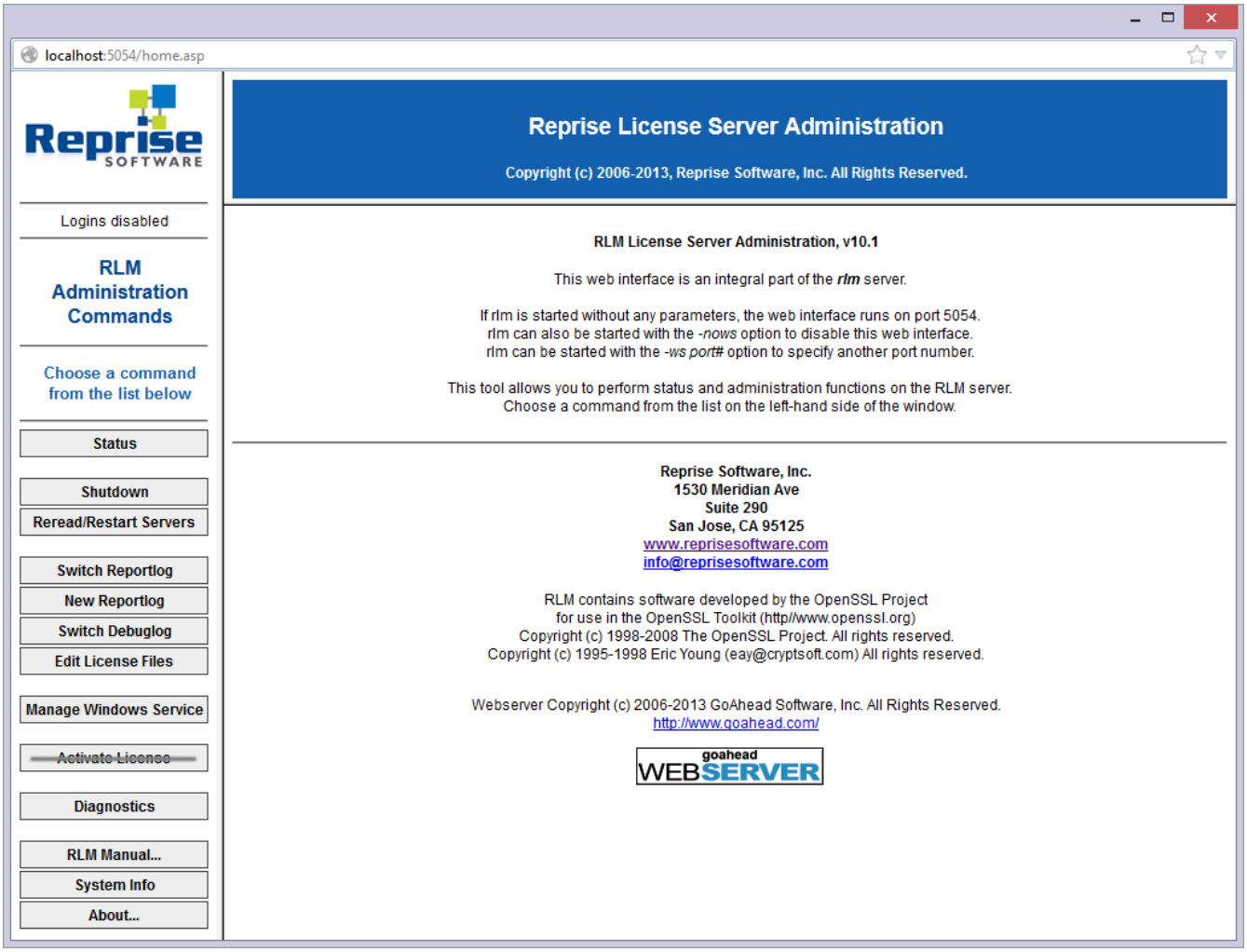
The web interface

1. (If the network license server is not running) In the Start menu, click the **License Server** shortcut (on Windows Vista or higher: right-click > Run as administrator). A console window opens.
2. Start a web browser.
3. Go to **http://localhost:5054**. The Reprise License Server Administration web interface opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/RLM-Home.png})

NOTES

- Do not use the **Activate License** button!
- To get an overview of all options, click the RLM Manual... button on the left or go to the RLM License Administration Manual.

Figure 1.119


localhost:5054/home.asp
Reprise SOFTWARE
Logins disabled
RLM Administration Commands
Choose a command from the list below
Status
Shutdown
Reread/Restart Servers
Switch Reportlog
New Reportlog
Switch Debuglog
Edit License Files
Manage Windows Service
Activate License
Diagnostics
RLM Manual...
System Info
About...
Reprise License Server Administration
Copyright (c) 2006-2013, Reprise Software, Inc. All Rights Reserved.
RLM License Server Administration, v10.1
This web interface is an integral part of the rlm server.
If rlm is started without any parameters, the web interface runs on port 5054.
rlm can also be started with the -nows option to disable this web interface.
rlm can be started with the -ws port# option to specify another port number.
This tool allows you to perform status and administration functions on the RLM server.
Choose a command from the list on the left-hand side of the window.
Reprise Software, Inc.
1530 Meridian Ave
Suite 290
San Jose, CA 95125
www.reprisesoftware.com
info@reprisesoftware.com
RLM contains software developed by the OpenSSL Project
for use in the OpenSSL Toolkit (http//www.openssl.org)
Copyright (c) 1998-2008 The OpenSSL Project. All rights reserved.
Copyright (c) 1995-1998 Eric Young (eay@cryptsoft.com) All rights reserved.
Webserver Copyright (c) 2006-2013 GoAhead Software, Inc. All Rights Reserved.
http://www.goahead.com/
goahead WEBSERVER

Start the server automatically

The network license server can be started automatically when booting the license server computer.

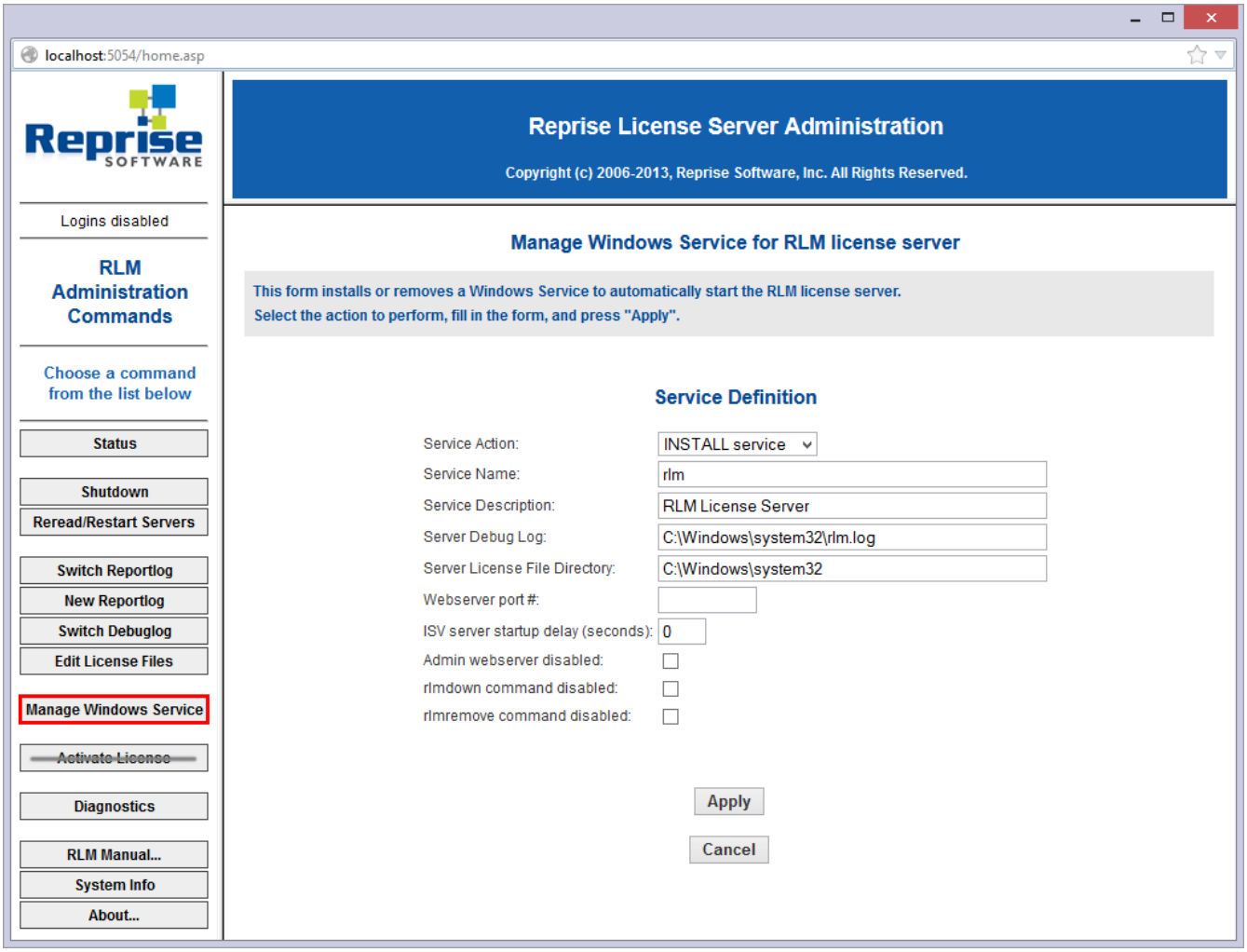
NOTES

- On Windows Vista or higher, the network license server must be started with administrator rights. Otherwise the service installation will fail. See Start the Network license server (1.11.15).
- The procedure below is for servers on Windows platform only. To automatically start a network license server on a Linux platform, see the RLM License Administration Manual.

1. Click the **Manage Windows Service** button on the left. The Manage Windows Service for RLM license server page opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/RLM-ManageWindowsService.png})
2. Click the **Apply** button.

The service is now installed but not started because the server is already running. The next time the server computer will be booted, the service will start the network license server automatically.

Figure 1.120


localhost:5054/home.asp
Reprise SOFTWARE
Logins disabled
RLM Administration Commands
Choose a command from the list below
Status
Shutdown
Reread/Restart Servers
Switch Reportlog
New Reportlog
Switch Debuglog
Edit License Files
Manage Windows Service
Activate License
Diagnostics
RLM Manual...
System Info
About...
Reprise License Server Administration
Copyright (c) 2006-2013, Reprise Software, Inc. All Rights Reserved.
Manage Windows Service for RLM license server
This form installs or removes a Windows Service to automatically start the RLM license server.
Select the action to perform, fill in the form, and press "Apply".
Service Definition
Service Action: INSTALL service
Service Name: rlm
Service Description: RLM License Server
Server Debug Log: C:\Windows\system32\rlm.log
Server License File Directory: C:\Windows\system32
Webserver port #:
ISV server startup delay (seconds): 0
Admin webserver disabled:
rlmdown command disabled:
rlmremove command disabled:
Apply
Cancel

Check the license and server status

You can check the status of all network licenses and of the server.

1. Click the **Status**button on the left. The page Status for ”rlm” on [LicenseServer] (port 5053) opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/RLM-ServerStatus.png})

2. Click the **bricsys** button in the Server Status column of the ISV Servers table to view the status of the Bricsys Network license. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/RLM-ISVStatus.png})

Figure 1.121

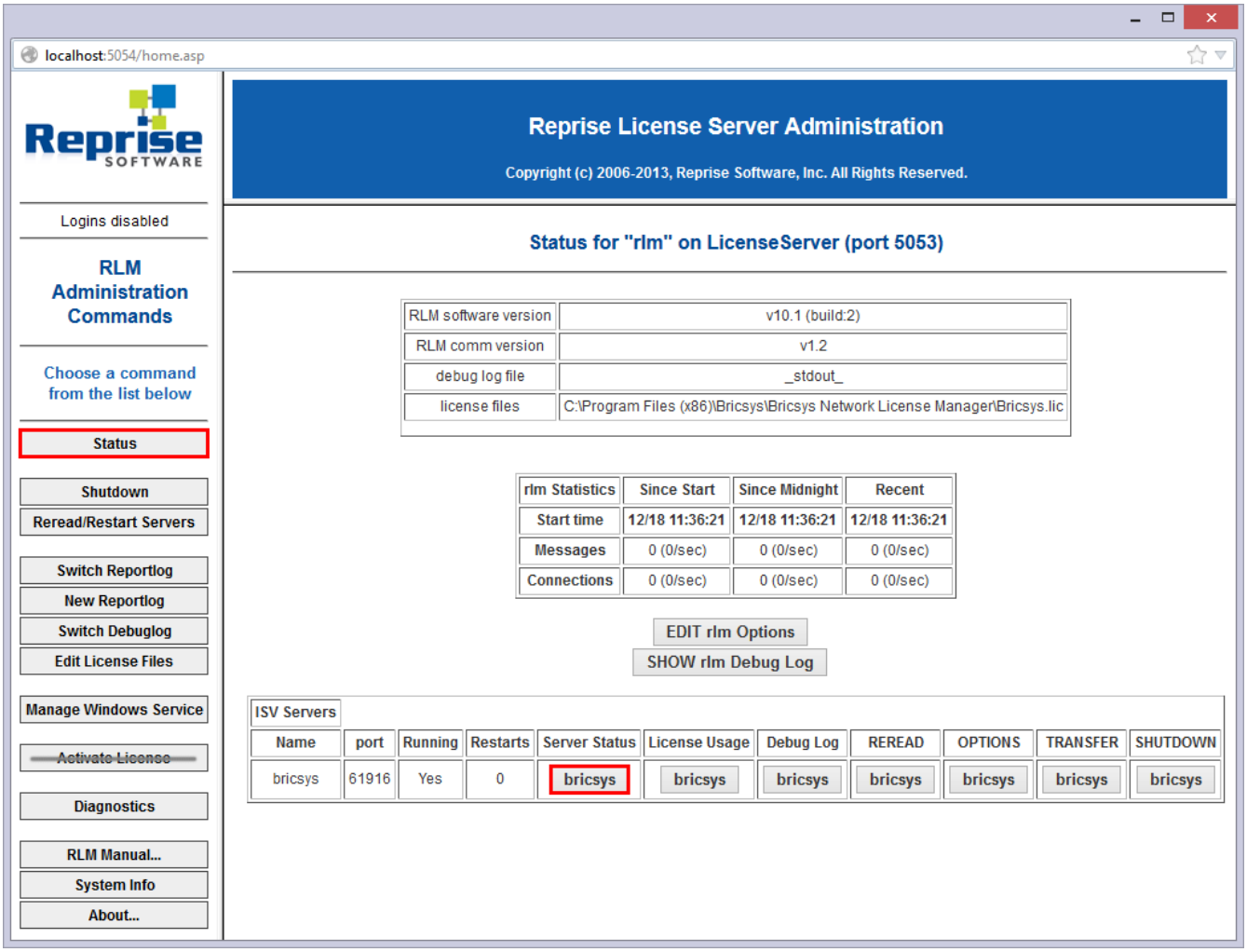
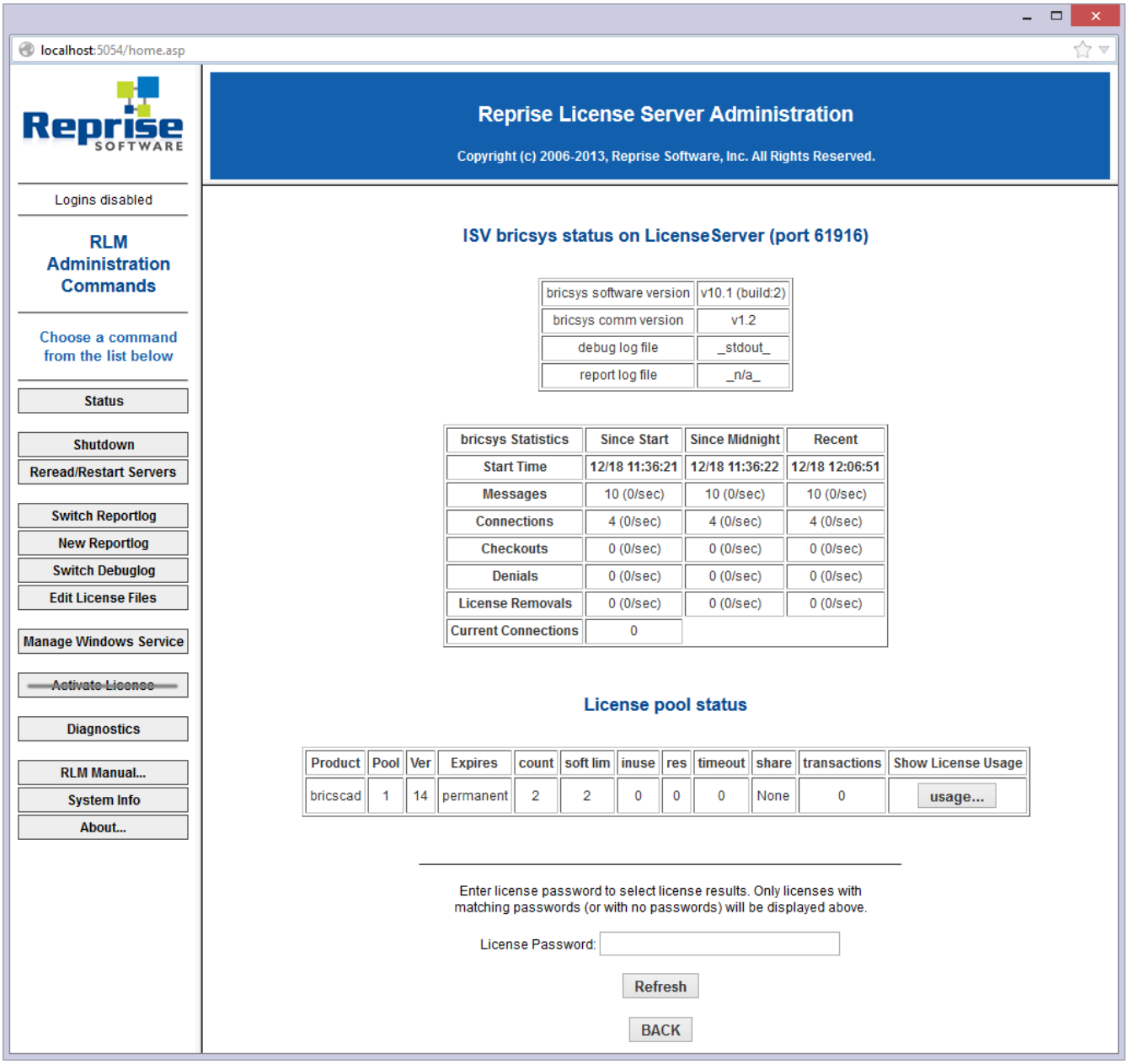

localhost:5054/home.asp
Reprise SOFTWARE
Logins disabled
RLM Administration Commands
Choose a command from the list below
Status
Shutdown
Reread/Restart Servers
Switch Reportlog
New Reportlog
Switch Debuglog
Edit License Files
Manage Windows Service
Activate License
Diagnostics
RLM Manual...
System Info
About...
Reprise License Server Administration
Copyright (c) 2006-2013, Reprise Software, Inc. All Rights Reserved.
Status for "rlm" on LicenseServer (port 5053)
RLM software version v10.1 (build:2)
RLM comm version v1.2
debug log file _stdout_
license files C:\Program Files (x86)\Bricsys\Bricsys Network License Manager\Bricsys.lic
rlm Statistics Since Start Since Midnight Recent
Start time 12/18 11:36:21 12/18 11:36:21 12/18 11:36:21
Messages 0 (0/sec) 0 (0/sec) 0 (0/sec)
Connections 0 (0/sec) 0 (0/sec) 0 (0/sec)
EDIT rlm Options
SHOW rlm Debug Log
ISV Servers
Name port Running Restarts Server Status License Usage Debug Log REREAD OPTIONS TRANSFER SHUTDOWN
bricsys 61916 Yes 0 bricsys bricsys bricsys bricsys bricsys bricsys bricsys

Figure 1.122

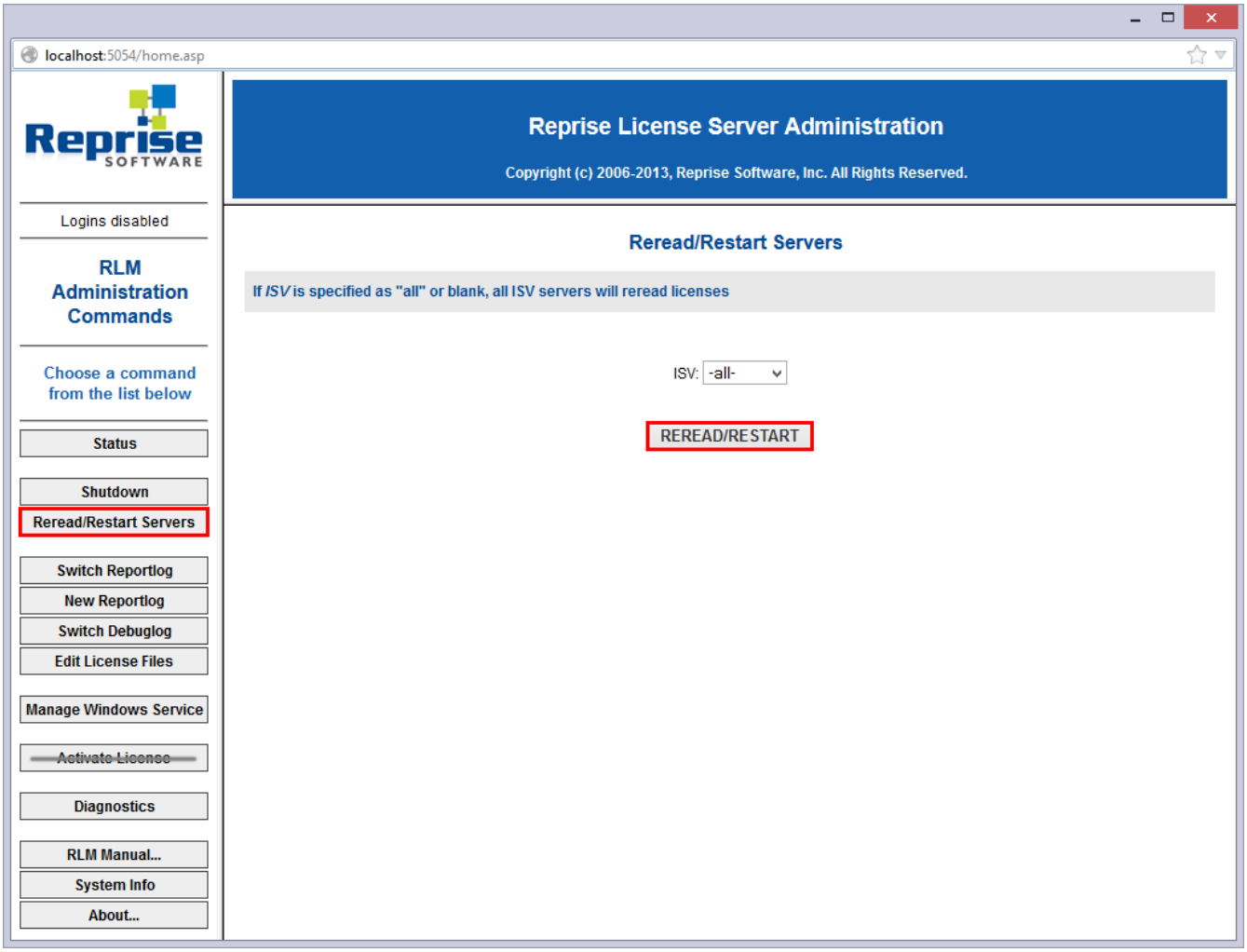

localhost:5054/home.asp
Reprise SOFTWARE
Logins disabled
RLM Administration Commands
Choose a command from the list below
Status
Shutdown
Reread/Restart Servers
Switch Reportlog
New Reportlog
Switch Debuglog
Edit License Files
Manage Windows Service
Activate License
Diagnostics
RLM Manual...
System Info
About...
Reprise License Server Administration
Copyright (c) 2006-2013, Reprise Software, Inc. All Rights Reserved.
ISV bricsys status on LicenseServer (port 61916)
bricsys software version v10.1 (build:2)
bricsys comm version v1.2
debug log file _stdout_
report log file _n/a_
bricsys Statistics Since Start Since Midnight Recent
Start Time 12/18 11:36:21 12/18 11:36:22 12/18 12:06:51
Messages 10 (0/sec) 10 (0/sec) 10 (0/sec)
Connections 4 (0/sec) 4 (0/sec) 4 (0/sec)
Checkouts 0 (0/sec) 0 (0/sec) 0 (0/sec)
Denials 0 (0/sec) 0 (0/sec) 0 (0/sec)
License Removals 0 (0/sec) 0 (0/sec) 0 (0/sec)
Current Connections 0
License pool status
Product Pool Ver Expires count soft lim inuse res timeout share transactions Show License Usage
bricscad 1 14 permanent 2 2 0 0 0 None 0 usage...
Enter license password to select license results. Only licenses with matching passwords (or with no passwords) will be displayed above.
License Password:
Refresh
BACK

Reread the license / Restart the server

After activating a Network license while the server was running, you have to reread the license / restart the server.

1. Click the **Reread/Restart Servers** button on the left. The Reread/Restart Servers page opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/RLM-RereadRestart.png})
2. Click the **Reread/Restart Servers** button. The server restarts and rereads the Network license.

Figure 1.123


localhost:5054/home.asp
Reprise SOFTWARE
Logins disabled
RLM Administration Commands
Choose a command from the list below
Status
Shutdown
Reread/Restart Servers
Switch Reportlog
New Reportlog
Switch Debuglog
Edit License Files
Manage Windows Service
Activate License
Diagnostics
RLM Manual...
System Info
About...
Reprise License Server Administration
Copyright (c) 2006-2013, Reprise Software, Inc. All Rights Reserved.
Reread/Restart Servers
If ISV is specified as "all" or blank, all ISV servers will reread licenses
ISV: -all-
REREAD/RESTART

Related topics

Configure the proxy settings (1.11.4) Windows network license server setup (1.11.15) Linux network license server setup (1.11.16) Use a Network license (1.11.11) RLM License Administration Manual RLM FAQ

1.11.18 Advanced configuration

Once the Network license server is set up, BricsCAD can report error -17 when trying to connect to server. Reasons can be due to the presence of an antivirus and/or a firewall and/or a proxy server.

Antivirus

If a antivirus software is running on the server and blocking the License Server program (rlm.exe on Windows, rlm on Linux/Mac), the RLM service needs be unblocked in the antivirus. See your antivirus configuration on how to unblock services.

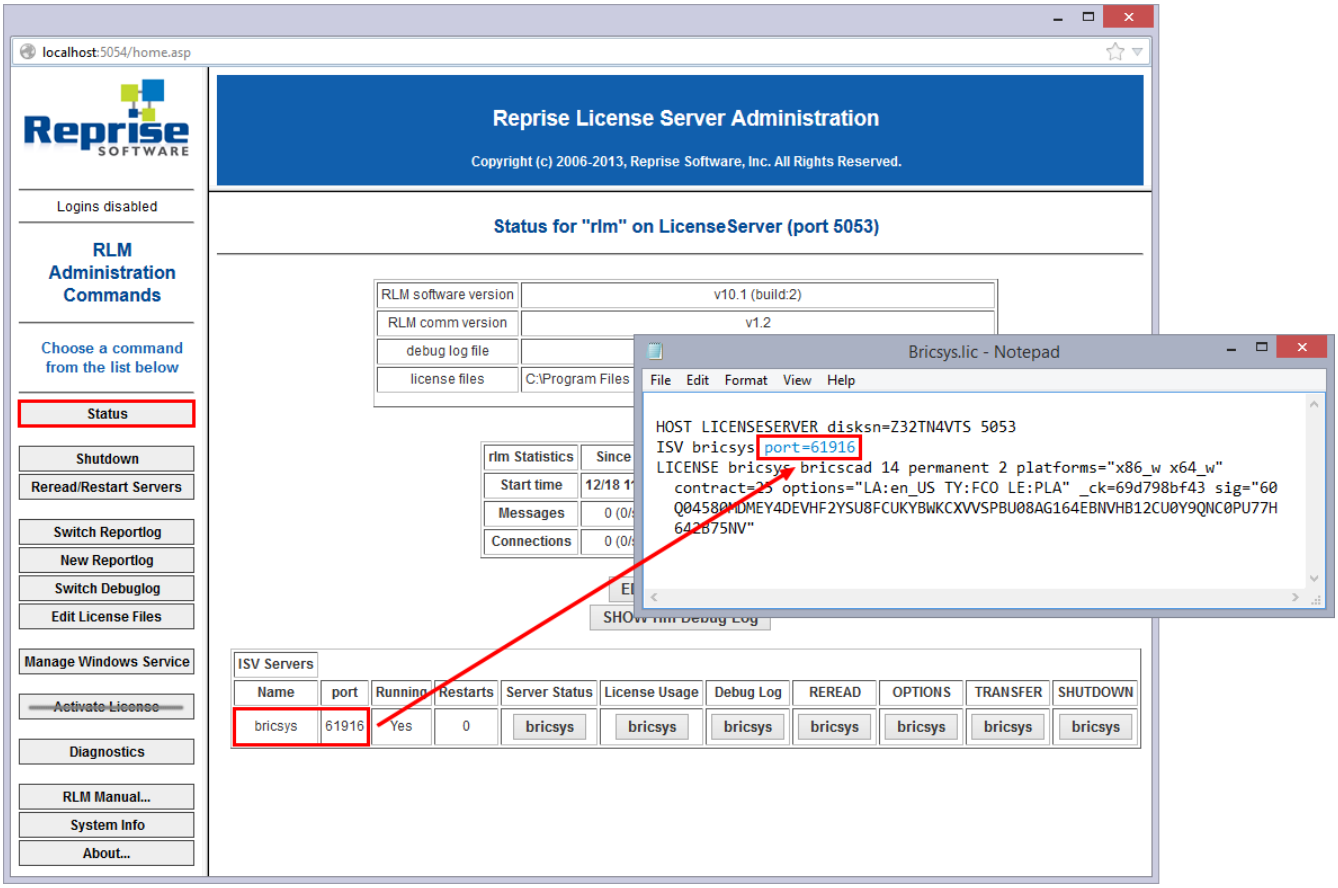
Firewall

If a firewall is installed between the server and BricsCAD, two port numbers need to be opened in the firewall: the port number 5053 and the port number assigned to the Bricsys license server This port number is dynamically assigned when the License Server starts. To open this port in the firewall, it must be first set in the Bricsys.lic file on the server. Proceed as follows:

1. Open the License Server Administration web interface at **http://localhost:5054**.
2. Click the **Status** button on the left. The page Status for ”rlm” on [LicenseServer] (port 5053) opens.
3. In the ISV Servers table, look at the bricsys row. The port number is displayed in the second column. Copy this port number.
4. Open the Bricsys.lic file in a text editor (Notepad, Gedit, ...). The default location of the file is C:\Program Files (x86)\Bricsys\Bricsys Network License Manager. You can fix the port number by modifying the line ISV bricsys into: ISV bricsys port=[port number] where [port number] must be replaced by port number copied from the ISV Servers table. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/BricsCAD/source/ART/Firewa
5. In the firewall, open both numbers 5053 and the port number you added in the Bricsys.lic file.
6. Shut down and restart the License Server program.

Note: If this does not solve it, add the License Server program (rlm.exe on Windows, rlm on Linux/Mac) as an exception onto the firewall exceptions, then restart the License Server again.

Figure 1.124



Proxy server

If a proxy server exists between BricsCAD and the License Server, proxy settings should be configured in BricsCAD. See how to configure the proxy settings permanently for Network License use (1.11.4).

Related topics

Configure the proxy settings (1.11.4) Network license server setup (1.11.14)

1.12 Interface

1.12.1 Keyboard Shortcuts

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/keyboardshortcutsheader-1.png}) Are you avoiding a key BricsCAD performance tool? Are you a BricsCAD user who favors using menus over using the keyboard for command input? If you are, you might want to reconsider. You need to know that the most often used commands in BricsCAD have fast keyboard equivalents. These keyboard shortcuts can be a great performance enhancer, especially when toggling settings. To help you get started using them, we've created a map of BricsCAD's keyboard shortcuts. Your computer's keyboard is sitting right in front of you – don't ignore it! Using BricsCAD keyboard shortcuts can boost your productivity over using menus alone. You can do things much faster with keyboard shortcuts then you can by clicking through multiple menu panels. Don't waste any more of your precious time – get to know the time-saving BricsCAD keyboard shortcuts. At Bricsys, we're all about speeding your CAD workflow. An easy-to-print PDF version of the BricsCAD keyboard shortcut map is available here. The following keyboard shortcuts are available:

Keyboard	Description
F1	Opens the BricsCAD Help.
F2	Toggles the display of the Prompt History window.
F3	Toggles Entity Snaps (1.15.15).
F4	Toggles the tablet mode (1.12.8).
F5	Sets the isoplane (see Using Isometric snap (1.15.11)).
F6	Toggles the Dynamic UCS (1.15.18) feature.
F7	Toggles the display of the Grid (1.15.11).
F8	Toggles the Ortho (1.15.2) setting. Disables Polar Tracking (1.15.16), if on.
F9	Toggles the Snap (1.15.11) setting.
F10	Toggles the Polar Tracking (1.15.16) setting. Disables Ortho (1.15.2), if on.
F11	Toggles the Entity Snap Tracking (1.15.17) setting.
F12	Toggles the Quad (1.12.5) display.
Alt+F8	Launches the VbaRun (6.32.6) command.
Alt+F11	Launches the VbaIde (6.32.1) command.
Ctrl+1*	Toggles the Properties bar (1.12.4).
Ctrl+2*	Opens the Drawing Explorer (1.14.2).
Ctrl+9*	Toggles the display of the command bar.
Ctrl+0*	Toggles the display of user interface entities as specified by the ScreenCleanOptions (1.13.11) system variable. Executes the CleanScreenOn (6.9.22) and CleanScreenOff (6.9.21) commands.
Ctrl+A	Selects all entities.
Ctrl+B	Toggles the Snap (1.15.11) setting.
Ctrl+C	Copies the selection to the clipboard (CopyClip (6.9.52) command).
Ctrl+Shift+C	Copies the selection along with a base point (CopyBase (6.9.51) command).
Ctrl+E	Sets the isoplane (see Using Isometric Snap (1.15.11)).
Ctrl+F	Launches the Find (6.14.8) command.
Ctrl+G	Toggles the display of the Grid (1.15.11).
Ctrl+H	Toggles the value of the PICKSTYLE system variable, which controls the selection of groups and associative hatches.
Ctrl+I	Sets the readout of the coordinate field (1.15.3) in the Status Bar (1.12.2).
Ctrl+J	Repeats the previously issued command.
Ctrl+K	Attach a hyperlink to an object or modify an existing hyperlink (Hyperlink (6.16.12) command).
Ctrl+L	Toggles the Ortho (1.15.2) setting.
Ctrl+M	Repeats the previously issued command.
Ctrl+N	Creates a new drawing (New (6.22.4) command).
Ctrl+O	Opens an existing drawing (Open (6.23.8) command).
Ctrl+P	Prints the current drawing (Print (6.24.46) command).
Ctrl+Shift+P	Toggles the Properties bar (1.12.4).

Ctrl+Q	Quits the application; prompts to save changes (Quit (6.25.8) command).
Ctrl+R	Iterates through viewports.
Ctrl+S	Saves the current drawing (QSave (6.25.4) command).
Ctrl+T	Turns the tablet on/off.
Ctrl+V	Pastes the contents of the Clipboard into the drawing (PasteClip (6.24.10) command).
Ctrl+Alt+V	Pastes the contents of the Clipboard in a specified format (PasteSpec (6.24.11) command).
Ctrl+Shift+V	Pastes the contents of the Clipboard as a block (PasteBlock (6.24.9) command).
Ctrl+X	Cuts the selection and puts it on the clipboard (CutClip (6.9.59) command).
Ctrl+Y	Redoes the last action undone (Redo (6.26.8) command).
Ctrl+Z	Undoes the last action (Undo (6.30.5) command).
Ctrl+[	Cancels the running command.
Ctrl+\	Cancels the running command.
Shift+F2	Toggles the display of the command bar.
Shift+F3	Turns the Status Bar on/off (StatBar (6.27.45) command).
Shift+F4	Turns the Scroll Bars on/off (ScrollBar (6.27.10) command).
Shift+F8	Launches the Visual Basic Project Manager (VbaMan (6.32.4) command).
Shift+F11	Launches the VBA COM Add-In Manager (AddInMan (6.4.4) command).
PgUp	Moves the view up.
PgDn	Moves the view down.
Shift+Left Arrow	Moves the view to the left.
Shift+Right Arrow	Moves the view to the right.
Shift+Up Arrow	Moves the view up.
Shift+Down Arrow	Moves the view down.

* Do not use the numeric keypad. On AZERTY-keyboards: do not press the Shift-key when hitting the number key.

Figure 1.125

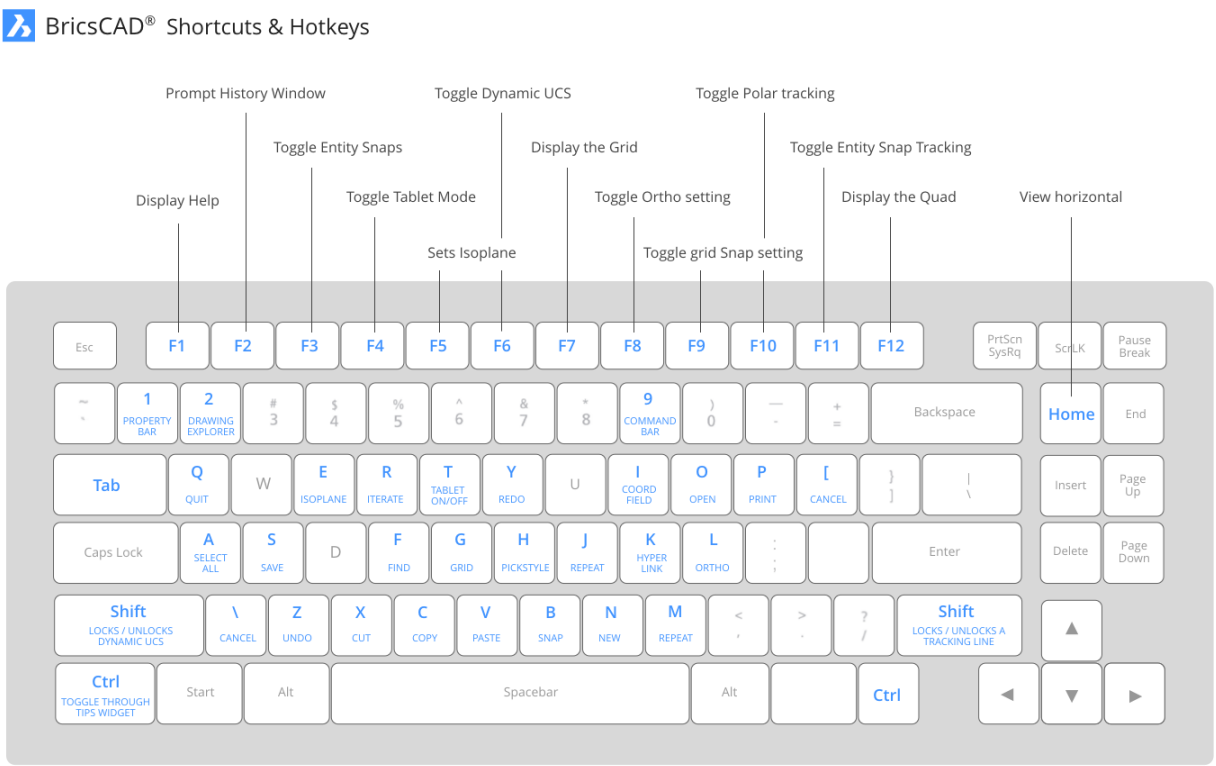
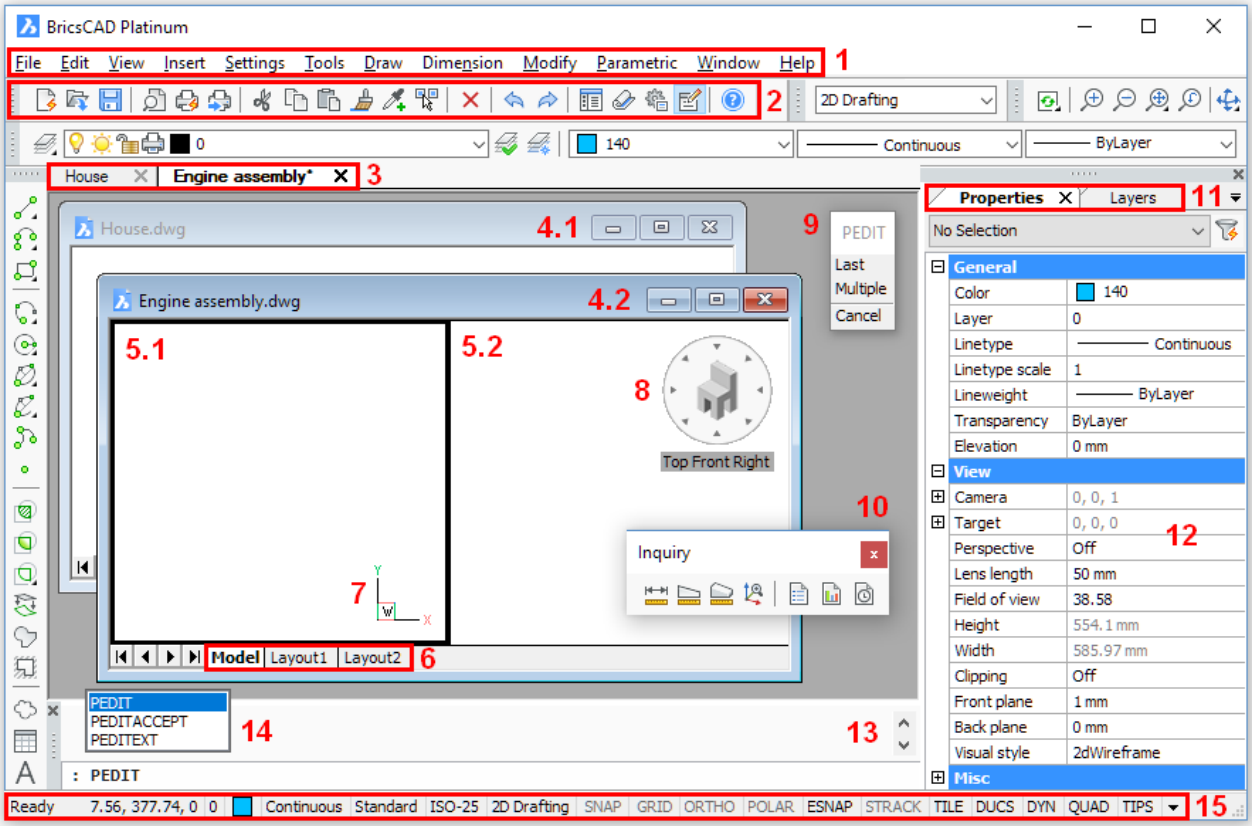

BricsCAD® Shortcuts & Hotkeys
Prompt History Window
Toggle Dynamic UCS
Toggle Polar tracking
Toggle Entity Snaps
Display the Grid
Toggle Entity Snap Tracking
Display Help
Toggle Tablet Mode
Toggle Ortho setting
Display the Quad
View horizontal
Sets Isoplane
Toggle grid Snap setting
Esc
F1
F2
F3
F4
F5
F6
F7
F8
F9
F10
F11
F12
PrtScn SysRq
ScrLK
Pause Break
Backspace
Home
End
Tab
Q QUIT
E ISOPLANE
R ITERATE
T TABLET ON/OFF
Y REDO
I COORD FIELD
O OPEN
P PRINT
CANCEL
Insert
Page Up
Caps Lock
A SELECT ALL
S SAVE
F FIND
G GRID
H PICKSTYLE
J REPEAT
K HYPER LINK
L ORTHO
Enter
Delete
Page Down
Shift
LOCKS / UNLOCKS DYNAMIC UCS
Z UNDO
X CUT
C COPY
V PASTE
B SNAP
N NEW
M REPEAT
Shift
LOCKS / UNLOCKS A TRACKING LINE
Ctrl
TOGGLE THROUGH TIPS WIDGET
Start
Alt
Spacebar
Alt
Ctrl

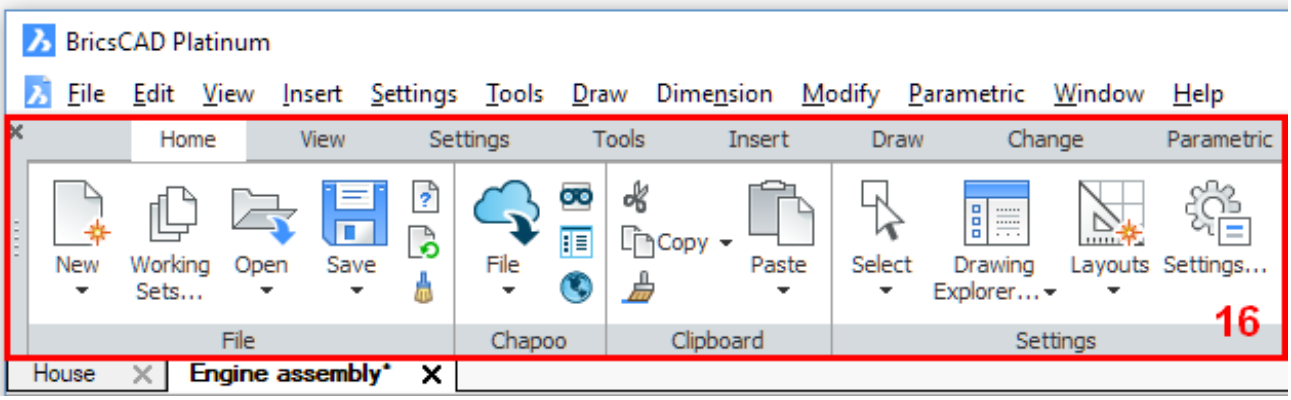
Figure 1.126



1.12.2 The BricsCAD Application Window

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/00ApplicationWindow.png})
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/00ApplicationWindowRibbonDoctab

Figure 1.127



Components of the BricsCAD application window:

1. Menu Bar
2. Toolbar (docked)
3. Document tabs
4. Drawing windows
5. Drawing viewports
6. Window tabs
7. UCS icon
8. Look From widget
9. Prompt Menu
10. Toolbar (floating)
11. Panel tabs
12. Properties panel (see Dockable Panels (1.12.7))
13. Commandline
14. Suggestion list
15. Status Bar
16. Ribbon

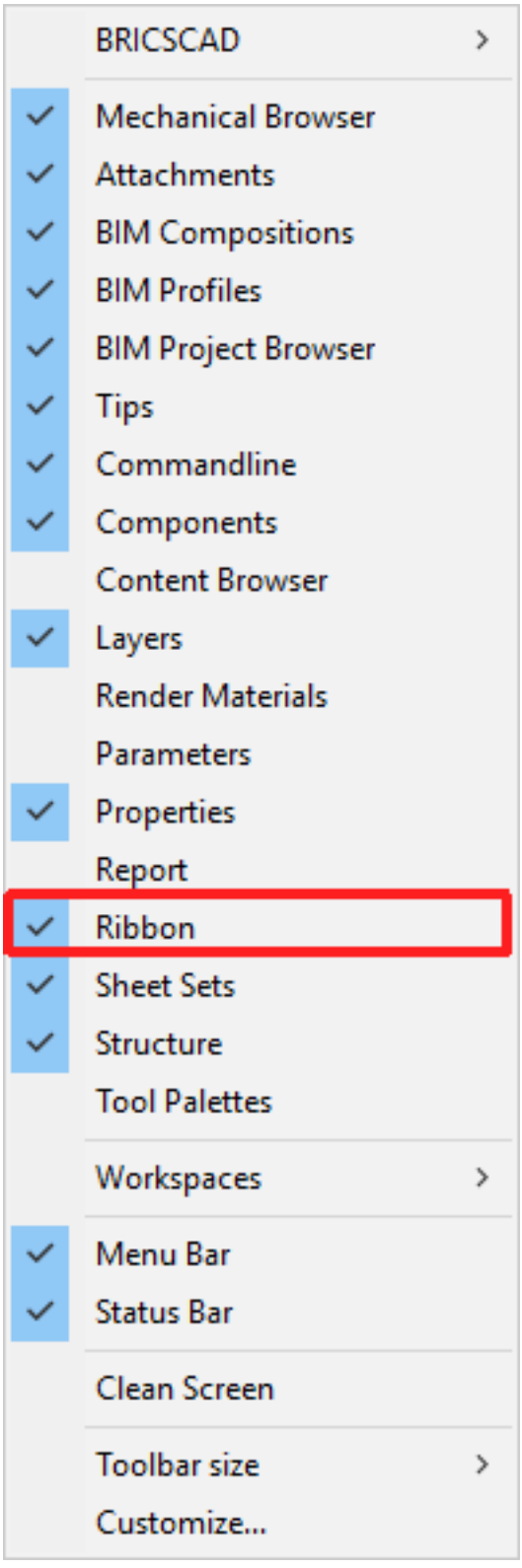
Tutorial: Exploring the BricsCAD User Interface Click here (1.2.1) to watch.

To display the ribbon

1. Execute the Ribbon (6.26.31)command.
2. Right click an empty space in the Layout or Drawing tab area, then choose **Ribbon** in the context menu.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/ContextMenuRibbon.png})

Figure 1.128


BRICSCAD
Mechanical Browser
Attachments
BIM Compositions
BIM Profiles
BIM Project Browser
Tips
Commandline
Components
Content Browser
Layers
Render Materials
Parameters
Properties
Report
Ribbon
Sheet Sets
Structure
Tool Palettes
Workspaces
Menu Bar
Status Bar
Clean Screen
Toolbar size
Customize...

To hide the ribbon

1. Execute the RibbonClose (6.26.33) command.
2. Right click an empty space in the Layout or Drawing tab area, then choose **Ribbon** in the context menu.

Working with document tabs

Document tabs allow to:

- Easily switch between all open drawings.
- Drag and drop the tabs to change their order.
- Click the **X** at the right-hand side of tab to close it.

Right click on a document tab to display a context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1

Item	Description
Close	Closes the current drawing (Close (6.9.25) command).
Close Left Tabs	Closes the drawings at the left of the current tab.
Close All But This	Closes all drawings, but the current drawing.
Close All	Closes all drawings (WCloseAll (6.33.5)command).
Open	Displays the **Open Drawing** dialog (Open (6.23.8) command).
New	Displays the **Create New Drawing** dialog New (6.22.4) command.
Save	Saves the current drawing (Save (6.27.1) command).
Save All	Saves all open drawings (SaveAll (6.27.2)command).
Save As	Launches the SaveAs (6.27.3) command.
Position	Lets you choose the position of the document tabs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/
Duplicate Tab	Saves a copy of the current drawing in a new tab, while keeping the original drawing open. Displays the **Save As** dialog.
Restore Last Session	Opens the drawings that were open in the previous BricsCAD session.
Open Folder	Opens the folder of the current drawing.
Cascade	Cascades the windows of the current drawings (WCascade (6.33.3) command).
Tile Horizontally	Tiles the windows of the current drawings horizontally (WhTile (6.33.9) command).
Tile Vertically	Tiles the windows of the current drawings vertically (WvTile (6.33.19) command).

The display and location of document tabs is controlled through user preferences. In the Settings (6.27.23) dialog go to **Program Options / Display / Document tabs** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/ **ShowDocTabs** : controls the display of document tabs. **DocTabPosition** : controls the location of the document tabs: top, bottom, left or right.

Figure 1.129

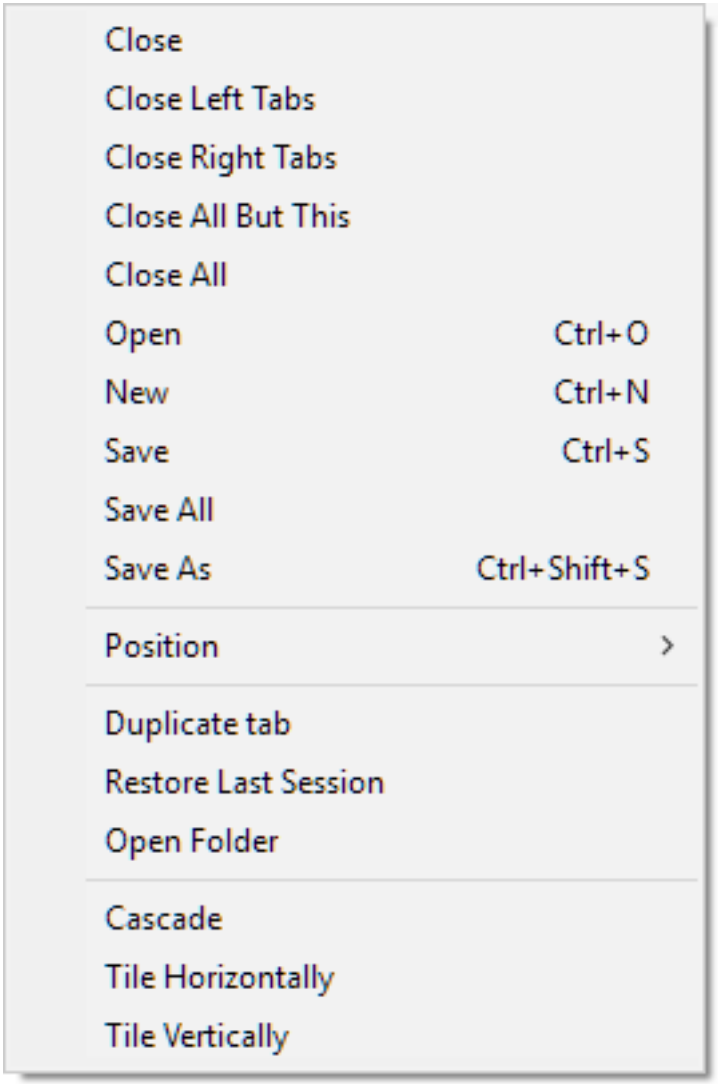
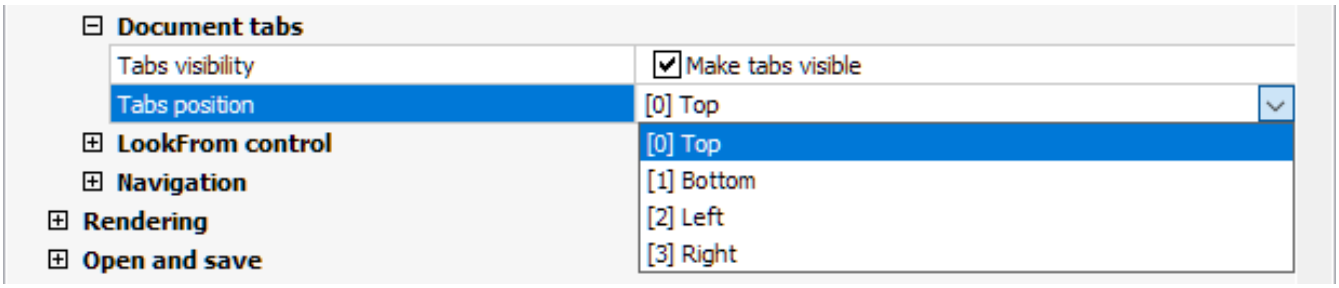


Figure 1.130



Open / Close the command line

Do one of the following:

- Choose **Command Bar**in the **View** Menu.
- Move the cursor to a docked toolbar, then right click and choose **Commandline**in the context menu.
- Double click the **Status** field at the left hand side of the **Status Bar** .

The **Command Window** closes if it was open and vice versa. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS

1. **Close** button
2. Press and hold the left mouse button to move the commanline.
3. **Command History List** field: the content of the history list field can be copied to the command edit field.
4. **Command Edit** field If the Enable flag of the AUTOCOMPLETEMODE (1.13.11) system variable is set, command entries are completed automatically as you type. All commands and/or system variables and/or user preferences that contain the current character string display in the **Suggestion** list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/00AutoCompletion.png}) Do one of the following:
   - Press Enter if the currently completed command displays.
   - Choose a command in the **Suggestion** list .

   Autocomplete options are: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/00A
5. **Command history** scroll buttons

NOTES

- You can modify the height of the docked command line by dragging its top or bottom edge, depending on its docking position, bottom or top.
- The size of a floating command line can be adjusted by dragging one of its edges.
- Use the arrow-up and arrow-down keys of the keyboard to browse the command history.
- When the command line is closed:
  - Command options and keyboard entries display in the status bar.
  - Press the F2 function key to toggle the Command History (6.29.13) window.
  - TheCLIPROMPTLINES (1.13.11) system variable sets the number of command history lines that display shortly at the bottom of the graphics window. SetCLIPROMPTLINES=0 to disable this feature.

Figure 1.131



Figure 1.132

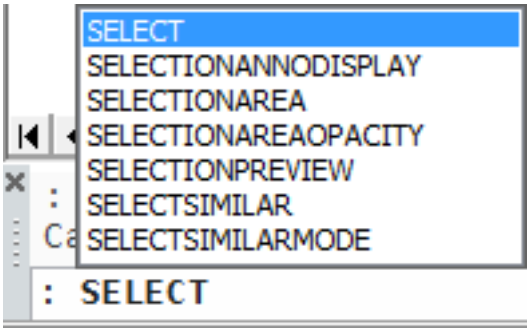


Figure 1.133

Auto complete mode	0x002F (47)
1	Enable
2	Auto-Append
4	Suggestion List
8	Display Icons (unsupported)
16	Exclude the display of System Variables
32	Display preference variables
Auto complete delay	0.3

To set the command line properties

Tutorial: Customizing the Command Prompt in BricsCAD Click here to watch.

1. Open the Settings (6.27.23) window.
2. Go to **Program Options / User Preferences / Command Line.** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel

Figure 1.134

Command line	
Command line font name	Verdana
Command line font size	8
Command line list background color	(255,255,221)
Command line list foreground color	(0,0,0)
Command line edit background color	(248,248,248)
Command line edit foreground color	(0,0,0)
Command line state	Command line is visible
Prompt prefix	:
Scroll history	256
Text evaluation	All responses to prompts for text strings and attribute values are taken literally

Open / Close the status bar

Do one of the following:

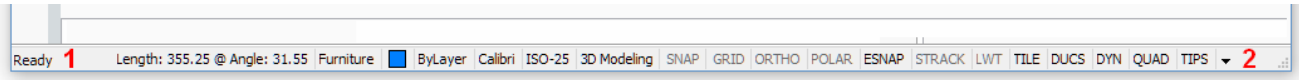
- Choose **Status Bar** in the **View** Menu.
- Press the **Shift + F3** keyboard shortcut.

The **Status Bar** closes if it was open and vice versa.

Working with the status bar

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/00StatusBar.png})

Figure 1.135



1. Status field:

- displays the status of the software.
- when the cursor is in a menu or on a toolbar: gives a brief description of the tool or menu item under the cursor.
- when the command window is closed: displays the command prompts, tool options and keyboard entry.
- double click the status field to open/close the **Commandline**.

2. Field List button: Click the arrow button ( ) to display a list of available **Status Bar** fields. Click an item in the list to toggle the display of the corresponding field in the **Status Bar** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
Cursor coordinate values: displays the coordinates of the current cursor position (depending on the setting of the **COORDS**variable).

1. Click to cycle the **Coordinate** setting. The sequence is **Geopgraphic** , **Relative** , **Absolute, Off.**
2. Right click, then choose an option in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU

Current Layer: displays the name of the current layer.

1. Click to open the **Layer Explorer**.
2. Right click to select the current layer in the context menu.
3. Right click and choose **Properties** in the context menu to open the Drawing Explorer - Layers (6.20.2).

Current Color: displays the current color.

1. Click to open the **Select Color** dialog window.
2. Right click to select one of the basic colors. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCA
3. Right click and choose **Select Color...** to select a color (1.17.4).

Current Linetype: displays the name of the current linetype.

1. Click to open the **Linetype Explorer** .
2. Right click to select the current linetype.
3. Right click and choose **Properties** in the context menu to open the Drawing Explorer - Linetypes (6.20.33).

Current Text Style: displays the name of the current text style.

1. Click to open the **Styles Explorer** .
2. Right click to select the current text style.
3. Right click and choose **Properties** in the context menu to open the Drawing Explorer - Text Styles (6.27.53).

Current Dimension Style: displays the name of the current dimension style

1. Click to open the **Dimensions Settings** dialog window.
2. Right click to select the current dimension style.
3. Right click and choose **Properties** in the context menu to open the Drawing Explorer - Dimension Styles (6.11.20).

Current Work Space: Indicates the current workspace.

1. Click to set the value of the WSCURRENT (1.13.11) system variable in the commandline.
2. Right click, then select the current workspace. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/B
3. Choose **Customize** in the context menu to open the Customize - Workspaces (1.13.10) dialog box.

Snap: Toggles the SNAPMODE (1.13.11) system variable.

1. Click to toggle Snap (6.27.38) **On** (SNAP) / **Off** (SNAP).

Figure 1.136

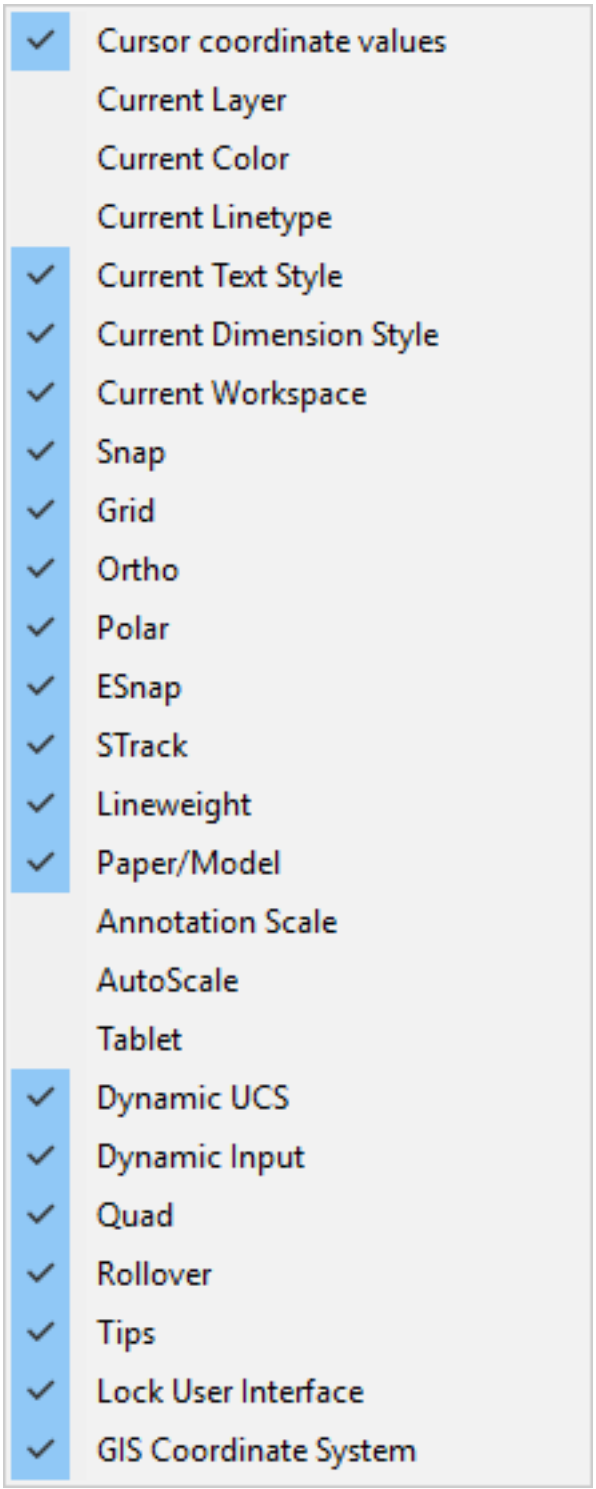

Cursor coordinate values
Current Layer
Current Color
Current Linetype
Current Text Style
Current Dimension Style
Current Workspace
Snap
Grid
Ortho
Polar
ESnap
STrack
Lineweight
Paper/Model
Annotation Scale
AutoScale
Tablet
Dynamic UCS
Dynamic Input
Quad
Rollover
Tips
Lock User Interface
GIS Coordinate System

Figure 1.137

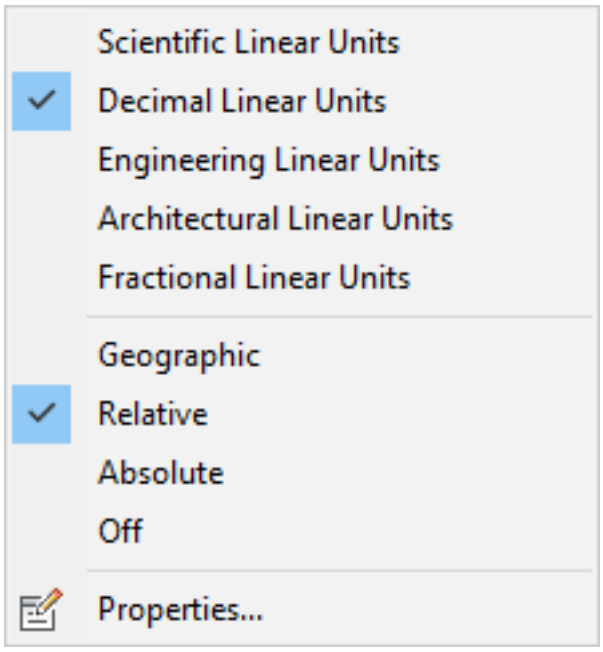


Figure 1.138

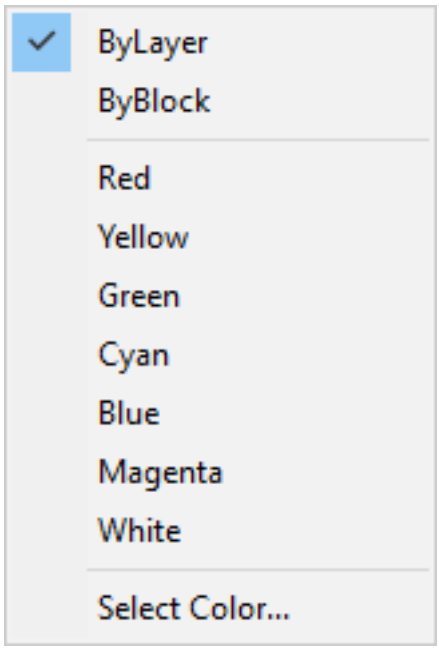


Figure 1.139

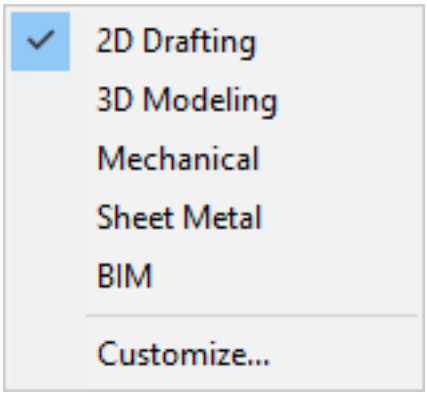


Figure 1.140

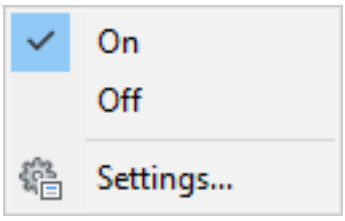
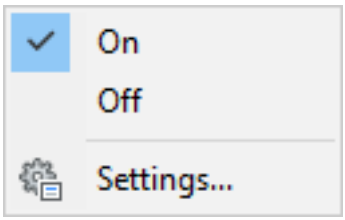


Figure 1.141



2. Right click, then choose **Settings** to edit the **Snap and Grid** settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

Grid: Toggles the GRIDMODE (1.13.11) system variable.

1. Click to toggle the display of the **GridOn** (GRID) / **Off** (GRID) in the current viewport.
2. Right click, then choose **Settings** to edit the **Snap and Grid** settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

Ortho: Toggles the ORTHOMODE (1.13.11) system variable; turning **Orthogonal mode** on automatically disables **Polar Tracking.**

1. Click to toggle Orthogonal mode (1.15.2)**On** (ORTHO) /**Off** (ORTHO).
2. Right click, then choose **Settings** to edit the **Orthomode** setting. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h

Polar: Toggles the **Polar Tracking** option of the AUTOSNAP (1.13.11) system variable.

1. Click to toggle Polar Tracking (1.15.16) **On** (POLAR) / **Off** (POLAR). Turning **Polar Tracking** on automatically disables **Orthogonal mode** .
2. Right click, then choose **Settings** to edit the **Polar Tracking** settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc

Esnap: Toggles the **Turn of all snaps** option of the OSMODE (1.13.11) system variable.

1. Click to toggle Entity Snaps (1.15.15) **On** (ESNAP) / **Off** (ESNAP).
2. Right click, then choose **Settings** to edit the **Entity Snaps** settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

Strack: Toggles the **Entity Snap Tracking** option of the AUTOSNAP (1.13.11) system variable.

1. Click to toggle Snap Tracking (1.15.17) **On** (STRACK) / **Off** (STRACK).
2. Right click, then choose **Settings** to edit the **Snap Tracking** settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

Figure 1.142

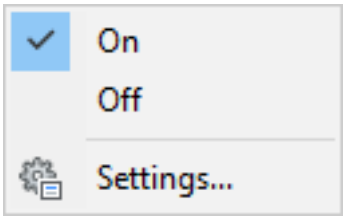


Figure 1.143

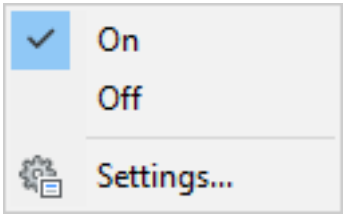
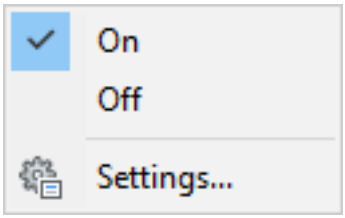


Figure 1.144



Lineweight (LWT): Toggles the LWDISPLAY (1.13.11) system variable.

1. Click to toggle the display of Line Weights (1.17.5) **On** (LWT) or **Off** (LWT).
2. Right click, then choose **Settings** to edit the **Lineweights** settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

Paper/Model (1.16.12): Allows to choose between model space and paper space.

- When working in Model Space, the field reads: **Tile** (= Model space with tiled viewports). Click to switch to the most recently opened paper space layout. Right click to select a paper space layout: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/00LayoutListModel.png})
- When working in a layout, the field reads: **M:Layout:** Model space with floating viewports or **P:Layout**: Paper space Click to toggle between paper space and model space with floating viewports. Right click, then choose: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/00LayoutList.png})
- **Paper Space**: Switch to paper space
- **Model Space**: Switch to model space with floating viewports
- **Model**: Switch to model space with tiled viewports
- **Layout**: Switch to the selected paper space layout.

Annotation Scale: Displays the value of theCANNOSCALE (1.13.11) (= current annotation scale) system variable. Right click:

1. Select an annotation scale in the list.
2. Choose **Custom** to launch the ScaleListEdit (6.27.7) command.
3. Check the **Hide Xef scales** option to list annotation scales of the current drawing only.

Autoscale: Click to toggle theANNOAUTOSCALE (1.13.11)system variable Tablet: Initializes the use of a drawing tablet (1.12.8). Dynamic UCS (DUCS): Click to toggle the **UCSDETECT (1.15.18)**system variable On (DUCS) or Off (DUCS). Dynamic Input (DYN): Toggles the **Switch all off temporarily** option of the DYNMODE (1.13.11)system variable.

Figure 1.145

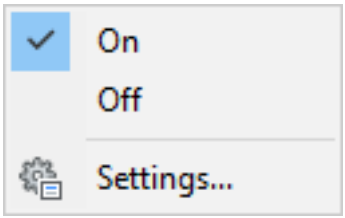


Figure 1.146

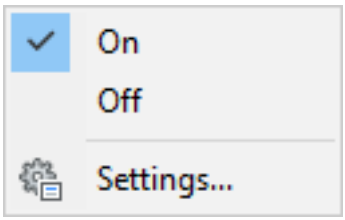


Figure 1.147

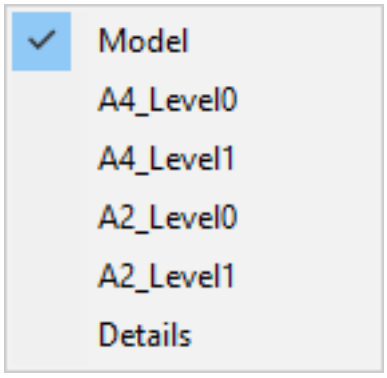


Figure 1.148

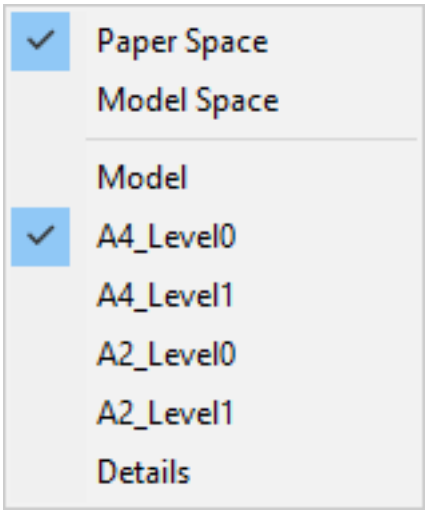


Figure 1.149

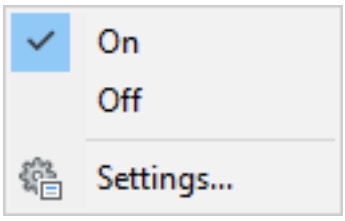
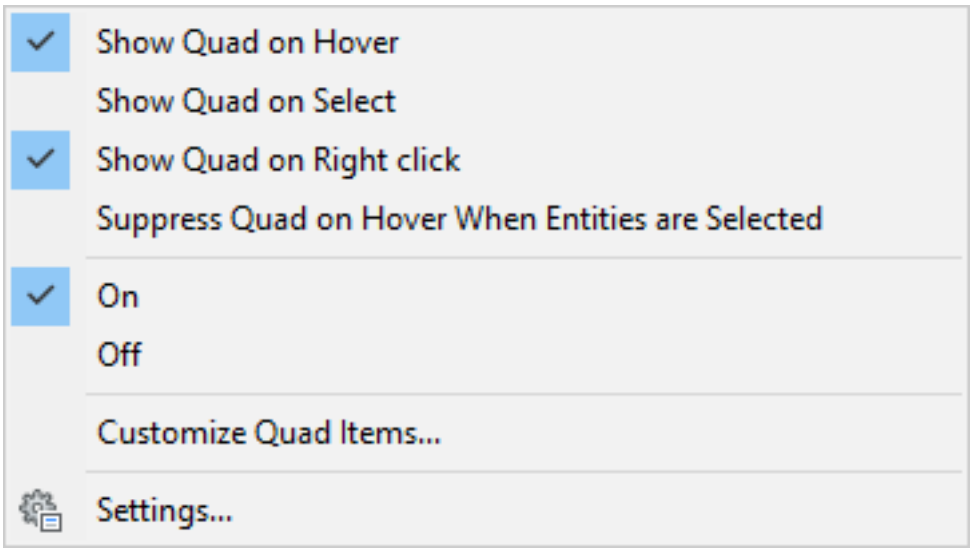


Figure 1.150



1. Click to toggle the display of Dynamic Dimensions (1.15.9)**On** (DYN) or **Off** (DYN).
2. Right click, then choose **Settings** to edit the **Dynamic Dimensions** settings. (\vref {fig:C:/Dev/Task-2448/images/staticbri

Quad: Toggles the **Switch off** option of the QUADDISPLAY (1.13.11)system variable.

1. Click to toggle the display of the Quad (1.12.5) **On** (QUAD) or **Off** (QUAD).
2. Right click, then choose an option in the context menu.
3. Right click, then choose **Settings** to edit the **Quad** settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e

Rollover tips (1.12.5) **(**RT**)**: Toggles the ROLLOVERTIPS (1.13.11)system variable.

1. Click to toggle the display of rollover tips.
2. Right click to select the display options for rollover tips: **On hovering entities** or **No rollover tips** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/00RolloverTipsContext.png})

Tips : Toggles the TIPS system variable. If TIPS = ON, an assistant widget reveals options available using the Ctrl-key. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/PolySolidJustificationTips.png} Tips when creating PolySolids (6.24.44).

Figure 1.151

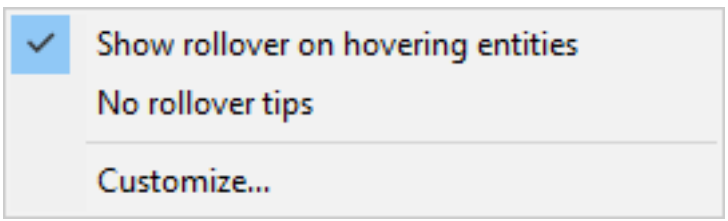


Figure 1.152



Figure 1.153

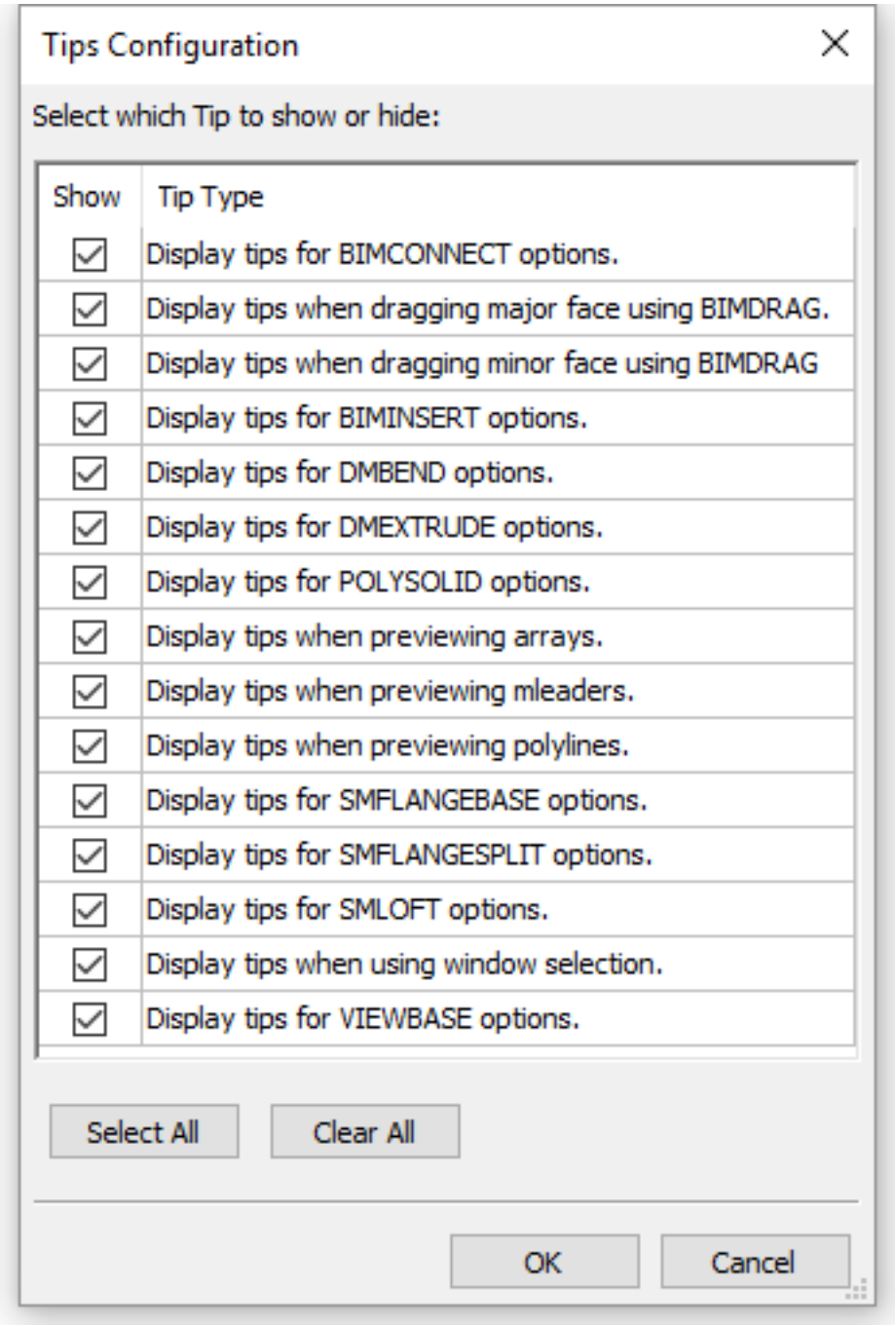


- Click to toggle the display of the TIPS widget On/off.
- Right click to display a context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Inte

- Click **On** or **Off** to set the display of the the TIPS widget.
- Click **Configure** to display the Tips Configuration dialog. Select which tips show when TIPS = ON. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/00TipsConfigurationDialog.png})

Lock User Interface: Allows to lock the location and size of toolbars and dockable panels. Press Ctrl to override. Click the **LOCKUI** field to toggle the lock. Press Ctrl to override. Which user interface elements are affected is controlled through the LOCKUI (1.13.11)system variable. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/

Figure 1.154


Tips Configuration
Select which Tip to show or hide:
Show
Tip Type
Display tips for BIMCONNECT options.
Display tips when dragging major face using BIMDRAG.
Display tips when dragging minor face using BIMDRAG
Display tips for BIMINSERT options.
Display tips for DMBEND options.
Display tips for DMEXTRUDE options.
Display tips for POLYSOLID options.
Display tips when previewing arrays.
Display tips when previewing mleaders.
Display tips when previewing polylines.
Display tips for SMFLANGEBASE options.
Display tips for SMFLANGESPLIT options.
Display tips for SMLOFT options.
Display tips when using window selection.
Display tips for VIEWBASE options.
Select All
Clear All
OK
Cancel

GIS Coordinate System : Displays the name of the current GIS coordinate system.

Figure 1.155



Display or hide scroll bars

To toggle scroll bars on / off:

1. (option) Hold down the Shift key, then press the **F4** function key.
2. (option) Choose **Scroll Bars** in the **View** menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/B

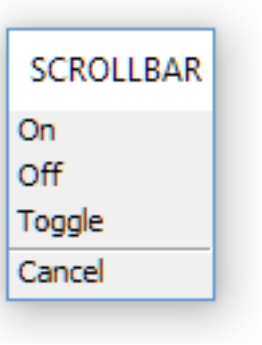
3. (option) Type **scrollbar** in the command bar, then do one of the following:
4. type **off** in the command bar or choose **Off** in the context menu.
5. type **on** in the command bar or choose **On** in the context menu.
6. type **T** in the command bar or choose **Toggle** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

NOTE	Use the **-**ToolPanel (6.29.21) command to show, hide or toggle the display of the various tool panels.

Figure 1.156



Figure 1.157



Hide/show a selection of user interface elements

The CleanScreenOn (6.9.22) and CleanScreenOff (6.9.21) commands allow to hide or show a selection of user interface elements simultaneously. Which user interface elements are affected by these commands is defined by the CLEANSCREENOPTIONS (1.13.11) system variable. The Ctrl+0 keyboard shortcut toggles the display of user interface elements. On AZERTY-keyboards: do not use the numeric keypad; do not press the Shift-key when hitting the number key.

Figure 1.158



1.12.3 Toolbars

The availability of a command depends on the license Level (1.10.3) (Shape - Classic - Pro - Platinum - Mechanical - Bim - Ultimate) as indicated in the command articles in the Command Reference section of the Bricys Help Center.

To set the icon size

You can choose between small (16 x 16) large (32 x 32) or extra large (64 x 64) icons on toolbars.

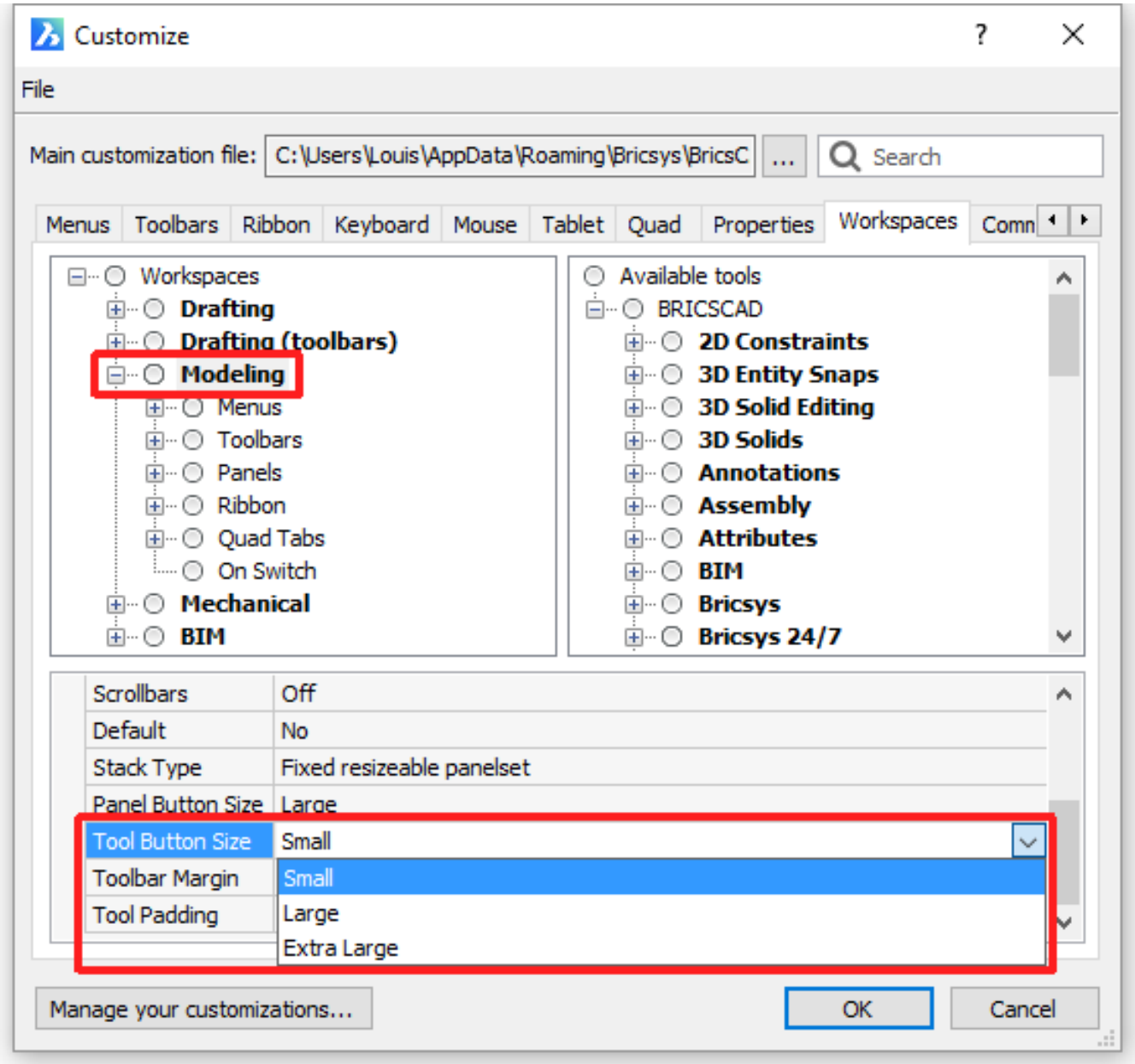
1. Move the cursor over a toolbar, then right click.
2. Choose **Toolbar size** then select either **Small Icons**,**Large Icons**or **Extra Large Icons**.



Small Large Extra Large

The icon size is saved in the **Tool Button Size** property of the current Workspaces (1.13.10). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/WorkspaceToolButtonSize.png})

Figure 1.159


Customize
File
Main customization file:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsC
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Properties
Workspaces
Comn
Workspaces
Drafting
Drafting (toolbars)
Modeling
Menus
Toolbars
Panels
Ribbon
Quad Tabs
On Switch
Mechanical
BIM
Available tools
BRICSCAD
2D Constraints
3D Entity Snaps
3D Solid Editing
3D Solids
Annotations
Assembly
Attributes
BIM
Bricsys
Bricsys 24/7
Scrollbars
Off
Default
No
Stack Type
Fixed resizeable panelset
Panel Button Size
Large
Tool Button Size
Small
Toolbar Margin
Small
Tool Padding
Large
Extra Large
Manage your customizations...
OK
Cancel

Main Toolbars

2D Constraints (1.15.7)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/2
3dCompare (6.3.3)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/3
3D Constraints (1.19.4)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/3
3D Entity Snaps (1.15.14)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/3
3D Solid Editing	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/3
3D Solids	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/3
Access	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/A 1Toolbar.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/A 2Toolbar.png})
Arcs (1.17.16)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/A
Assembly (3.6.1)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/A 1Toolbar.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/A 2Toolbar.png})
BIM	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/B 1Toolbar.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/B 2Toolbar.png})
Block Edit (6.6.7)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/B
Bricsys 24/7 (**??**)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/B
Circles (1.17.15)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/C
Compare (6.10.41)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/C
Design intent (1.18.3)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/D
Dimensions (1.21.2)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/D 1Toolbar.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/D 2Toolbar.png})
Dimensional Constraints (1.15.6)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/D
Direct Modeling (1.18.2)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/D
Draw (1.17.2)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/D
Drawing Views (3.10.1)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/D
Draw Order (1.23.9)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/D

Ellipses (1.17.17)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/E
Elliptical Arcs (1.17.18)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/E
Entity Data (6.13.3)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/E
Entity Properties	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/E
Entity Snaps (1.15.15)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/E
Images (1.24.1)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/I
Insert (6.17.13)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/I
Inquiry (1.15.19)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/I
Look From	Replaced by the Look From control (6.20.39).
Layout (1.16.15)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/L
Mechanical	Renamed to Assembly (3.6.1)
Meshes (1.6.17)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/M
Modify (1.23.1)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/M 1Toolbar.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/M 2Toolbar.png})
Parameters (6.24.5)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/P
Point Clouds (6.24.32)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/P
Ref Edit (6.26.12)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/P
Rendering (1.26.2)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/P
Sections (6.27.11)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/S
Selection Modes (1.23.5)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/S
Settings (6.27.23)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/S
Sheet Metal (3.8.1)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/S 1Toolbar.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/S 2Toolbar.png})
Standard	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/S 1Toolbar.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/S 2Toolbar.png})
Tools	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/T
View (6.31.1)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/V
Workspaces (1.13.10)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/W

Toolbar Flyouts

On some tool buttons a small black arrow at the bottom right corner of the icon indicates a flyout is available. A flyout holds a group of related tools. Press and hold the left mouse button to expand the flyout and choose one of the flyout tools. The tool that was last chosen remains visible in the collapsed toolbar. Each of the flyouts can be opened as a separate toolbar. Drag the button of a toolbar flyout into the drawing area to open it as a floating toolbar.

Parent	Flyout	
2D Constraints (1.15.7)	Dimensional Constraints (1.15.6)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Geometric Constraints (1.15.8)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Assembly (3.6.1)	Assembly Visualization (3.6.3)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Assembly Explode (6.8.6)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
BIM	BIM Circles (6.9.20)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 CirclesToolbar.png})
	BIM Classify (6.7.5)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 ClassifyToolbar.png})
	BIM Connect (2.4.4)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 ConnectToolbar.png})
	BIM Databases (2.6.3)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 DatabasesToolbar.png})
	BIM Grids (6.7.12)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 GridsToolbar.png})
	BIM Lines (1.17.33)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 LinesToolbar.png})
	BIM Manipulate (6.7.8)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 ManipulateToolbar.png})
	BIM Polygons (6.24.43)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 PolygonsToolbar.png})
	BIM Propagate (6.7.20)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 PropagateToolbar.png})
	BIM Rooms (6.7.26)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 RoomsToolbar.png})
	BIM Sections (6.7.27)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 SectionsToolbar.png})
	BIM Site Design (6.27.41)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 SiteDesignToolbar.png})
	BIM Curtain Wall (6.7.7)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 CurtainWallToolbar.png})
	BIM Structural (6.7.15)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 StructuralToolbar.png})
	BIM Tag (6.7.33)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 TagToolbar.png})

	BIM Visibility (6.16.11)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 VisibilityToolbar.png})
	BIM Windows (6.7.36)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 WindowsToolbar.png})
Dimensions	Center Marks and Lines (1.21.11)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Direct Modeling (1.18.2)	Deform (1.6.10)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Draw (1.17.2)	Lines (1.17.9)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	polylines (1.17.12)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Polygon (1.17.20)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Boundaries	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Meshes	3D Mesh Faces	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	3D Mesh Surfaces	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Modify (1.23.1)	Arrays (1.23.19)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Parametric (6.24.6)	Design Table (6.10.26)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Rendering (1.26.2)	Lights (1.26.4)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Settings Entity Properties	Layers (1.16.2)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Sheet Metal (3.8.1)	Bend (6.28.2)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Flange (6.28.13)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Form (6.28.16)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Junction (6.28.18)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Relief (6.28.22)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Repair (6.28.24)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Split (6.28.28)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Unfold (6.28.29)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Tools	Attributes (1.22.11)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Drawing Views	Detail View Types (6.31.10)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Section Types (6.31.10)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
View	Hide/Show Entities (1.16.3)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Real-Time Motion (1.16.8)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Redraw/Regen (1.16.4)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
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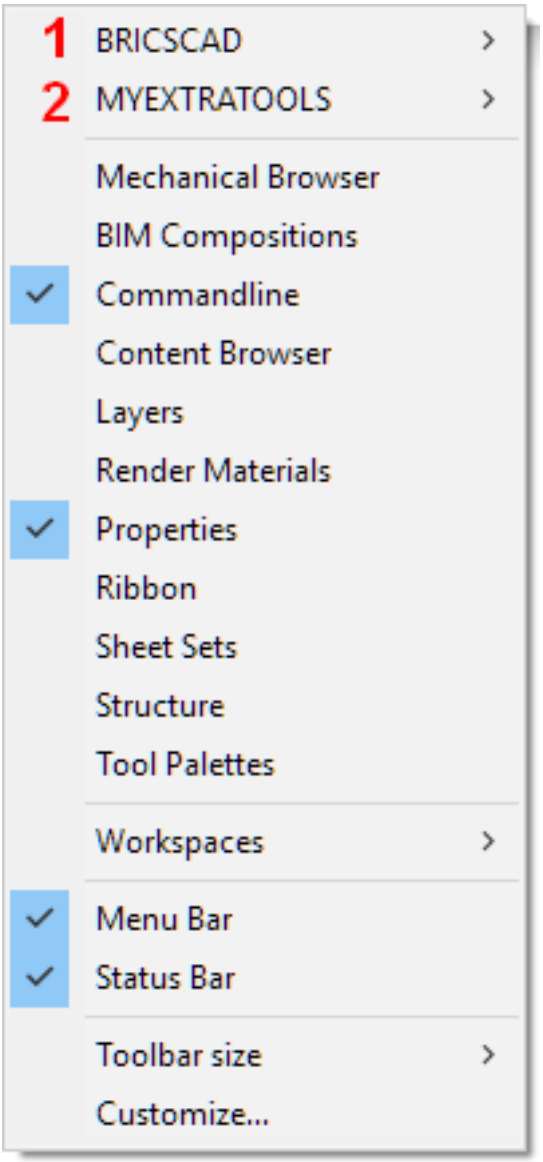
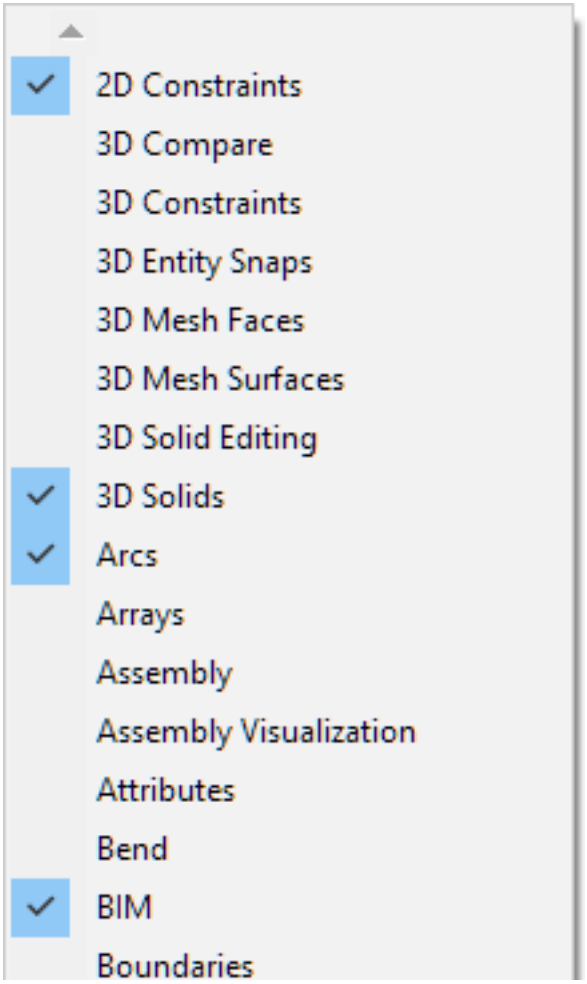
Figure 1.160



Opening a toolbar

1. In **Windows** : Hover over a toolbar or a blank portion of the toolbar area, then right click. In **Linux** : Hover over a blank portion of the menu bar, status bar or toolbar area, then right click. A context menu displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/01Context.png}) (1) Main CUI file (1.13.4). (2) Partial CUI file(s) (1.13.4)
2. In the context menu choose either the Main CUI file (e.g. BRICSCAD) or one of the Partial CUI files (if any). A list of available toolbars in the selected CUI file displays. Toolbars that are already open are checked. If the list does not fit on the screen, click the arrow keys on the top and the bottom to scroll.(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/01ToolbarsContext.png})
3. Select the toolbar on the context menu. The toolbar opens.
4. (option) Dock the toolbar by dragging the toolbar by its title bar to one of the edges of the BricsCAD application window.

Figure 1.161


2D Constraints
3D Compare
3D Constraints
3D Entity Snaps
3D Mesh Faces
3D Mesh Surfaces
3D Solid Editing
3D Solids
Arcs
Arrays
Assembly
Assembly Visualization
Attributes
Bend
BIM
Boundaries

NOTES

- If you select a toolbar that was already open, it will be closed.
- To prevent a toolbar from docking, press and hold the Ctrl key while dragging the toolbar.
- Drag a toolbar by the grip at its left edge (when docked horizontally) () or top edge (when docked vertically) edge to undock.
- To dock a toolbar vertically in **Linux** , the **Position** property of the toolbar must be set to **Left** or **Right** .
- The LOCKUI (1.12.2) system variable allows to lock the location and size of toolbars and dockable panels (1.12.7). Press and hold the Ctrl-key to override. Click the **LOCKUI** field in the Status Bar (1.12.2) to toggle the locking of the user interface.

To tear off a flyout toolbar

1. Press and hold the left mouse button to drag the visible icon of a flyout toolbar into the drawing area.
2. Release the left mouse button to place the toolbar.

Closing a toolbar

1. In **Windows** : Place the cursor on a toolbar, then right click. In **Linux** : Place the cursor on the menu bar, then right click. A context menu displays.
2. Choose **BRICSCAD** on the context menu. Toolbars that are open are checked.
3. In the toolbar list, uncheck the toolbar you want to close.

NOTE Floating toolbar can be closed by clicking the **Close** button ().

To set the position and visible property of a toolbar

The **Visible** property determines whether a toolbar is visible when a CUI-file is loaded. The **Position** property determines where a toolbar displays when a CUI-file is loaded.

1. In the Customize dialog (1.13.3) click the **Toolbars** tab.
2. If necessary, expand the **BricsCAD** menu group.
3. Select the toolbar.
4. Choose **Position** in the **Properties** grid on the **Customize** dialog.
5. Click the settings field and choose the desired position in the list. The options are: **Floating** , **Top** , **Left** , **Bottom** , **Right**
6. Choose **Visible** in the **Properties** grid on the **Customize** dialog.
7. Click the settings field and choose either **Show** or **Hide** in the options list.

Figure 1.162



Figure 1.163

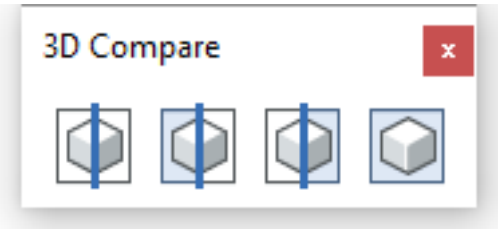


Figure 1.164



Figure 1.165

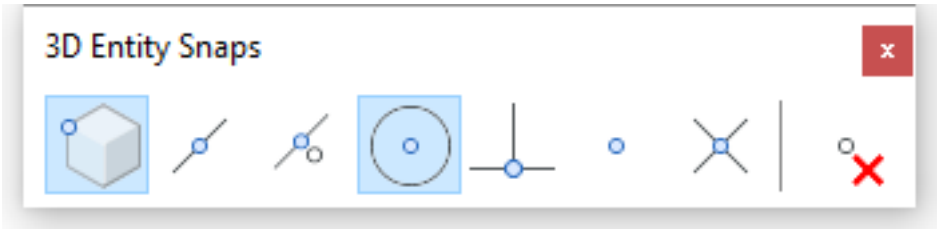


Figure 1.166



Figure 1.167



Figure 1.168

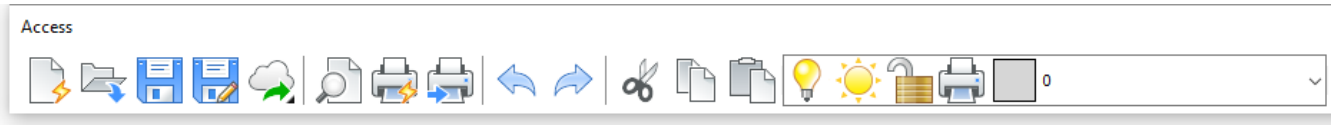


Figure 1.169



Figure 1.170



Figure 1.171



Figure 1.172

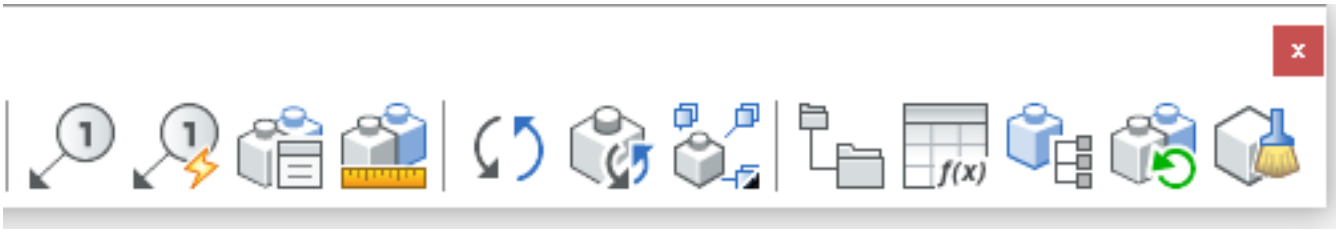


Figure 1.173



Figure 1.174



Figure 1.175

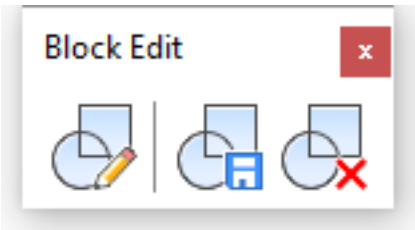


Figure 1.176

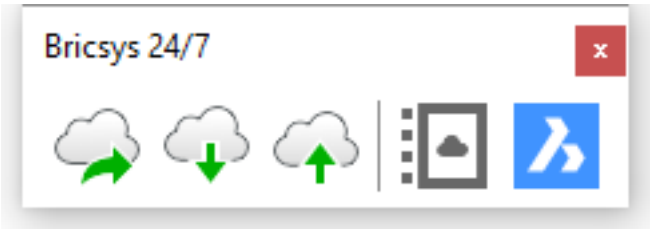


Figure 1.177



Figure 1.178

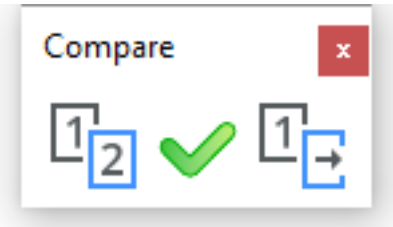


Figure 1.179



Figure 1.180



Figure 1.181



Figure 1.182



Figure 1.183



Figure 1.184



Figure 1.185

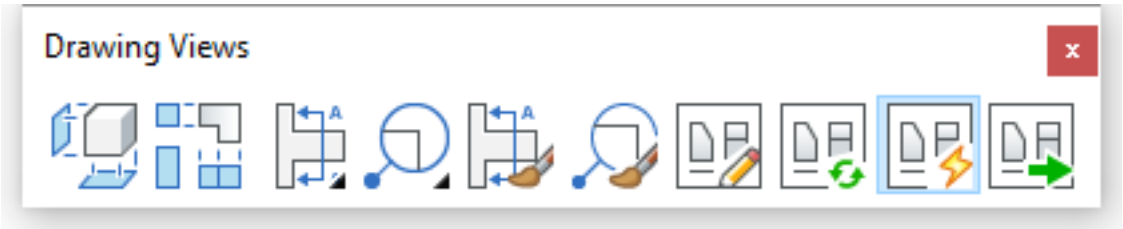


Figure 1.186



Figure 1.187



Figure 1.188

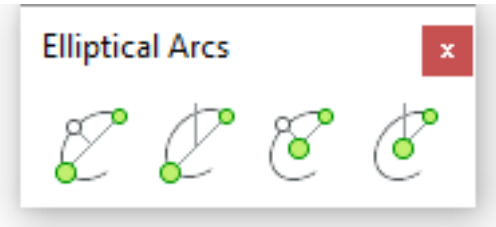


Figure 1.189

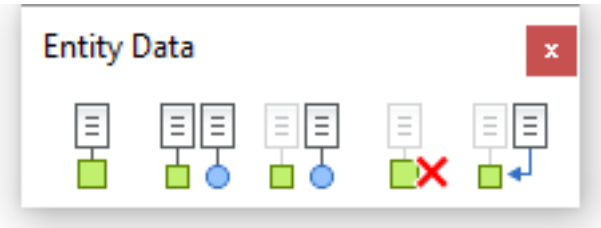


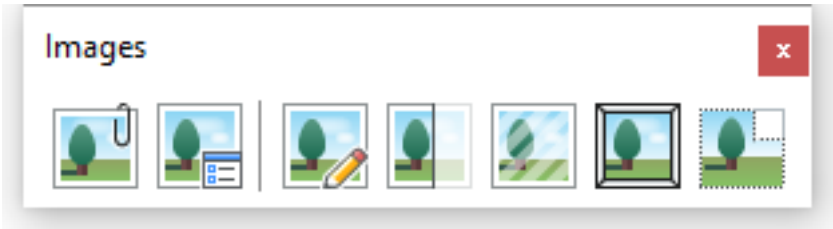
Figure 1.190



Figure 1.191



Figure 1.192



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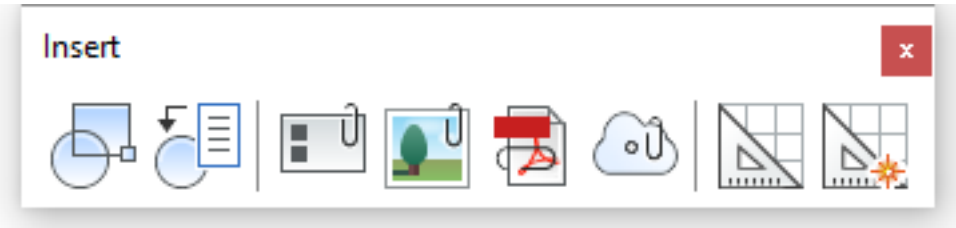


Figure 1.194

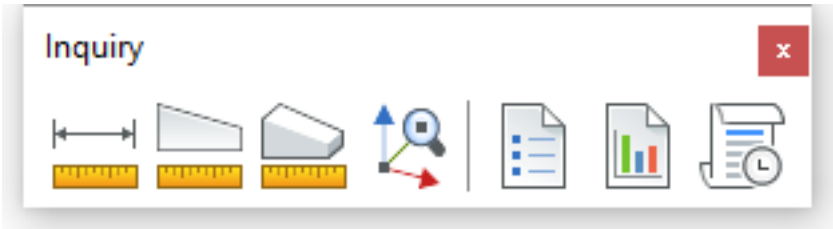


Figure 1.195

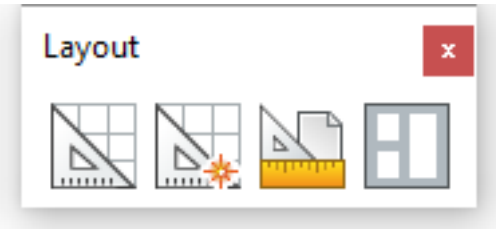


Figure 1.196



Figure 1.197

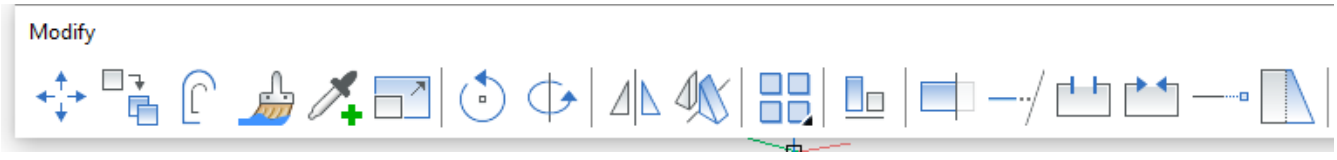


Figure 1.198



Figure 1.199

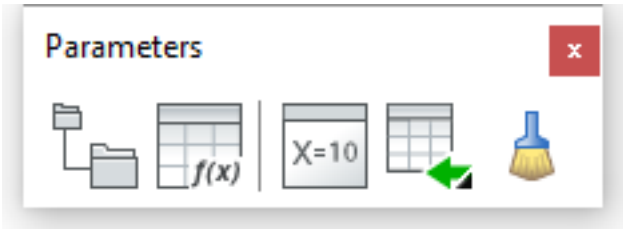


Figure 1.200

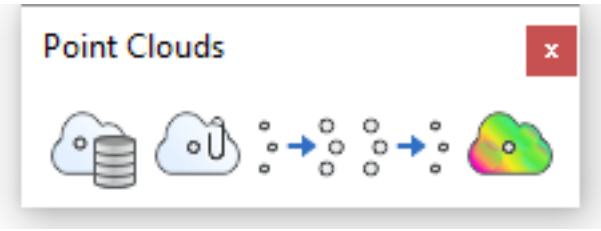


Figure 1.201



Figure 1.202

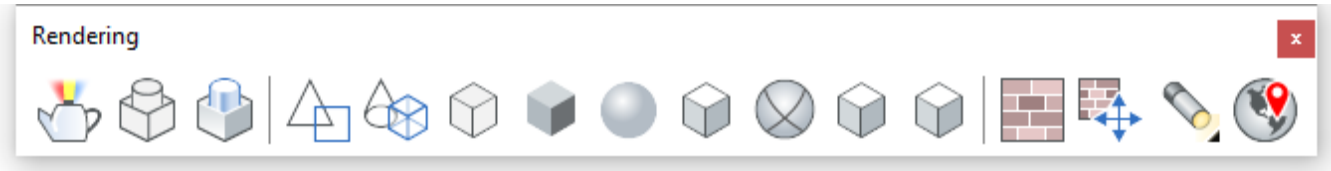


Figure 1.203



Figure 1.204

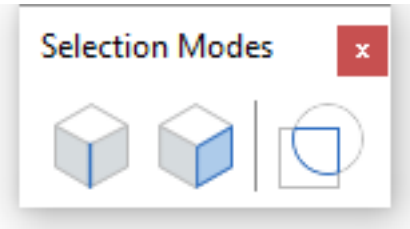


Figure 1.205



Figure 1.206



Figure 1.207



Figure 1.208



Figure 1.209

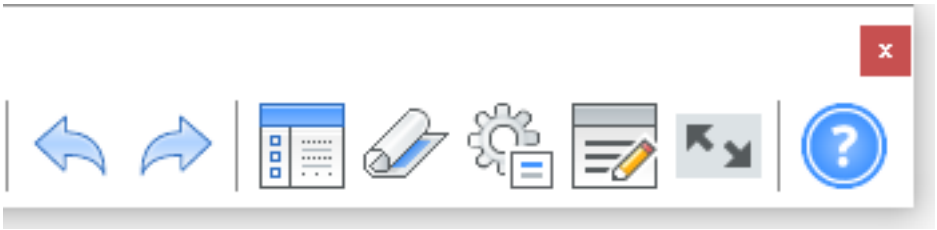
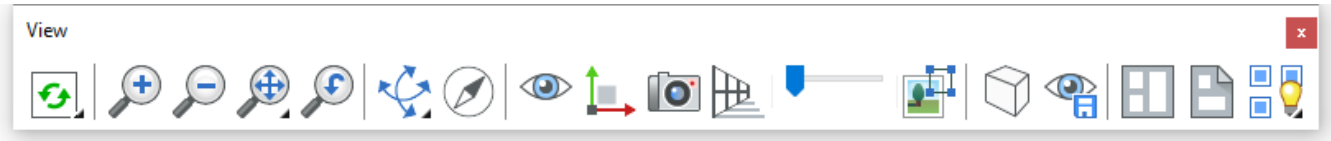


Figure 1.210



Figure 1.211



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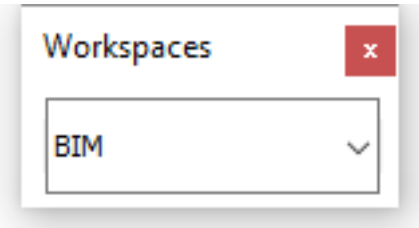
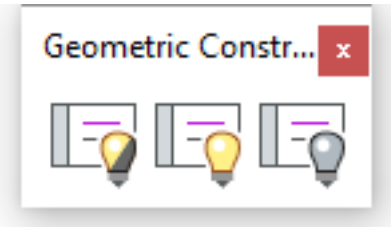


Figure 1.213



Figure 1.214



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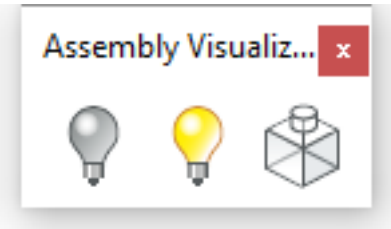
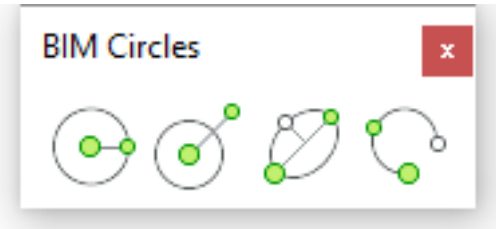


Figure 1.216



Figure 1.217



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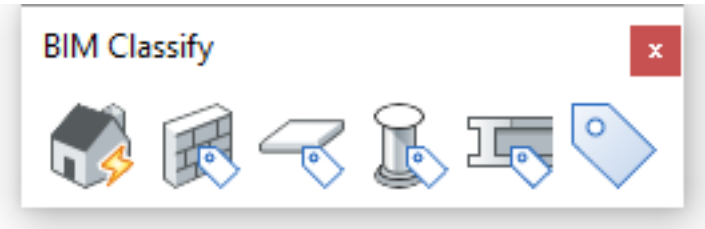


Figure 1.219



Figure 1.220

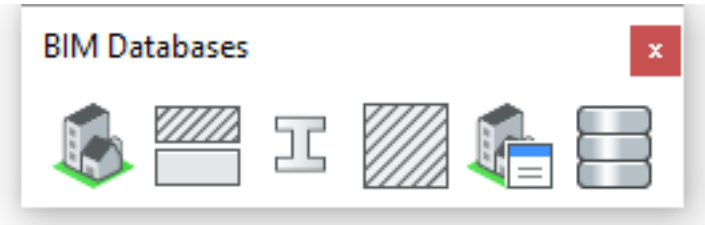


Figure 1.221

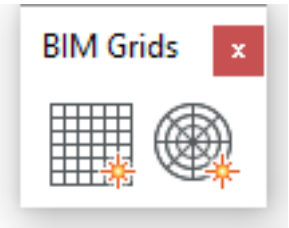


Figure 1.222



Figure 1.223

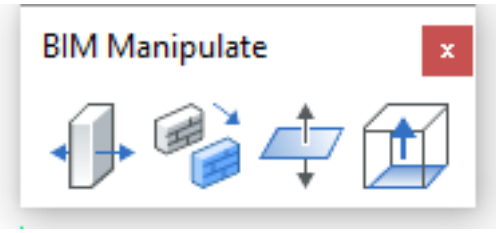


Figure 1.224

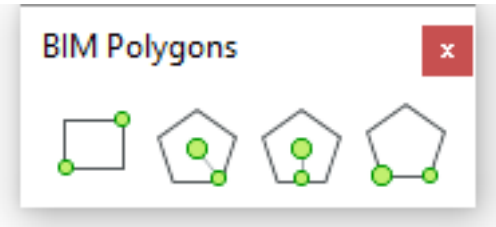


Figure 1.225



Figure 1.226

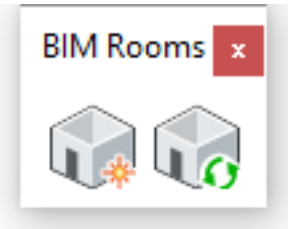


Figure 1.227



Figure 1.228



Figure 1.229

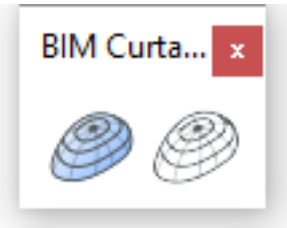


Figure 1.230

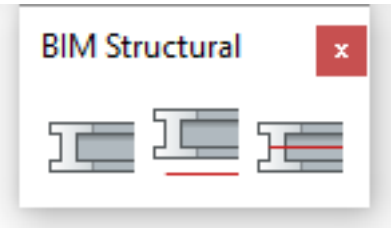


Figure 1.231

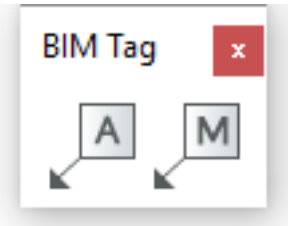


Figure 1.232

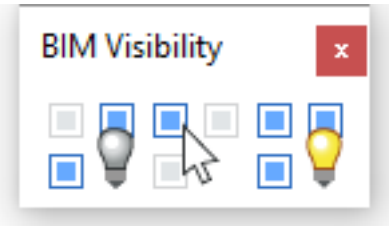


Figure 1.233

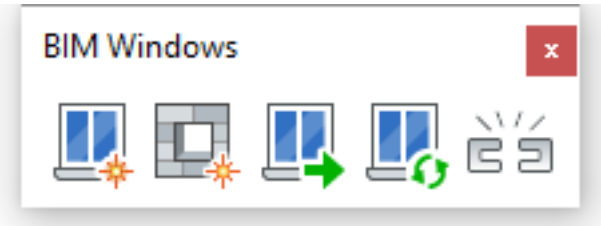


Figure 1.234

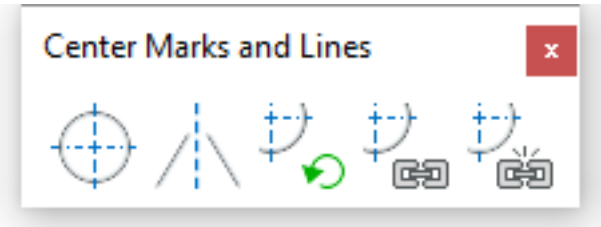


Figure 1.235



Figure 1.236



Figure 1.237

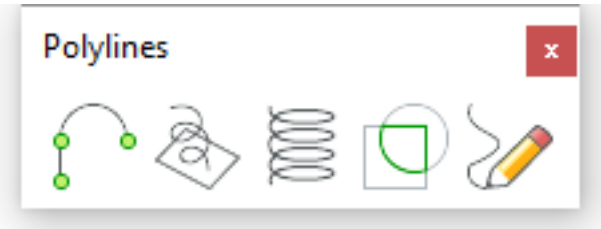


Figure 1.238

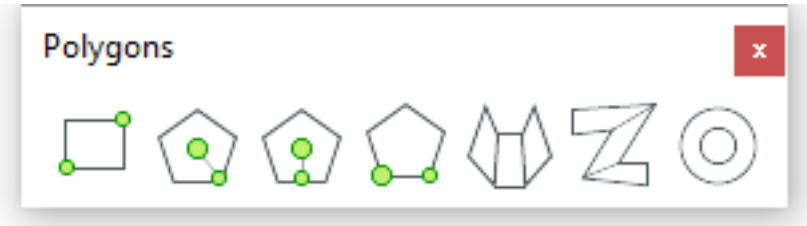


Figure 1.239

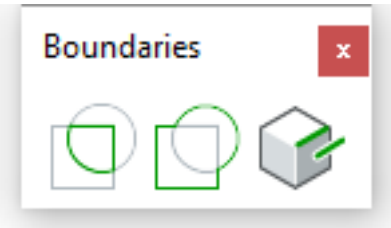


Figure 1.240

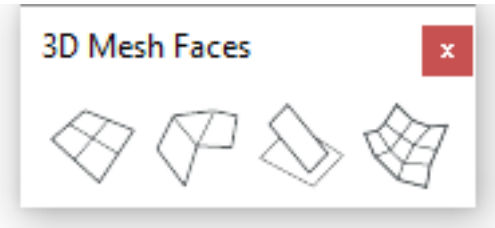


Figure 1.241

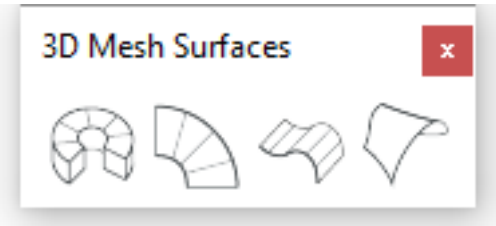


Figure 1.242



Figure 1.243

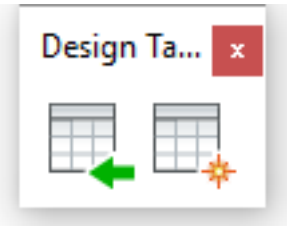


Figure 1.244

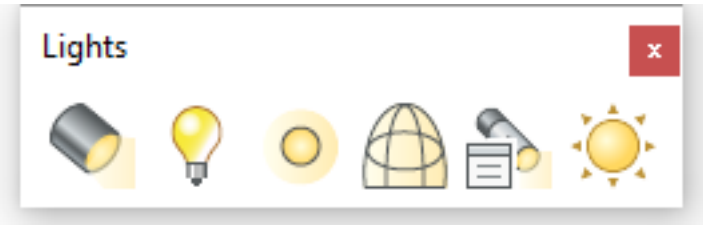


Figure 1.245



Figure 1.246

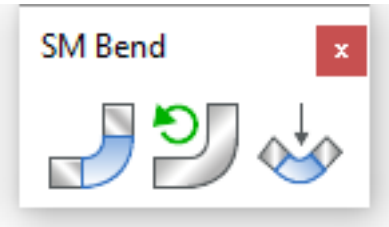


Figure 1.247

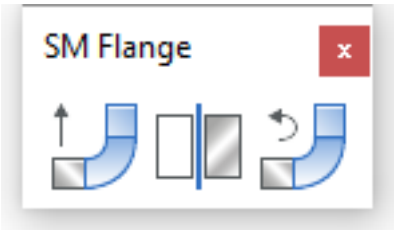


Figure 1.248



Figure 1.249

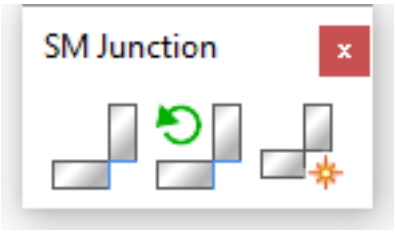


Figure 1.250



Figure 1.251

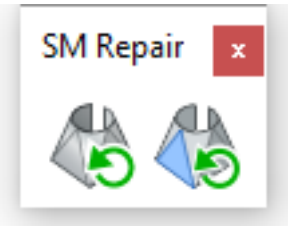


Figure 1.252

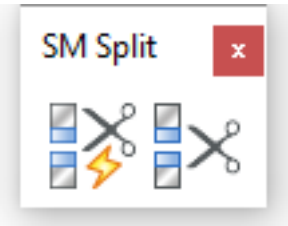


Figure 1.253

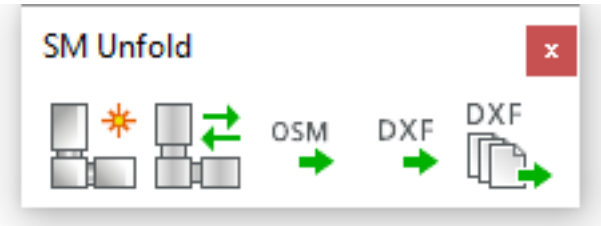


Figure 1.254



Figure 1.255

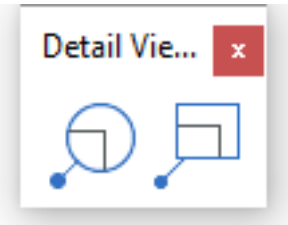


Figure 1.256

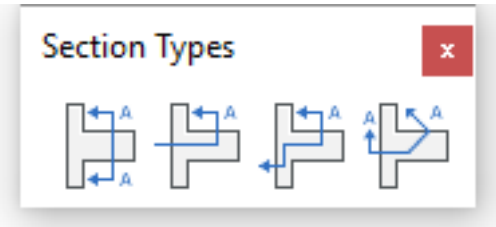


Figure 1.257

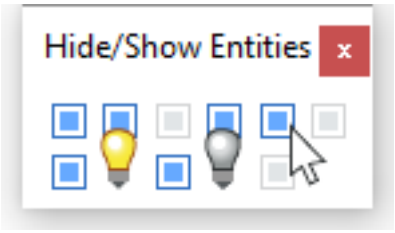


Figure 1.258



Figure 1.259

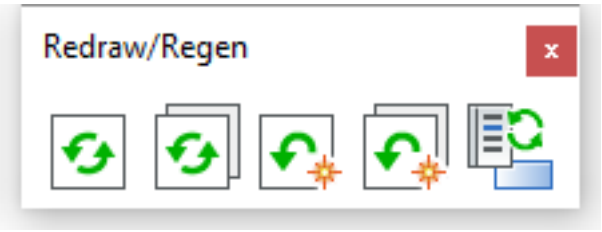


Figure 1.260



1.12.4 Properties Panel

Command: PROPERTIES (6.24.49) In the **Properties** panel you can:

- Set the current properties: **color** , **layer** , **linetype** , **linetypescale** and **lineweight**
- Edit the properties of a single entity
- Edit the shared properties of a selection set
- Edit endpoints of lines and vertices of polylines graphically
- Compose a selection set using Quick Select (1.23.7).

To open the Properties panel

Do one of the following:

- Click the **Properties** tool button ( ) on the **Standard** toolbar. The pressed state of the tool button indicates that the **Properties** panel is currently open. Click the tool button again to close the **Properties** panel.
- Right click when the cursor is on a toolbar or ribbon panel. A context menu displays. The marked items in the context menu are currently open. Select **Properties** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/DockablePanelsList-Properties.png})
- Double click an entity, such as lines, circles, arcs, 3D solids ... The **Properties** panel opens, showing the properties of the selected entity. Double clicking entities such as texts, hatches, blocks or polylines start an editing command (See the **Mouse** tab on the Customize (1.13.3) dialog). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/

- Choose **Properties** in the **Modify** menu.
- Type **properties** in the command window, then press Enter.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e**No selection** The current entity properties, current Elevation (6.13.4), View properties and Annotation scale (1.21.28) display	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e**Single entity selected** The properties of the selected entity display	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e**Multiple entities selected** The shared properties of the selected entities display. *Varies* indicates that the shared property has different values.

NOTE	The **Properties** panel is dockable panel (1.12.7). To dock the **Properties** panel, drag it by its title bar to either the left or right hand side of the BricsCAD application window or onto another dockable panel.

NOTE	Read-only fields display in grey. The content of a read-only field can be copied though. To copy a read-only field, click the field, then press Ctrl C.

Figure 1.261

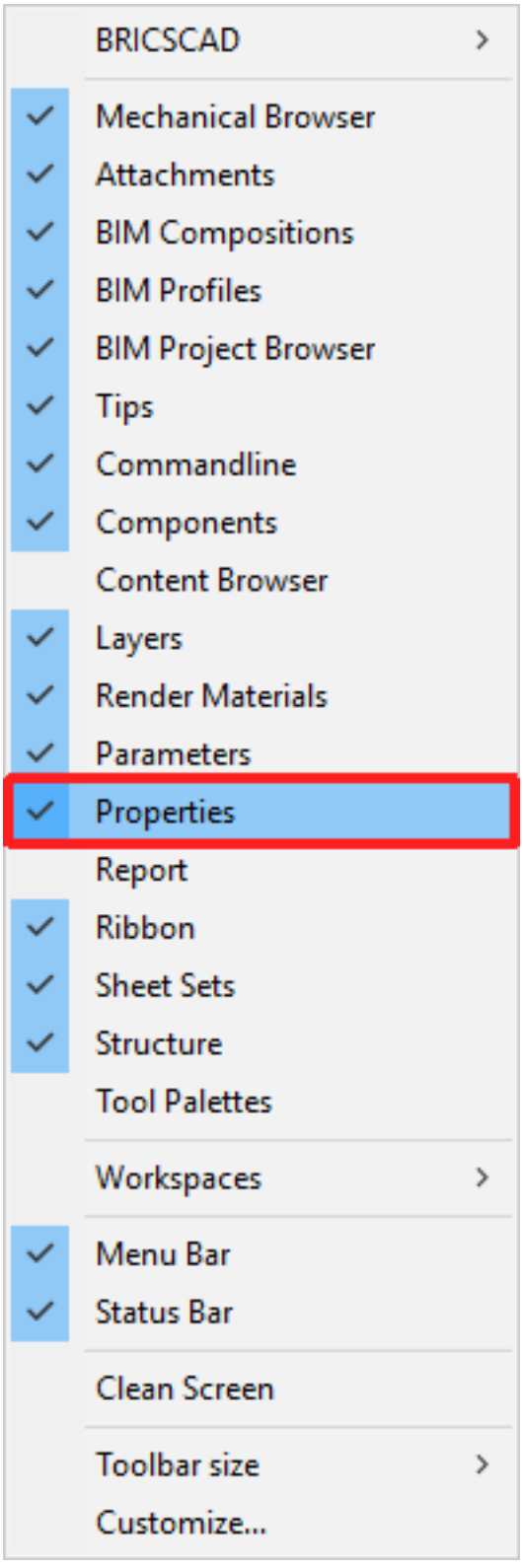
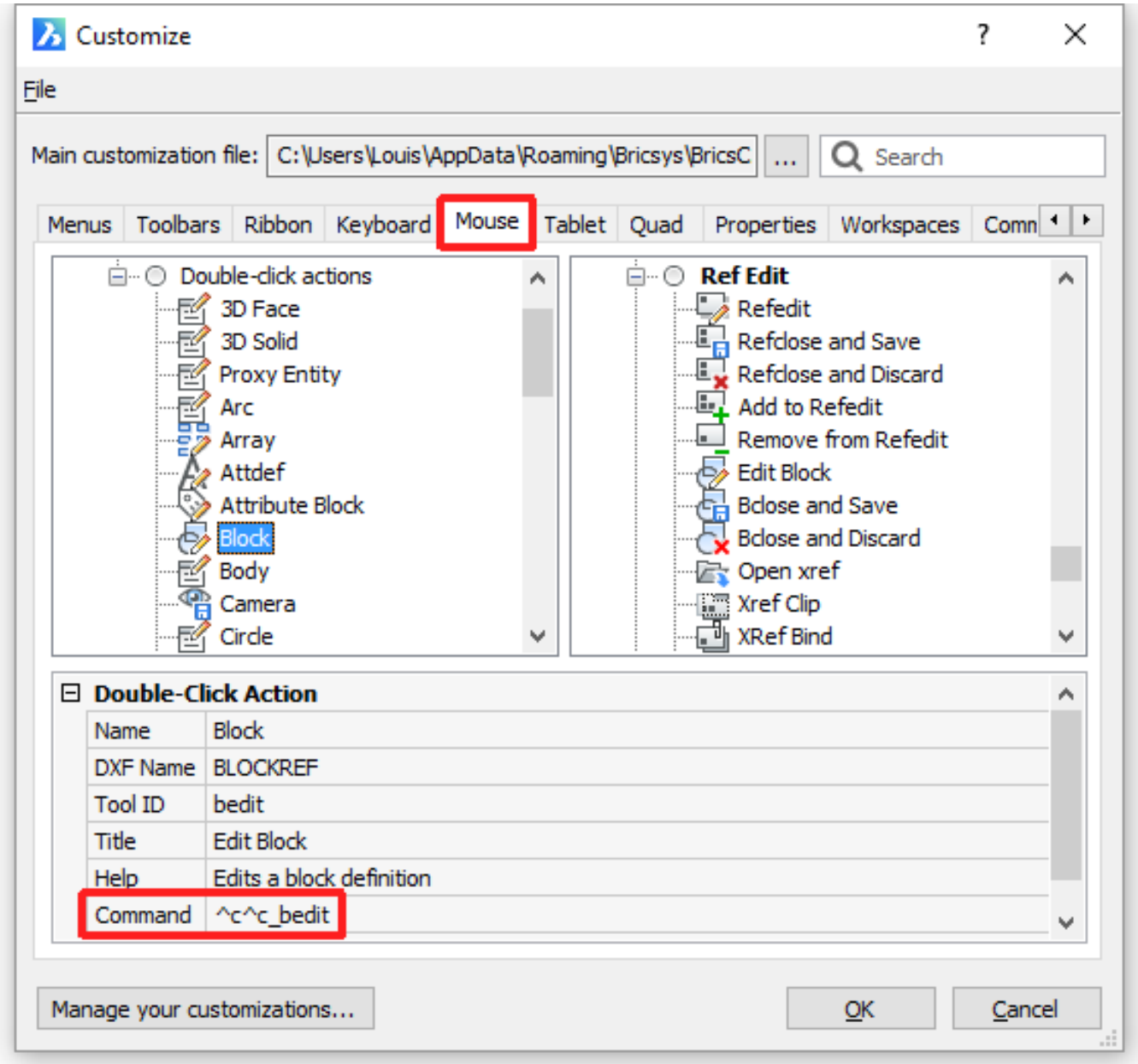

BRICSCAD
Mechanical Browser
Attachments
BIM Compositions
BIM Profiles
BIM Project Browser
Tips
Commandline
Components
Content Browser
Layers
Render Materials
Parameters
Properties
Report
Ribbon
Sheet Sets
Structure
Tool Palettes
Workspaces
Menu Bar
Status Bar
Clean Screen
Toolbar size
Customize...

Figure 1.262

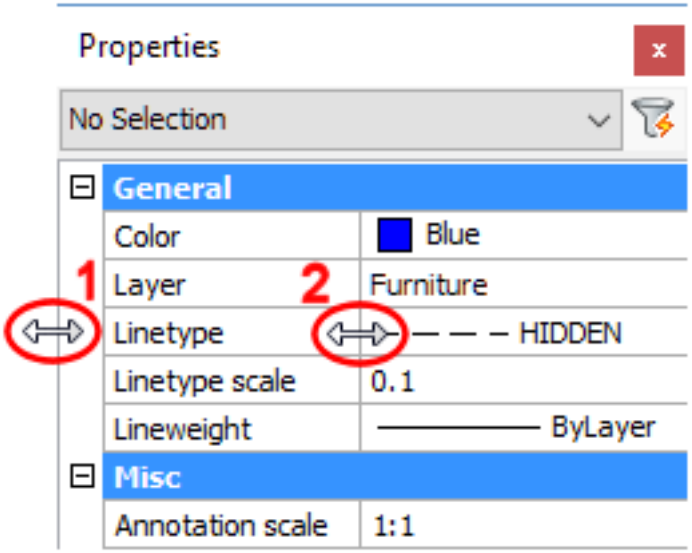

Customize
File
Main customization file:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsC
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Properties
Workspaces
Comn
Double-click actions
3D Face
3D Solid
Proxy Entity
Arc
Array
Attdef
Attribute Block
Block
Body
Camera
Circle
Ref Edit
Refedit
Refclose and Save
Refclose and Discard
Add to Refedit
Remove from Refedit
Edit Block
Bclose and Save
Bclose and Discard
Open xref
Xref Clip
XRef Bind
Double-Click Action
Name
Block
DXF Name
BLOCKREF
Tool ID
bedit
Title
Edit Block
Help
Edits a block definition
Command
^c^c_bedit
Manage your customizations...
OK
Cancel

To adjust the size of the Properties panel

1. Move the cursor over one of the edges of the **Properties** (1). The cursor turns into a double-headed arrow.
2. Press and hold the left mouse button to drag the edge of the **Properties** . (\vref {fig:C:/Dev/Task-2448/images/staticbrics
3. Move the cursor over the boundary between the **Setting Names** and the **Setting Fields** columns (2). The cursor turns into a double-headed arrow.
4. Press and hold the left mouse button to drag the boundary.

NOTE	When docked, only the left (or right) edge of the **Properties** is adjustable.

Figure 1.263



Setting the current properties in the Properties panel

1. (option) Click on **Color,** then click the down arrow button to select a color.
2. (option) Click on **Layer,** then click the down arrow to select a layer.
3. (option) Click on **Linetype** . then click on the down arrow to select a linetype.
4. (option) Click on **Linetype scale** , then type the new value in the **Linetype scale** field.
5. (option) Click on **Lineweight**, then click on the down arrow to select a lineweight.

Edit the properties of a single entity

1. Select the entity. The entity's properties display in the **Properties** panel.. (\vref {fig:C:/Dev/Task-2448/images/staticbrics
2. Click the property you want to modify. The settings field of the selected property is activated.
3. Type a new value in the settings field of the selected property or choose a setting from the list box, then press the **Enter** key or select another property. The entity is updated.
4. (option) repeat steps 2 and 3 to modify another property.
5. Press the **Escape** key to stop.

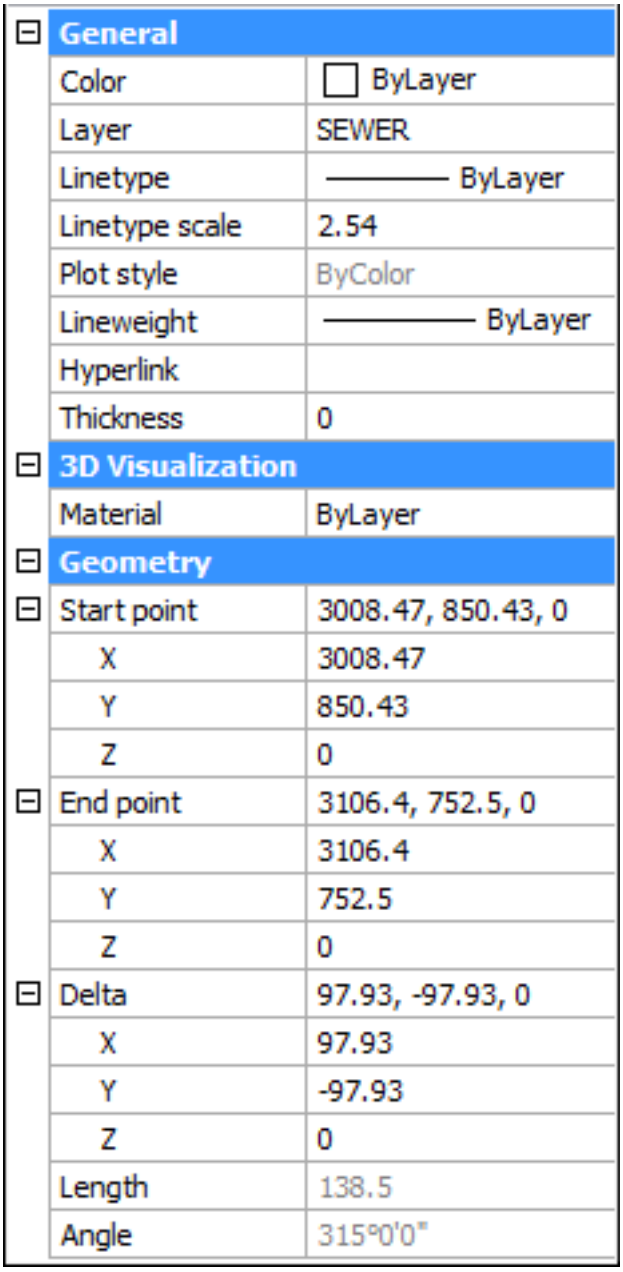
NOTES

- Properties of which the value displays in grey cannot be modified.
- Step 3: Properties (such as the General properties), which are chosen from a list are updated instantly.

To edit the endpoints of a line

1. Select the line.
2. On the **Properties Bar**, under **Geometry**, do one of the following:
   - Select the X-, Y- or Z- field, then enter a value.
   - Select either **Start point** or **End point**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD
3. An X indicates the point being edited.
   - Click the **Pick Point** button ( ).
   - Specify the point in the drawing.

Figure 1.264

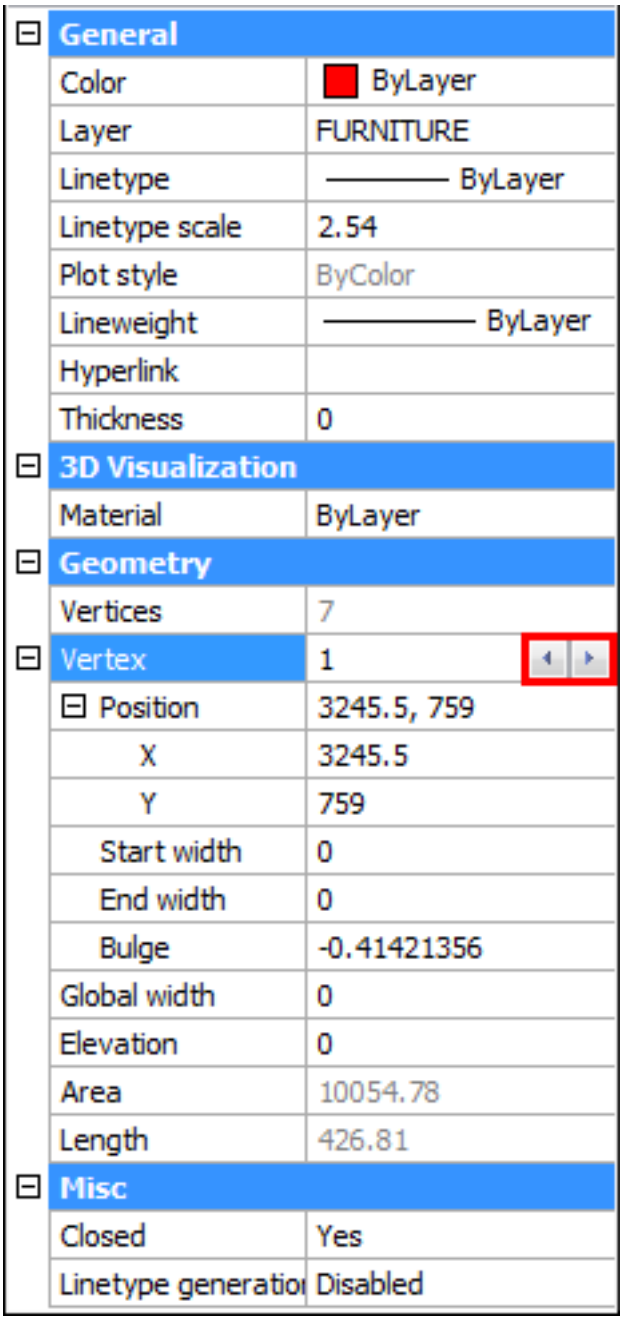


General	
Color	ByLayer
Layer	SEWER
Linetype	ByLayer
Linetype scale	2.54
Plot style	ByColor
Lineweight	ByLayer
Hyperlink	
Thickness	0
3D Visualization	
Material	ByLayer
Geometry	
Start point	3008.47, 850.43, 0
X	3008.47
Y	850.43
Z	0
End point	3106.4, 752.5, 0
X	3106.4
Y	752.5
Z	0
Delta	97.93, -97.93, 0
X	97.93
Y	-97.93
Z	0
Length	138.5
Angle	315°0'0"

To edit the vertices of a polyline

1. Select the polyline.
2. On the **Properties Bar** under **Geometry** , select **Vertex**.
3. Click the **Next/Previous** arrow buttons to select a vertex. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
   An X indicates the vertex being edited.
4. Do one of the following:
   - Select the X- or Y- field, then enter a value.
   - Under **Vertex** select **Position** and click the **Pick Point** button (), then specify the point in the drawing.
5. Press the **Escape** key to stop.

Figure 1.265


General
Color
ByLayer
Layer
FURNITURE
Linetype
ByLayer
Linetype scale
2.54
Plot style
ByColor
Lineweight
ByLayer
Hyperlink
Thickness
0
3D Visualization
Material
ByLayer
Geometry
Vertices
7
Vertex
1
Position
3245.5, 759
X
3245.5
Y
759
Start width
0
End width
0
Bulge
-0.41421356
Global width
0
Elevation
0
Area
10054.78
Length
426.81
Misc
Closed
Yes
Linetype generatio
Disabled

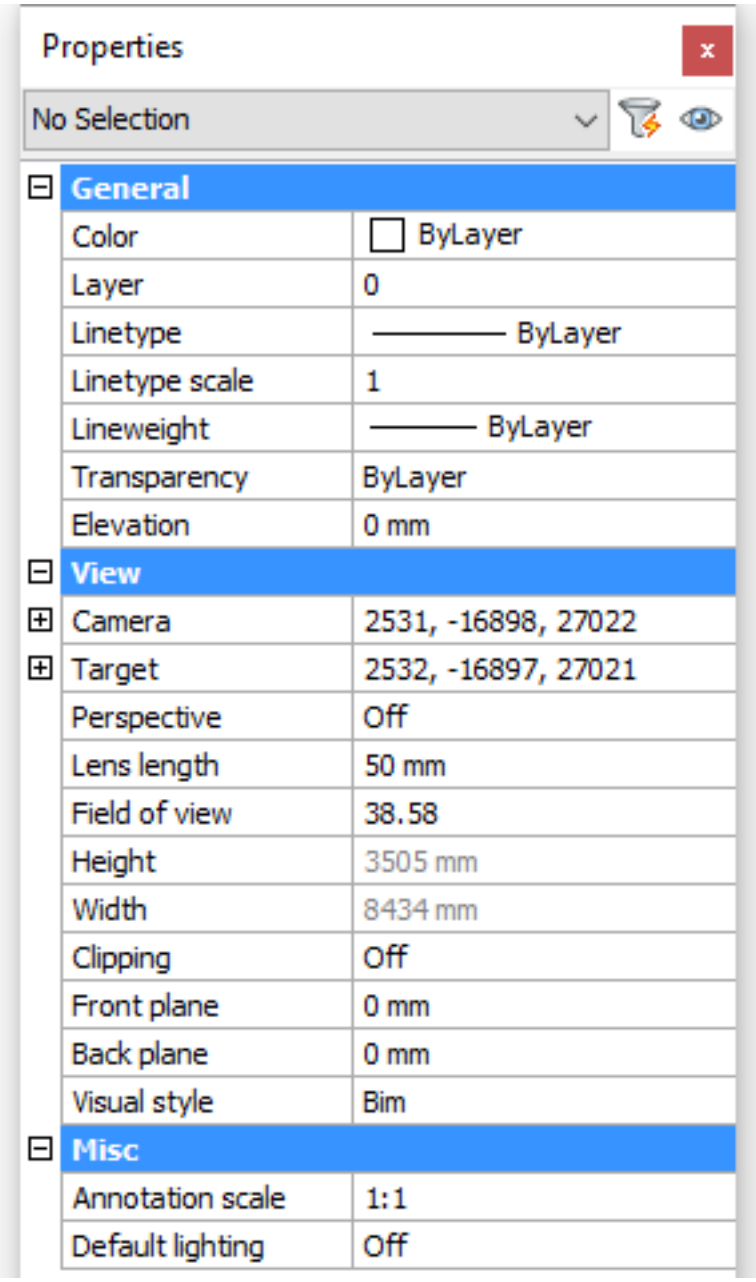
Edit the shared properties of a selection set

1. Select the entities. The shared properties display in the **Properties Bar**.
2. Click the property you want to modify. The settings field of the selected property is activated.
3. Type a new value in settings field of the selected property or choose a setting from the list box, then press the **Enter** key or select another property. All selected entities are updated simultaneously.
4. (option) Repeat steps 2 and 3 to modify another property.
5. Press the **Escape** key to stop.

NOTES

- *Varies* displays for shared properties which are defined differently. If you edit such property, all entities in the selection set will be equally defined for this property.
- Step 3: Properties (such as the General properties), which are chosen from a list are updated instantly.

Figure 1.266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General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Lineweight
ByLayer
Transparency
ByLayer
Elevation
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View
Camera
2531, -16898, 27022
Target
2532, -16897, 27021
Perspective
Off
Lens length
50 mm
Field of view
38.58
Height
3505 mm
Width
8434 mm
Clipping
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Front plane
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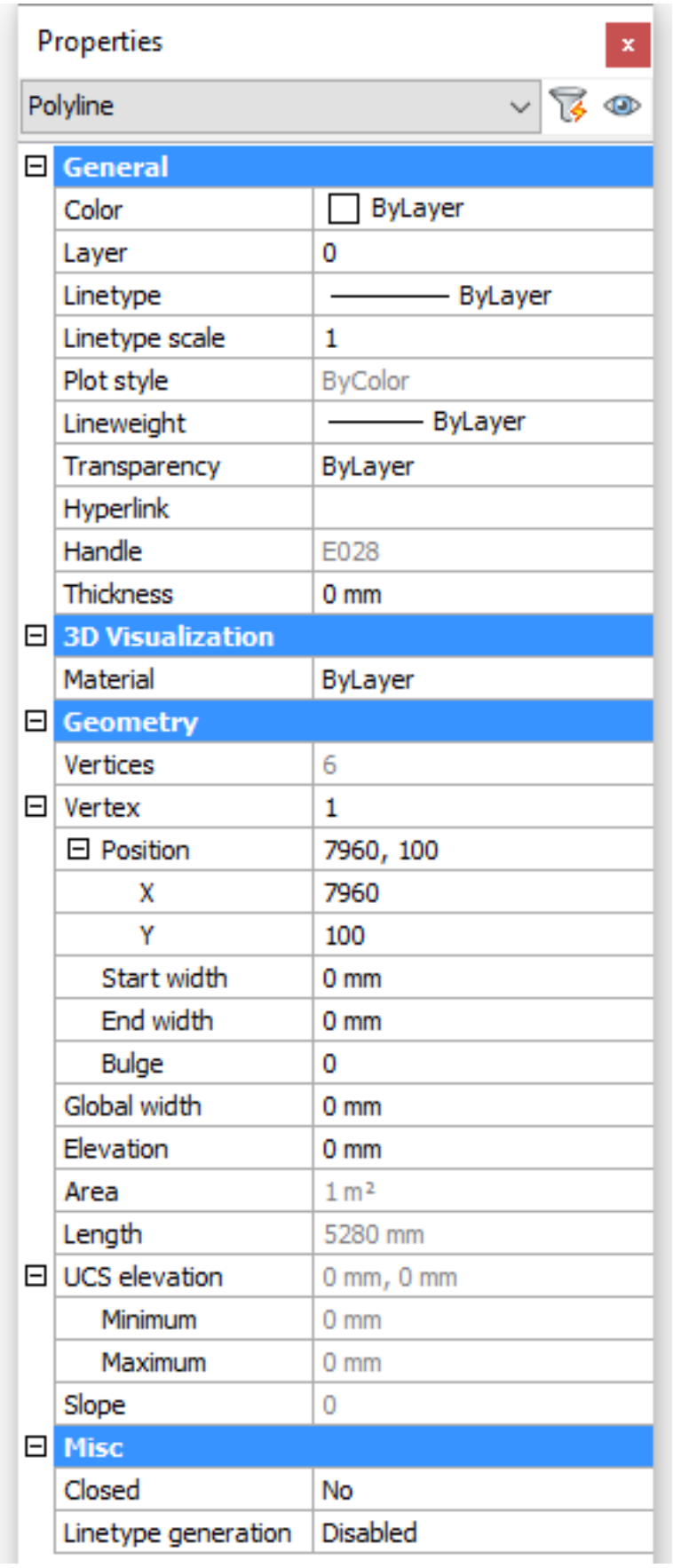
Back plane
0 mm
Visual style
Bim
Misc
Annotation scale
1:1
Default lighting
Off

Figure 1.267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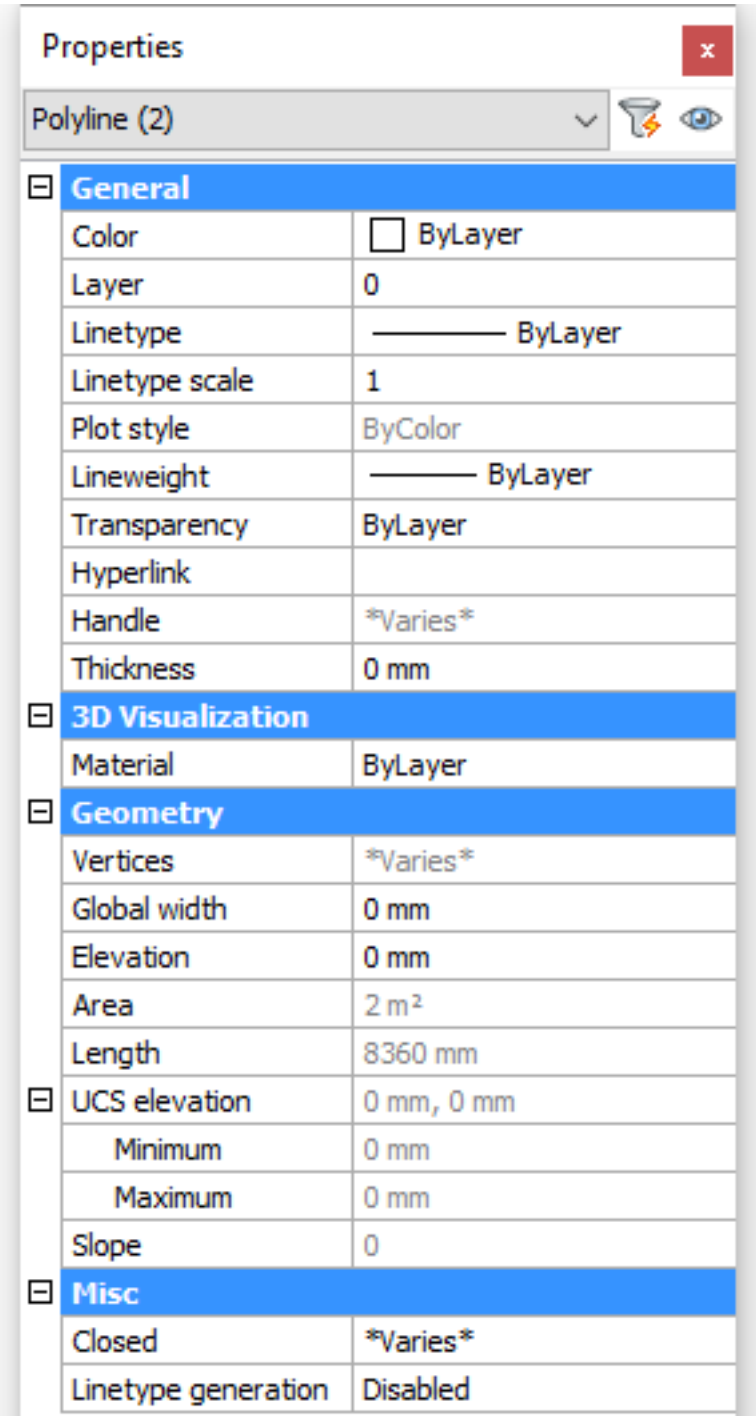
Polyline
General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Plot style
ByColor
Lineweight
ByLayer
Transparency
ByLayer
Hyperlink
Handle
E028
Thickness
0 mm
3D Visualization
Material
ByLayer
Geometry
Vertices
6
Vertex
1
Position
7960, 100
X
7960
Y
100
Start width
0 mm
End width
0 mm
Bulge
0
Global width
0 mm
Elevation
0 mm
Area
1 m²
Length
5280 mm
UCS elevation
0 mm, 0 mm
Minimum
0 mm
Maximum
0 mm
Slope
0
Misc
Closed
No
Linetype generation
Disabled

Figure 1.268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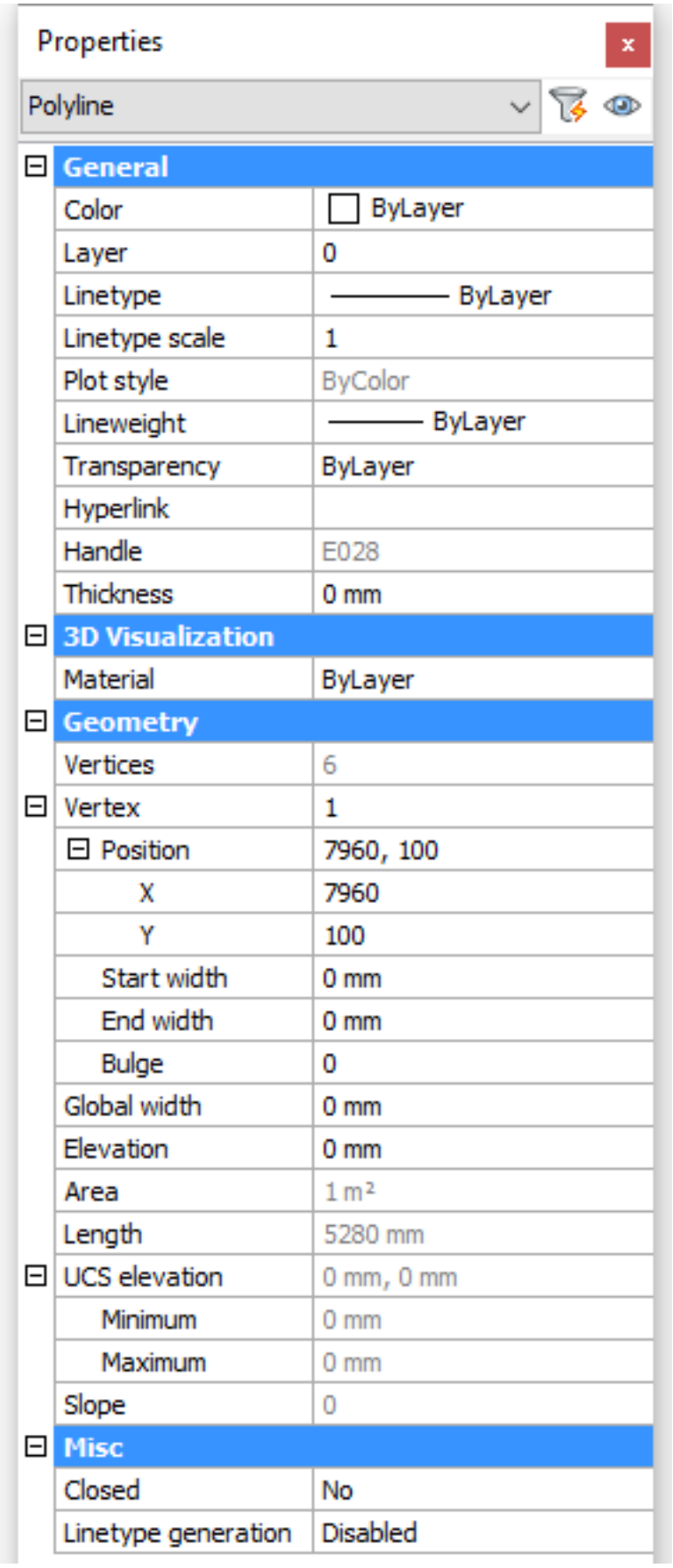
Polyline (2)
General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Plot style
ByColor
Lineweight
ByLayer
Transparency
ByLayer
Hyperlink
Handle
Varies
Thickness
0 mm
3D Visualization
Material
ByLayer
Geometry
Vertices
Varies
Global width
0 mm
Elevation
0 mm
Area
2 m²
Length
8360 mm
UCS elevation
0 mm, 0 mm
Minimum
0 mm
Maximum
0 mm
Slope
0
Misc
Closed
Varies
Linetype generation
Disabled

Figure 1.269


Properties
Polyline
General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Plot style
ByColor
Lineweight
ByLayer
Transparency
ByLayer
Hyperlink
Handle
E028
Thickness
0 mm
3D Visualization
Material
ByLayer
Geometry
Vertices
6
Vertex
1
Position
7960, 100
X
7960
Y
100
Start width
0 mm
End width
0 mm
Bulge
0
Global width
0 mm
Elevation
0 mm
Area
1 m²
Length
5280 mm
UCS elevation
0 mm, 0 mm
Minimum
0 mm
Maximum
0 mm
Slope
0
Misc
Closed
No
Linetype generation
Disabled

1.12.5 The Quad cursor menu

The Quad cursor menu, aka the Quad is an alternative to grip-editing of entities, offering a (much) richer set of editing operations while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs. While the Quad was initially conceived for easy and fast editing of 3D solids, the Quad is extended to allow enhanced editing and creating of all entity types.

To activate the Quad

Activation of the Quad is controlled by the QUADDISPLAY (1.13.11) system variable. To toggle the Quad On/Off, do one of the following:

- Click the QUAD (1.12.2) field in the Status bar (1.12.2).
- Press the **F12** function key.
- Edit the **QUADDISPLAY** system variable in the Settings (1.13.11) dialog.
- Type **QuadDisplay** in the command bar, then choose an option.

Editing the Quad settings

1. Right click the **QUAD** field in the Status bar (1.12.2), then choose **Settings** in the context menu. The **Settings** dialog opens, showing the **Quad settings** : (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD

- **Quad display:** Determines when the Quad displays.
  - Negative: Switches the Quad off.
  - 1: When the cursor is over an entity. The entity under the cursor highlights.
  - 2: When entities are selected
  - 4: On right click when no entities are selected and there is no entity under the cursor.
  - 8: Suppress the quad when hovering over entities that are not in the current selection set; set Show **Quad on right click** ON to display the Quad.
- **Quad default command launch** : Sets the action to launch the default Quad command.
  - 0: Click the button.
  - 1: Right click
- **Quad popup corner** : Controls the popup position of the Quad relative to the cursor position: **Upper** , **Center** or **Middle Right** .
- **Quad width** : Defines the number of commands in a row. Default = 6 commands.
- **Quad icon size** : Small icons (16 x 16 pixels), Large icons (32 x 32 pixels) or Extra Large icons (64 x 64 pixels).

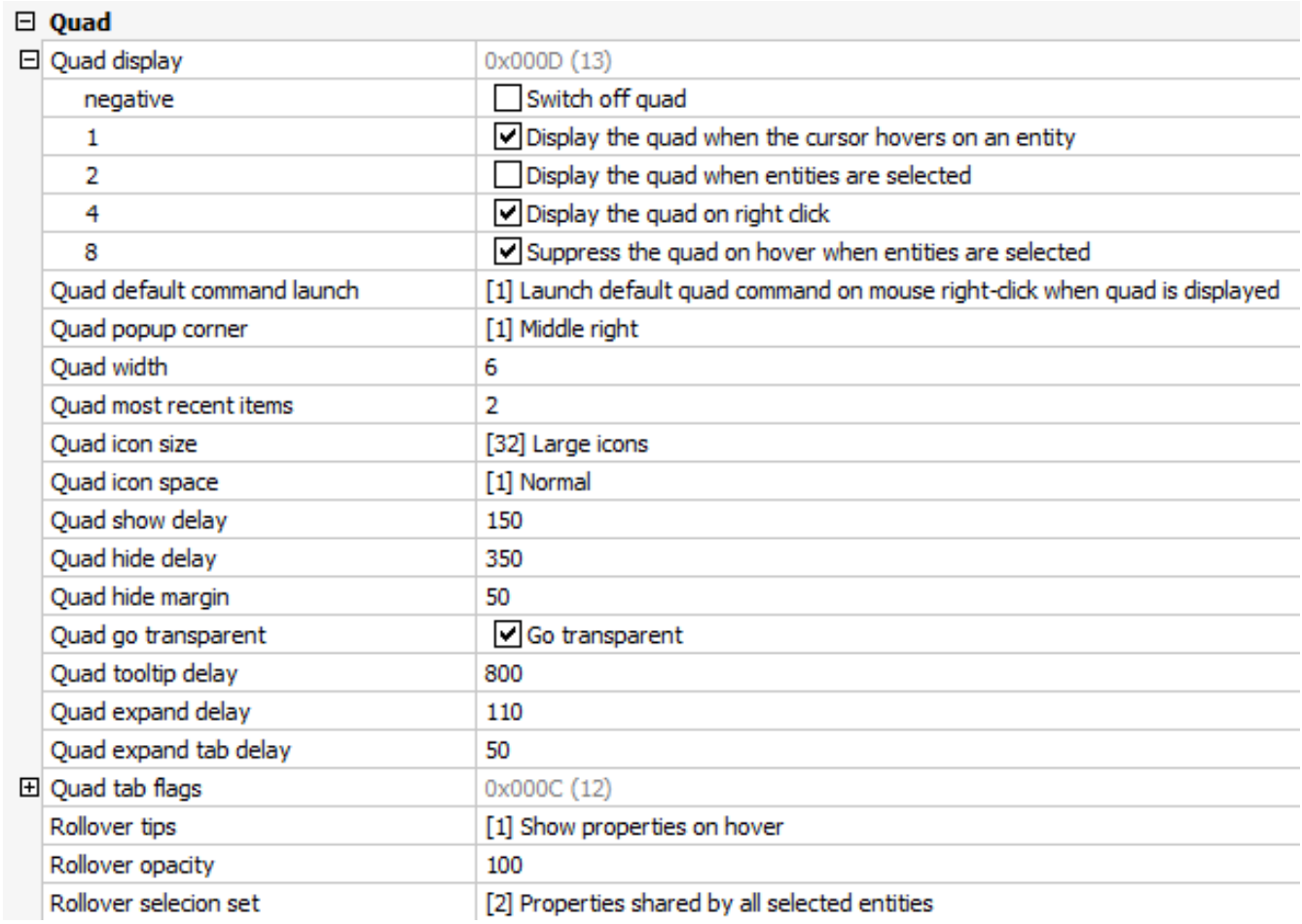
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadSmallIcons.png}) Small Icons	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadLargeIcons.png}) Large Icons
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadExtraLargeIcons.png}) Extra Large Icons	

- **Quad icon space** : Defines the space around the Quad tool buttons: narrow, normal or large

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadSmallIconsNarrow.png}) Narrow	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadSmallIconsNormal.png}) Normal	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadSmallIconsWide.png}) Wide

- **Quad show delay** : Sets the time lap in milliseconds between the highlighting of the entity and the Quad display.
- **Quad hide delay** : Sets the time lap in milliseconds to hide the Quad when the cursor is outside the **Quad hide margin** limit.
- **Quad hide margin** : Sets the distance in pixels the cursor can move away from the Quad. If the cursor moves outside this limit, the Quad is hidden.
- **Quad go transparent** : Defines whether the Quad becomes transparent when the mouse moves outside the Quad, but within the **Quad hide margin** .

Figure 1.270


Quad
Quad display
0x000D (13)
negative
Switch off quad
1
Display the quad when the cursor hovers on an entity
2
Display the quad when entities are selected
4
Display the quad on right click
8
Suppress the quad on hover when entities are selected
Quad default command launch
[1] Launch default quad command on mouse right-click when quad is displayed
Quad popup corner
[1] Middle right
Quad width
6
Quad most recent items
2
Quad icon size
[32] Large icons
Quad icon space
[1] Normal
Quad show delay
150
Quad hide delay
350
Quad hide margin
50
Quad go transparent
Go transparent
Quad tooltip delay
800
Quad expand delay
110
Quad expand tab delay
50
Quad tab flags
0x000C (12)
Rollover tips
[1] Show properties on hover
Rollover opacity
100
Rollover selecion set
[2] Properties shared by all selected entities

- **Quad tooltip delay** : Sets the delay time in milliseconds to display tooltips. If the value is negative, no tooltips display.
- **Quad expand delay** : Sets the time lap in milliseconds to expand the Quad after mouse-enter.
- **Quad expand tab delay** : Sets the time lap in milliseconds to expand a group tab after mouse-enter.
- **Rollover tips** : Specifies whether entity properties display in the Quad while hovering (1.12.5).
- **Rollover opacity** : Specifies the degree of opacity when the Quad is in rollover state. Values between 10 (very transparent) and 100 (fully opaque) are accepted.
- **Rollover selection set** : Controls the content of Rollover tips when multiple entities are selected.

2. To edit a setting, select the setting then do one of the following

   - Type a new value.
   - Click the check box.
   - Choose an option.

3. Close the **Settings** dialog.

The Quad layout

The layout of the Quad cursor menu depends on:

- The entity type under de cursor or the current selection set.
- The current Workspace (6.33.15).
- Geometric events such as intersection or tangency.
- When the cursor hovers over an entity, the entity highlights and the Quad displays the command most recently used with this entity type. When multiple entities are under the cursor, hit the TAB key to highlight all entities one by one. 

If QuadCommandLaunch = 1, right click without entering the Quad to launch the command. If QuadCommandLAUNCH = 0, click the command button.
- If Rollover Tips (2.1.2) = ON the properties of the entity under the cursor display in the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadRollOverTips.png}) If the Quad display is off, rollover tips display if:
  - SELECTIONPREVIEW (1.13.11) = 1 or 3
  - ROLLOVERTIPS = ON (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/Qua

- Click a property to edit: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadRo

Which properties display for each entity type is specified in the Properties tab (6.9.58) of the Customize dialog box. Select an entity type in the left pane, then check properties in the **Rollover** pane. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/CustomizeRolloverTipsEntities.png})

- Move the cursor to the Quad to display the most recently used commands with this entity type. A number of command category tabs display at the bottom. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/B 1.png})

  - The number of buttons is controlled by the **QuadWidth** user preference.

Figure 1.271

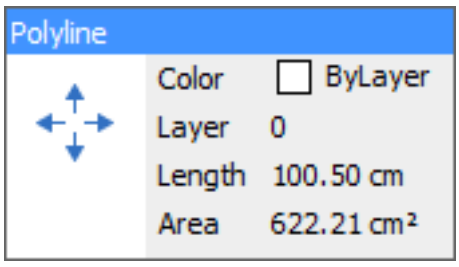


Figure 1.272

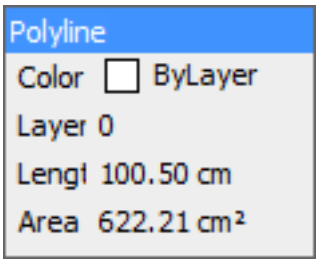
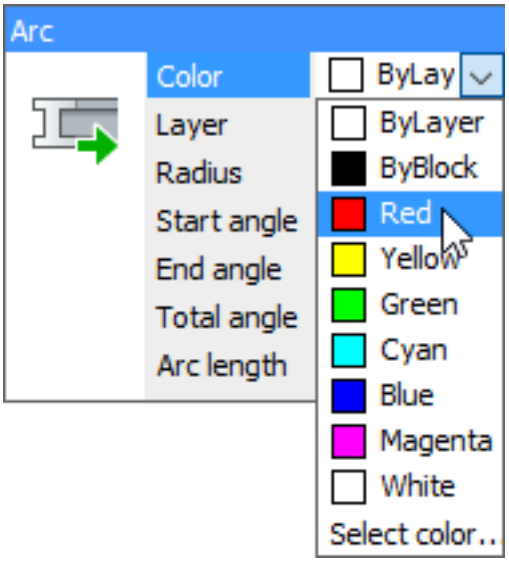


Figure 1.273



  - Which command group tabs display depends on the current Workspace (6.33.15). See To edit the workspace command groups (1.12.5) to switch closed command groups on.
  - Click an icon to launch the command.

- Move the cursor over the tabs to expand the command group under the cursor. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadExpanded-2.png}) Click an icon to launch the command.

Note: When the Quad is expanded, click the title bar of the Quad to display the Rollover tips.

Figure 1.274

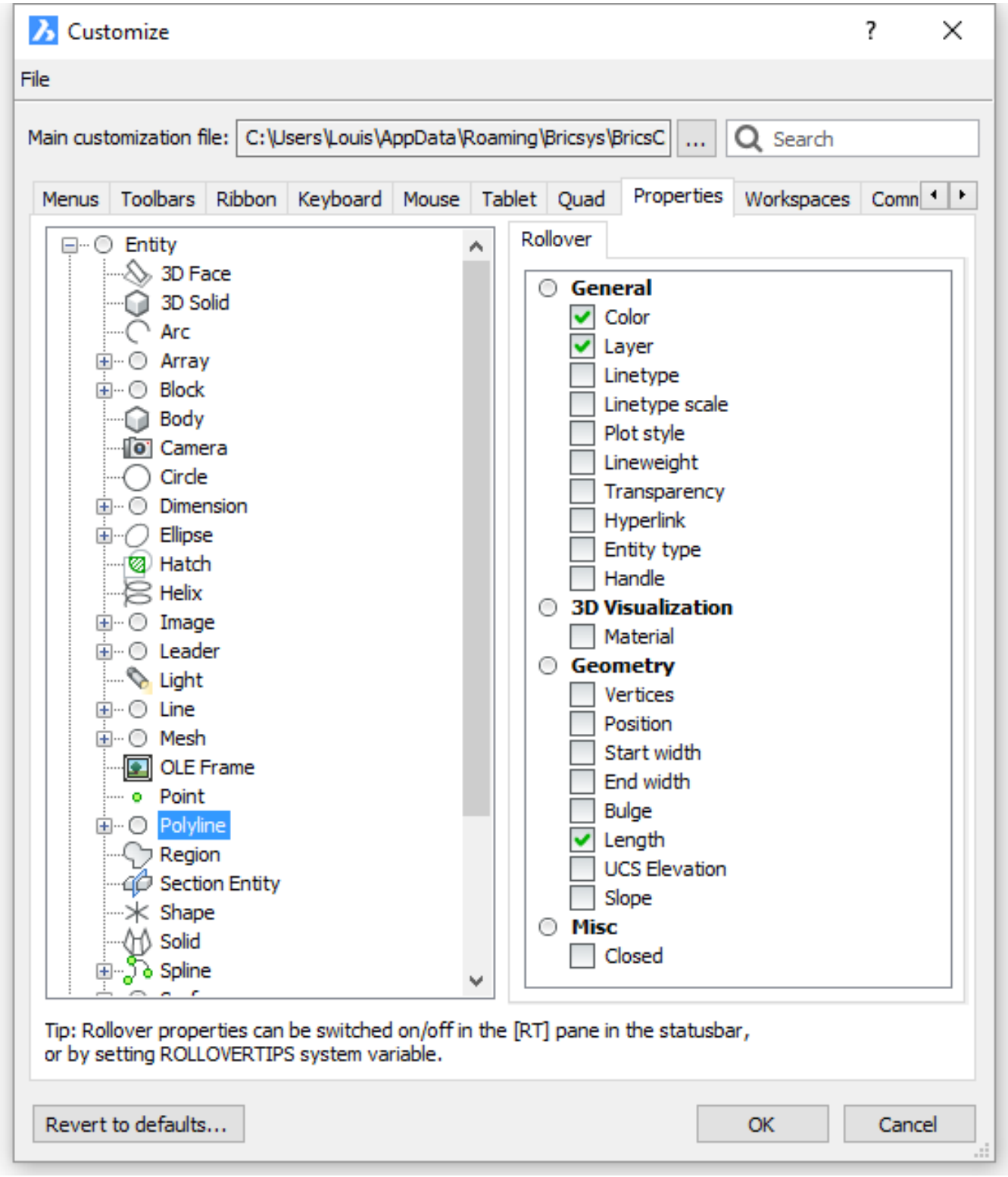


Figure 1.275

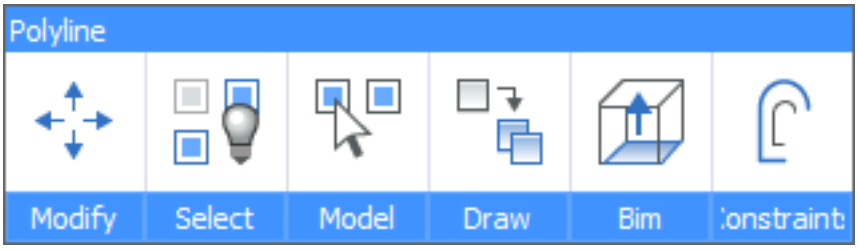
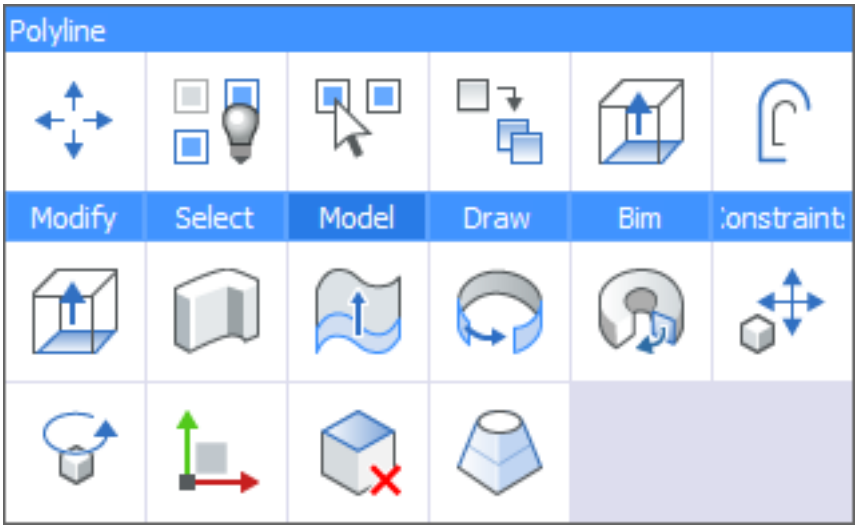


Figure 1.276



Using the Quad

1. Move the cursor to the entity you want to manipulate. The entity under the cursor highlights and the Quad displays the most recently used command with this entity type.
2. (option) Select more entities. See Selecting multiple entities.
3. Do one of the following:
   - Depending on the value of the QUADCOMMANDLAUNCH (1.13.11) system variable, either right click or click the Quad to repeat the most recently used command.
   - Move the cursor to the Quad and select a different command. See The Quad Layout.
4. Execute the command.

Tutorial: Experience the Quad Cursor in BricsCAD Click here (1.2.5) to watch

Selecting multiple entities

1. Move the cursor over the first entity, then click when the entity highlights. The entity is selected and remains highlighted.
2. Move the cursor to another entity, then click when the entity highlights. The entity is added to the selection set and remains highlighted.
3. (option) Repeat step 2 to select more entities.
4. (option) Move the cursor to a selected entity, then press and hold the Shift key and click. The entity is removed from the selection set.

NOTES

- When the Quad is active, SELECTIONPREVIEW (1.23.5) is neglected, unless a command prompts for entity selection.
- When working with solids:
  - The SELECTIONMODES (1.23.5) system variable controls which subentities are detected: edges, faces and/or boundaries. Holding down the Ctrl key inverts the current selection mode.
  - Click a button on the **Selection Modes** toolbar to toggle an option. A pressed button indicates the option is selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/
    *  Enable detection of 3D solid edges
    *  Enable detection of 3D solid faces
    *  Enable boundary detection
- Repeatedly press the TAB key to highlight all (sub)entities under the cursor one by one.

Selecting edges of faces or solids

Command: DmSelectEdges (6.12.25)

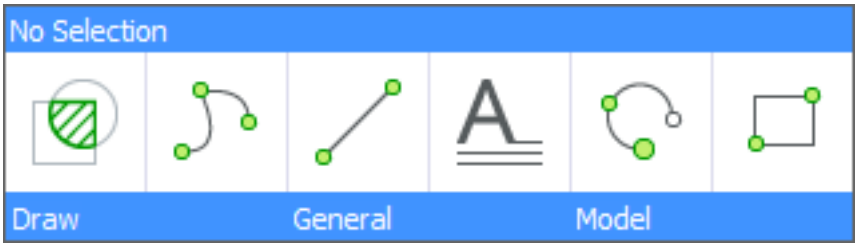
1. Select the faces and or solids. See Selecting multiple entities.
2. In the **Model** command group of the Quad cursor menu, click the **Select Edges** tool (). All edges of the selected faces and solids are selected.
3. (option) To remove edges from the selection set: press and hold the Shift key, then click the edge(s).

Using the Quad to create entities

If the flag of the QUADDISPLAY (1.13.11) system variable is set, entity creation commands are available in the Quad. To set the **Display the Quad on right click** option: Right click the Quad (1.12.2) field in the **Status Bar** and check the option in the context menu. The available entity creation commands in the Quad:

- The most recently used commands. The number of available commands depends on the **QuadWidth** user preference.
- A number of command group tabs, depending on the current workspace (6.33.15). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadNoSelection.png})

Figure 1.277



NOTE	If the cursor is on an entity, hold down the right mouse button longer to display the **No Selection** Quad.

Customizing the Quad

To open the **Customize/Quad** dialog box, do one of the following:

- Right click the Quad (1.12.2) field in the Status Bar (1.12.2) and choose **Customize Quad items...** in the context menu.
- Run the Customize (6.9.58) command, then select the **Quad** tab.

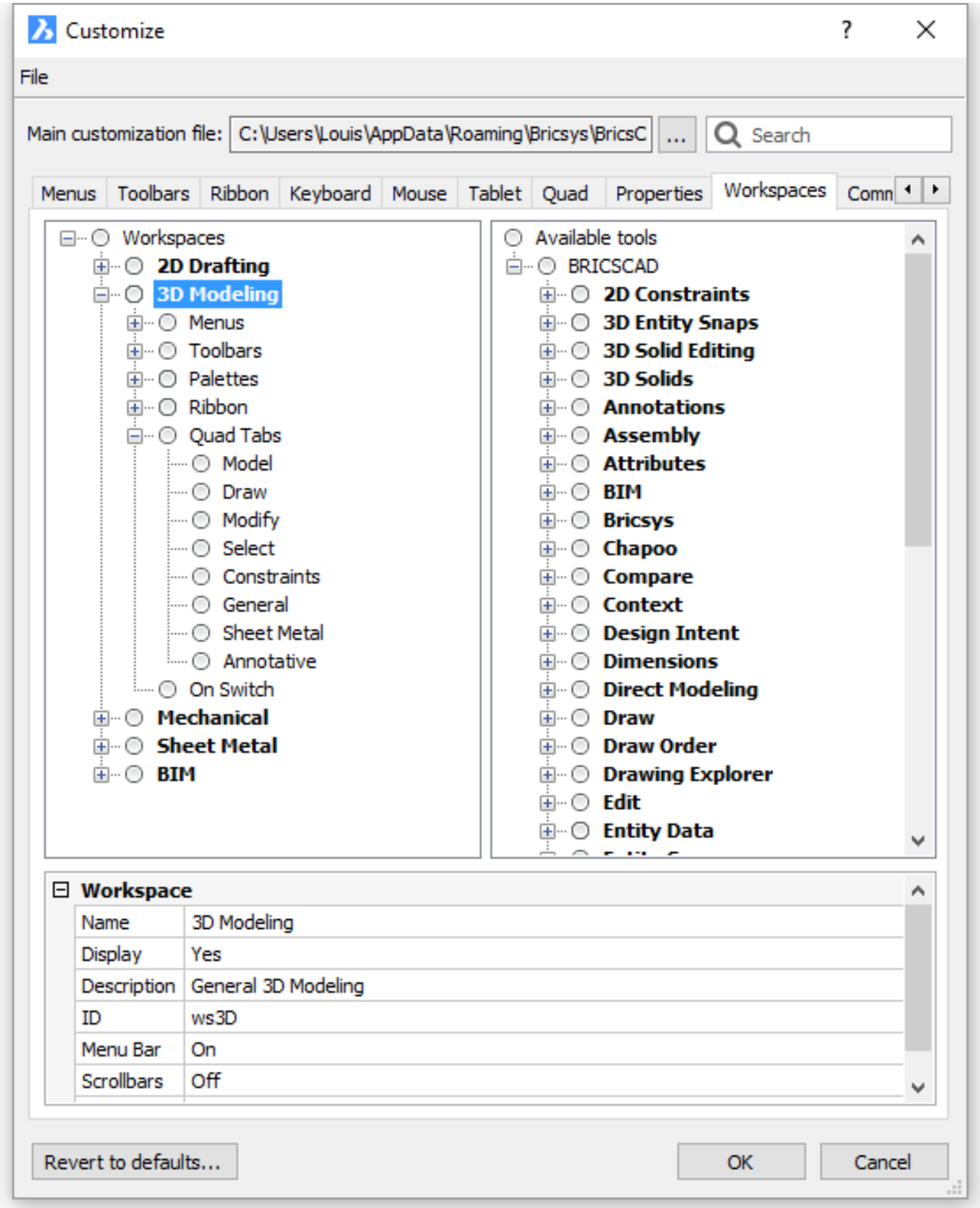
The **Customize/Quad** dialog box has two section tabs that are involved in customizing the Quad:

- The **Quad** section: In this section, the name and content of all Quad Tabs can be specified, similar to the way Ribbon Tabs are customized. Quad Tabs can be created, copied, renamed and deleted. Manage the content of a Quad Tab by dragging buttons from the right side of the dialog, or by using the right-click context menu to add, rename, move or delete Quad Buttons. The right side of the dialog contains the full list of all buttons which can be shown in the quad. These include the tools defined in the cui file, plus all internally defined Quad Buttons, which are listed under the new 'Built-in' branch at the bottom. These built-in buttons are similar to the tools, only they can not be edited: their name, icon, command string etc.. are read-only. The condition which specifies whether a certain Quad Button will appear in the quad when the cursor hovers on an entity, is specified for each Quad Button individually in the 'Object Filter' property, similar to the way Object Context Menu Items are configured. It is not possible to modify the Object Filter of a built-in Quad Button.
- The **Workspaces** section: This section defines which Quad Tabs will be available in the Quad, in which order, and in which workspace. In this interface, you can add/move/remove entire Quad Tabs. The contents of the Quad Tabs is not shown nor editable in this section, as this is done in the Quad section of the dialog. When a new Quad Tab is created in the Quad section, this Quad Tab is automatically added to the Quad in all workspaces.

To edit the Quad tabs of a workspace

1. Move the cursor over a toolbar, then right click and choose **Customize** in the context menu.
2. Click the **Workspace** tab on the **Customize** dialog box.
3. Expand the workspace you want to edit.
4. Expand the **Quad Tabs** list. Tabs that are currently switched off display in grey. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/QuadGroupOrderDialog.png})
5. To set the availability of a tab, select the tab then click the **Display** field in the Properties grid and choose **Yes** or **No**.
6. To add a tab do one of the following:
   - Right click the Quad Tabs node and choose Appand Quad tab in the context menu to add a tab at the bottom of the tab list.
   - Select a tab, then right click and choose Insert in the context menu to add a tab above the selected tab.
   - On the **Select Ribbon Tab** dialog select the tab you want to insert and click the **OK** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/SelectRibbonTabDialog.png})

Figure 1.278


Customize
File
Main customization file:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsC
...
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Properties
Workspaces
Comn
Workspaces
2D Drafting
3D Modeling
Menus
Toolbars
Palettes
Ribbon
Quad Tabs
Model
Draw
Modify
Select
Constraints
General
Sheet Metal
Annotative
On Switch
Mechanical
Sheet Metal
BIM
Available tools
BRICSCAD
2D Constraints
3D Entity Snaps
3D Solid Editing
3D Solids
Annotations
Assembly
Attributes
BIM
Bricsys
Chapoo
Compare
Context
Design Intent
Dimensions
Direct Modeling
Draw
Draw Order
Drawing Explorer
Edit
Entity Data
Workspace
Name
3D Modeling
Display
Yes
Description
General 3D Modeling
ID
ws3D
Menu Bar
On
Scrollbars
Off
Revert to defaults...
OK
Cancel

7. To remove a tab, select the tab, then right and choose **Delete** in the context menu.
8. To change the tab order: drag and drop the tabs in the list.

Figure 1.279

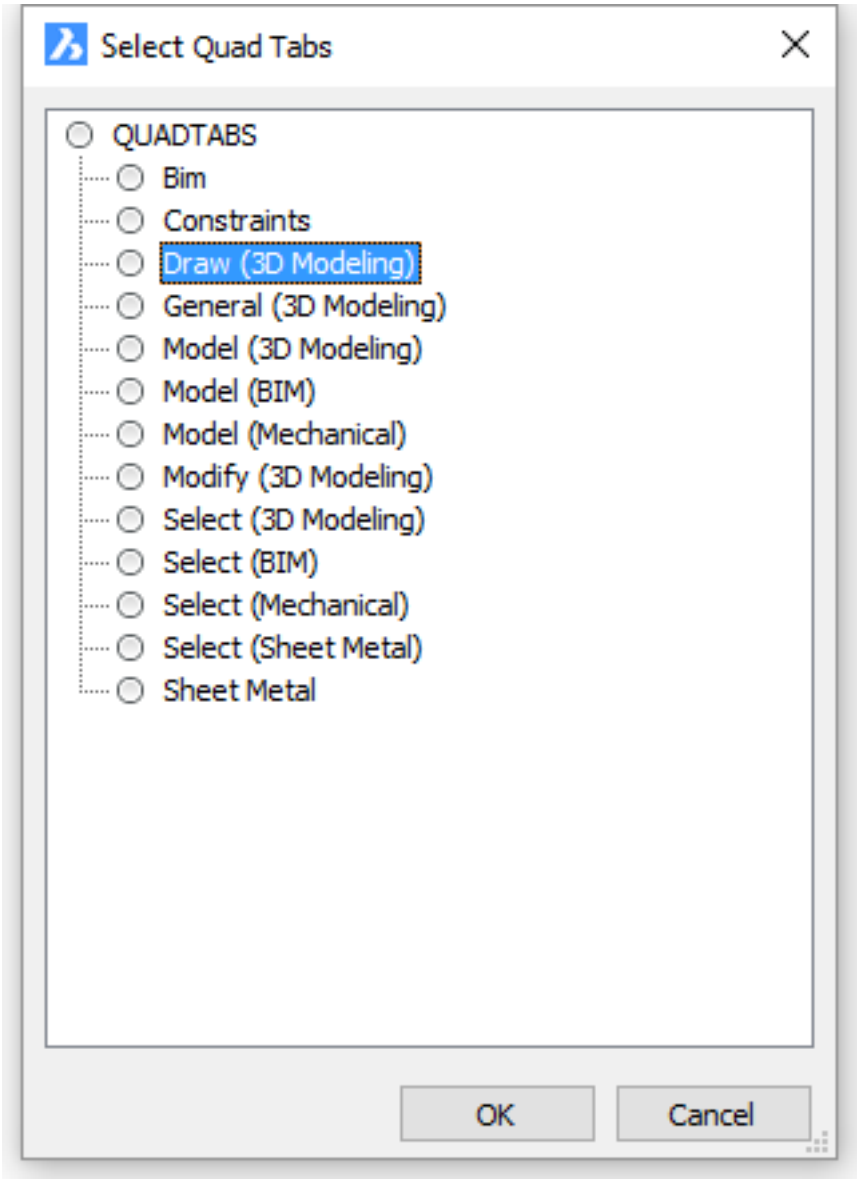


Figure 1.280



Figure 1.281



Figure 1.282

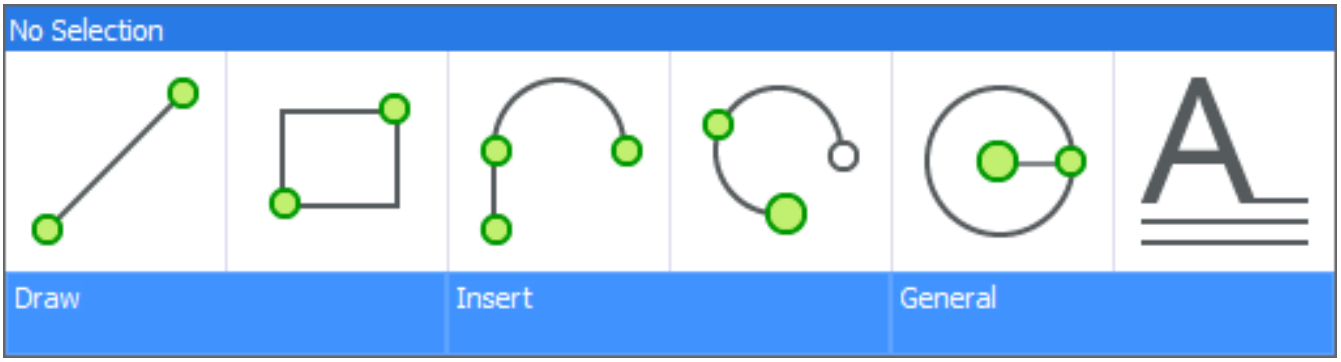


Figure 1.283



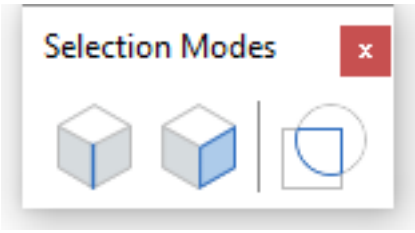
Figure 1.284



Figure 1.285



Figure 1.286



1.12.6 Prompt Menus

BricsCAD commands often provide several options. These options appear in the status bar or command bar and also in a prompt menu, which initially displays in the top right corner of the screen. Choose the appropriate option by clicking in the prompt menu. E.g. click **Radius** in the **PLINE** prompt menu to define the radius of the current arc segment of a polyline.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Ink-2448/images/staticbricsyscom/help/enUS/	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Ink-2448/images/staticbricsyscom/help/enUS/
Polyline prompt menu when drawing line segments	Polyline prompt menu when drawing arc segments

To control the display of prompt menus

Whether a prompt menu displays and its location is controlled through the **PROMPTMENU** system variable and the **PromptMenuFlags** user preference. Type **promptmenu** in the command bar, then type a value of 0 to 5 (see table below) and press Enter.

Value	Description
0	No prompt menus
1	Display prompt menu at user defined location (*)
2	Display prompt menu at top left corner
3	Display prompt menu at top right corner
4	Display prompt menu at bottom left corner
5	Display prompt menu at bottom right corner

(*) You can move the prompt menu by dragging it; future prompt menus will appear where you last placed it.

NOTES	The PROMPTMENU (1.13.11) system variable is saved in the registry and therefore applies to all drawings. The PromptMenuFlags (1.13.11) user preference has three options: • 1 = Show hidden options. Hidden options display in italics. • 2 = Hide the prompt menu when a command prompts to select entities. • 3 = Disable selection options in the prompt menu.

Tutorial: The Prompt Menu in BricsCAD and Its Many Uses

Click here (1.3.20) to watch

Tutorial: Customizing the Command Prompt in BricsCAD

Click here (1.3.9) to watch

Figure 1.287

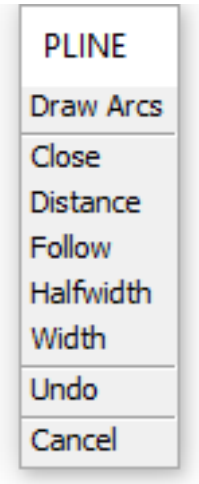


Figure 1.288

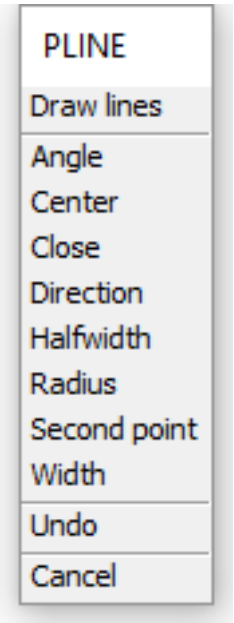
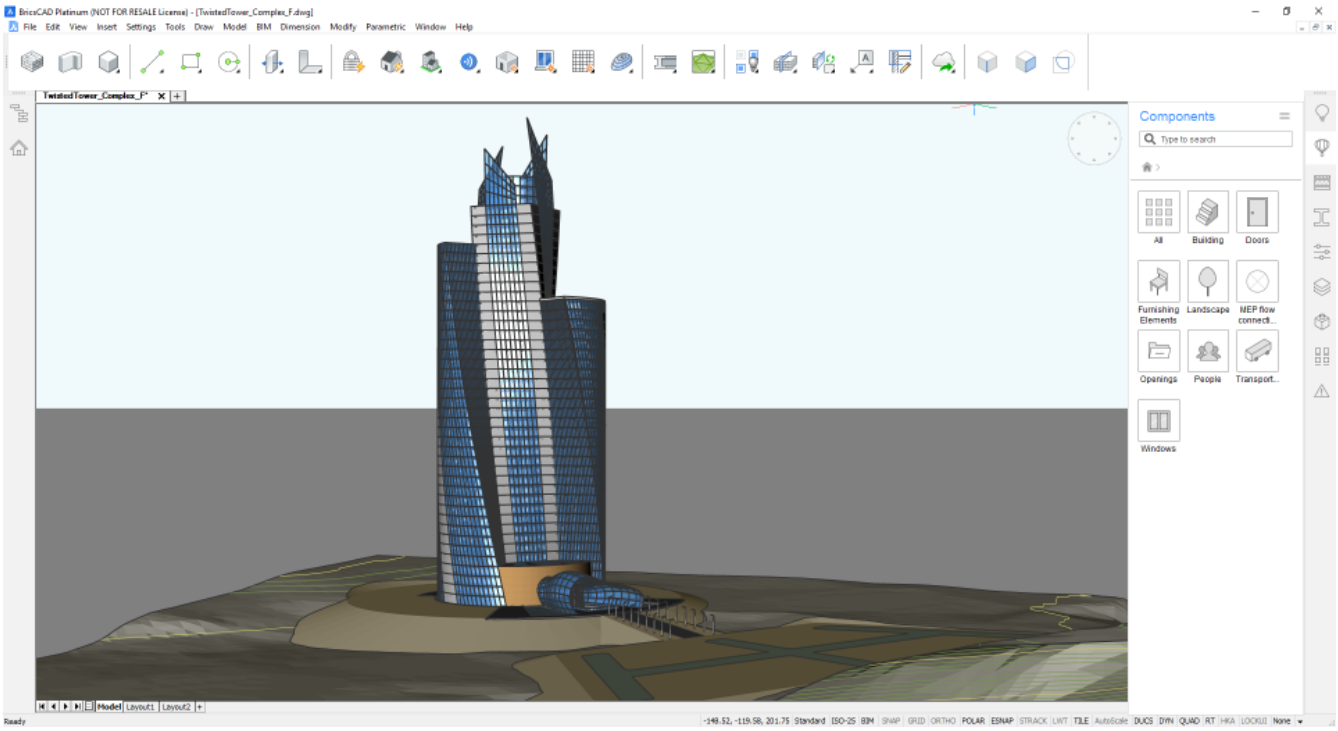


Figure 1.289



1.12.7 Dockable Panels

The following panels can be docked: Attachments (6.4.24),Command Bar (6.9.34), BIM Compositions (6.7.3), Components (6.9.61),Content Browser (6.9.41), Layers Panel (6.20.6), Mechanical Browser (3.2.2), Parameters (6.24.8),Properties Bar (6.24.49), Render Materials (6.21.6), Report (6.8.32),Ribbon (6.26.31), Sheet Sets (6.27.31), Structure (1.23.8), Tips (6.29.16),Tool Palettes, (6.29.19)

NOTE	Use the -TOOLPANEL (6.29.21) command to show, hide or toggle the display of the various tool panels at the command line.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/DockablePanelsScreen.png}) Panels are docked at the left or right side of the screen as defined in the Panels property of a workspace in the Customize | (1.13.3) Workspaces (1.13.10) dialog: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/

- Panels | Stack (LDOCK) define the panels stacked a the left side of the screen.
- Panels | Stack (RDOCK) define the panels stacked a the right side of the screen.

The stack type is defined in the Customize | (1.13.3) Workspaces (1.13.10) dialog through the Stack Type property of a workspace. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/WorkspaceStackType. The possible stack types are:

- Fixed resizable panel set
- Collapsible panel set
- Flyout panel set

Figure 1.290

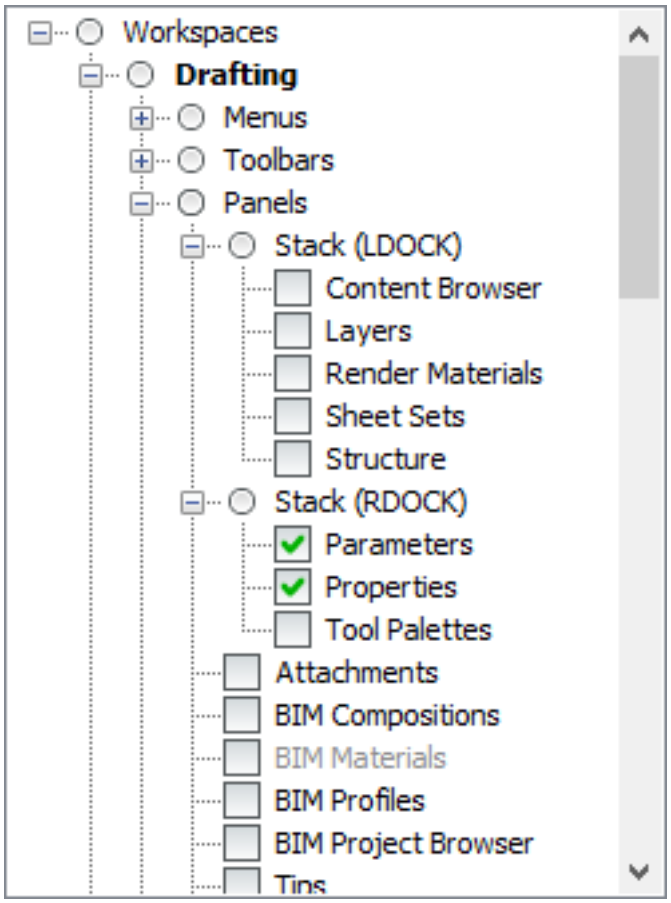
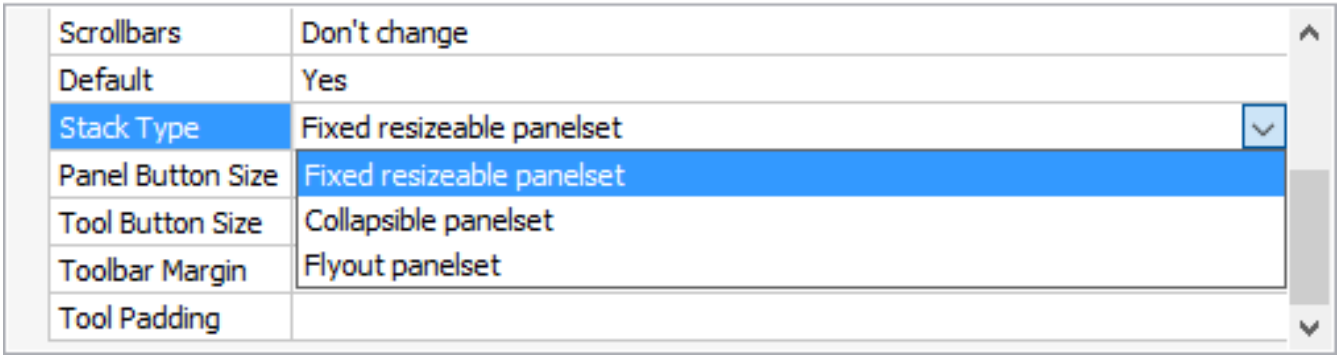


Figure 1.291

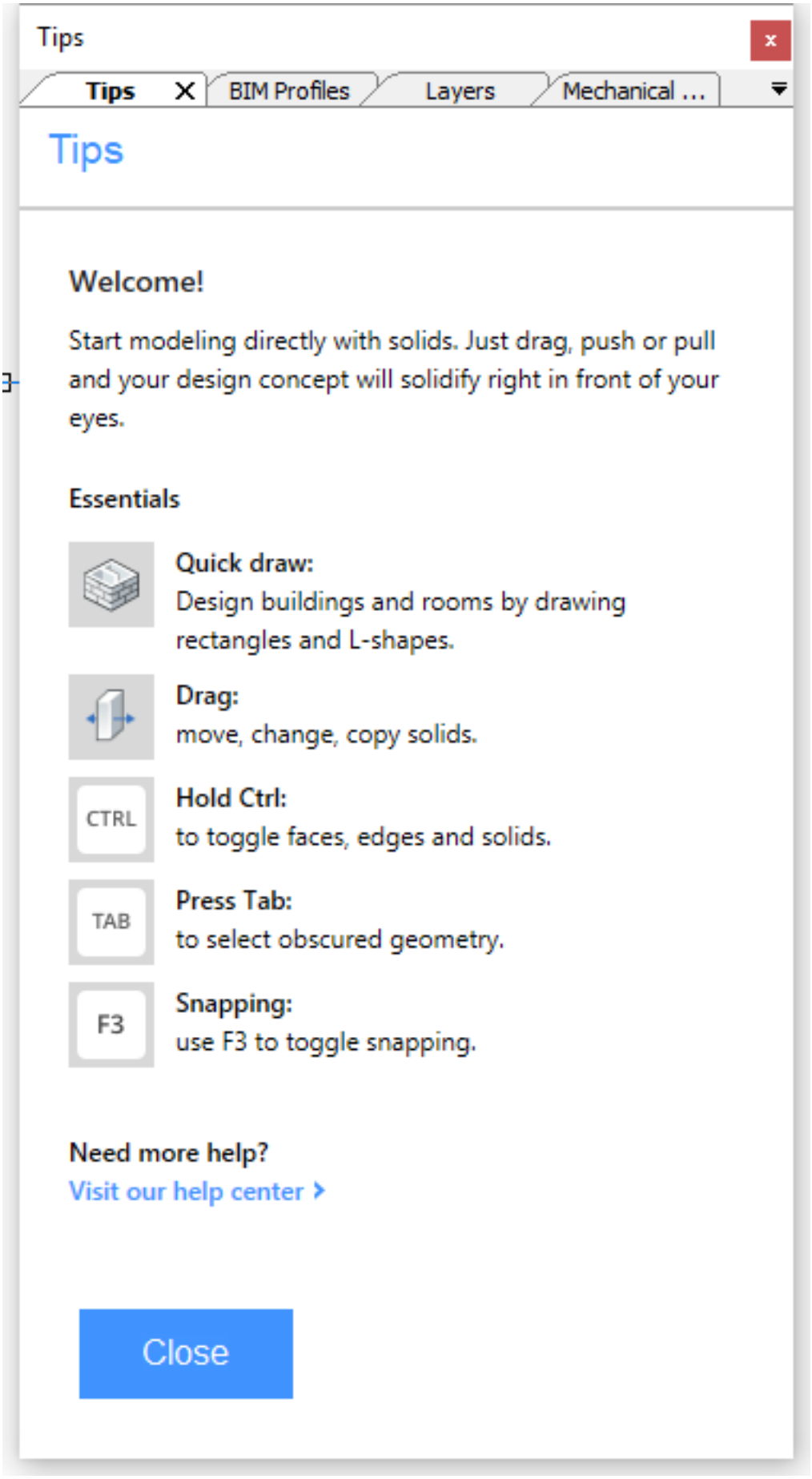


Fixed resizable panelset

Panels are docked next to each other or in a tabbed panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V

- Click a tab to display the panel.
- Press and hold the left mouse button to drag a tab to re-arrange the tab order.
- Drag an tab into the graphic screen to make it floating.

Figure 1.292

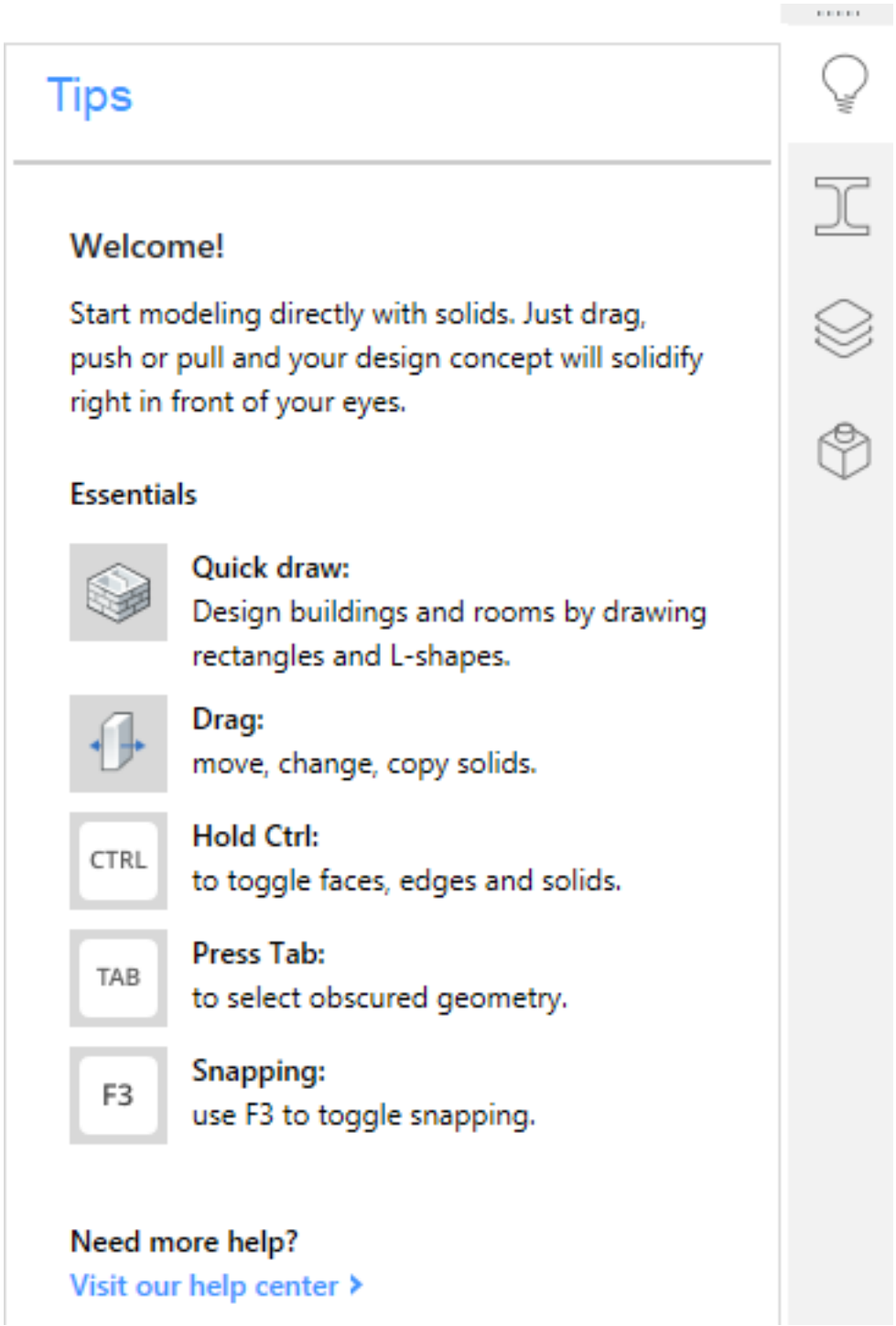

Tips
Tips
BIM Profiles
Layers
Mechanical ...
Tips
Welcome!
Start modeling directly with solids. Just drag, push or pull and your design concept will solidify right in front of your eyes.
Essentials
Quick draw:
Design buildings and rooms by drawing rectangles and L-shapes.
Drag:
move, change, copy solids.
CTRL
Hold Ctrl:
to toggle faces, edges and solids.
TAB
Press Tab:
to select obscured geometry.
F3
Snapping:
use F3 to toggle snapping.
Need more help?
Visit our help center
Close

Collapsible panelset andFlyout panelset

Panels are docked at a default location in an icon tab strip at the left or right side of the screen. Flyout panels overlap the drawing area, thus hiding part of the drawing. Collapsible panels narrow the drawing area when expanded. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/PanelSetFlyout.png})

- Click an icon to expand a panel. Only one panel in the same icon strip can be expanded.
- To collapse an expanded icon click its icon again.
- Press and hold the left mouse button to drag an icon to re-arrange the icon order.
- Drag the icon into the graphic screen or double click the icon to make the panel floating.

Figure 1.293

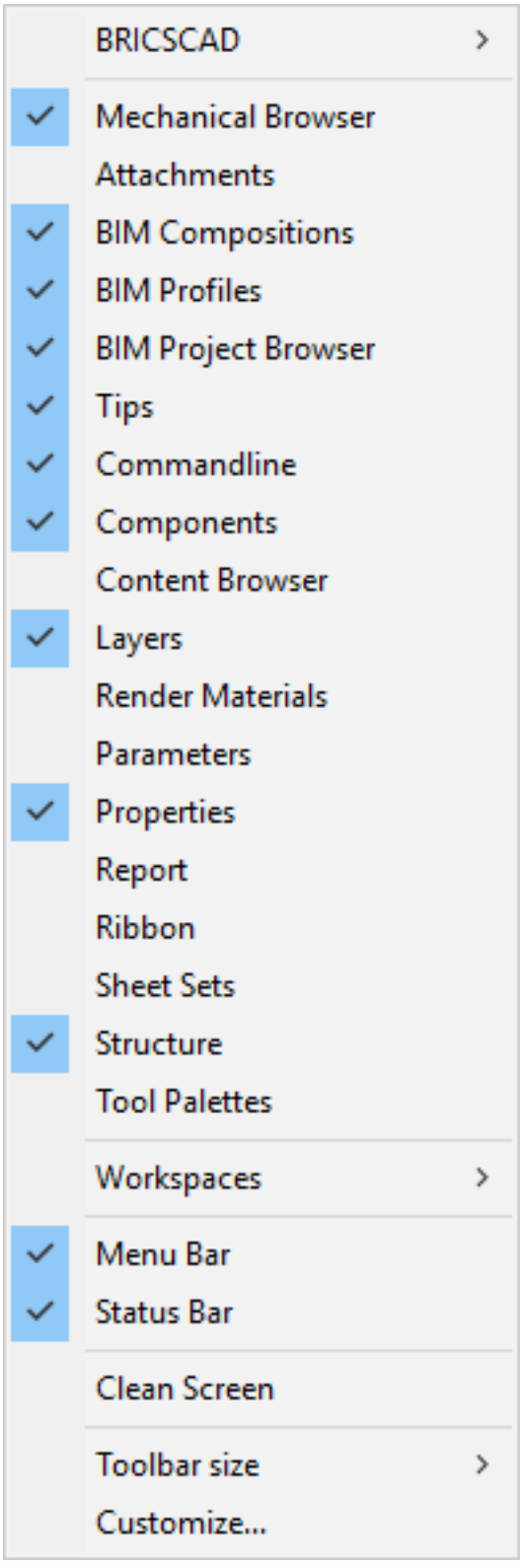

Tips
Welcome!
Start modeling directly with solids. Just drag, push or pull and your design concept will solidify right in front of your eyes.
Essentials
Quick draw:
Design buildings and rooms by drawing rectangles and L-shapes.
Drag:
move, change, copy solids.
CTRL
Hold Ctrl:
to toggle faces, edges and solids.
TAB
Press Tab:
to select obscured geometry.
F3
Snapping:
use F3 to toggle snapping.
Need more help?
Visit our help center ›

To open or close a panel

1. Move the cursor over a toolbar or ribbon panel, then right click. A context menu displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/DockablePanelsList.png}) Panels that are currently open are checked.
2. Click a panel in the menu to toggle its display.

Icon	Panel
	Components
	BIM Compositions
	Context Browser
	Layers
	Mechanical Browser
	BIM Profiles
	Project Browser
	Properties
	Render Materials
	Sheet Sets
	Structure
	Tips
	Tool Palettes

Figure 1.294


BRICSCAD
Mechanical Browser
Attachments
BIM Compositions
BIM Profiles
BIM Project Browser
Tips
Commandline
Components
Content Browser
Layers
Render Materials
Parameters
Properties
Report
Ribbon
Sheet Sets
Structure
Tool Palettes
Workspaces
Menu Bar
Status Bar
Clean Screen
Toolbar size
Customize...

To dock a floating panel

Although panels can be docked at either side of the graphical screen, they are best placed at the left or right side. Do one of the following:

- Drag the panel by its title bar. A rectangle indicates the boundary of the panel. Release the left mouse button when the boundary rectangle is over an edge of the graphical screen.
- Double click the title bar to dock the panel at its previous location.

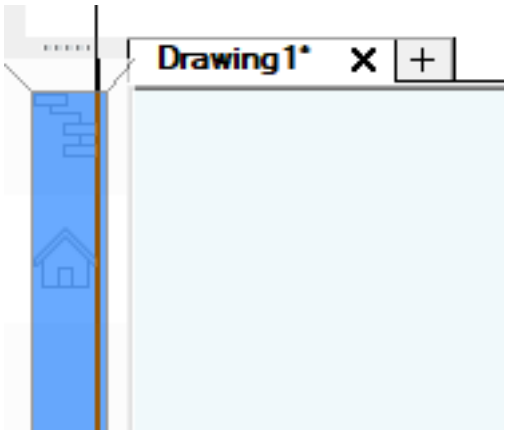
NOTE	The LOCKUI (1.13.11) system variable controls whether the current position docked and/or floating panels is locked or not. Press and hold the Ctrl-key to override. Click the **LOCKUI** field in the Status Bar (1.12.2) to toggle the locking of the user interface.

To move a panel to the othericon tab strip

1. Drag the icon to the other icon strip. The docking location selector displays. The blue zone indicates the selected docking position. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/DockPanel.p

2. Move the cursor over the center docking location then release the left mouse button.

Figure 1.295



1.12.8 Digitizing Tablet

Command: TABLET (6.29.7) **Overview**: The Tablet command configures (1.12.8) and calibrates (1.12.8) a digitizing tablet and toggles tablet mode (1.12.8). This command requires that Wintab32.dll from Wacom Technology, Corp. be installed on the computer. Download Wintab32.dll from the internet, then place the file in your system directory: C:\Windows\System32. A digitizing tablet, also called a graphics tablet, agraphics pad or drawing tablet, is a computer input device that allows you to hand-draw images and graphics, similar to the way one draws images with a pencil and paper. In a CAD program a digitizing tablet can be used to trace paper drawings into a drawing file or to launch drawing commands from a digitizing tablet overlay.

To use a digitizing tablet in BricsCAD

In order to prepare BricsCAD for using a digitizing tablet do the following:

- Install the appropriate driver for your tablet.
- Make sure Wintab32.dll exists in C:\Windows\System32.
- Download the tablet overlay and CUI files.
- Print the BricsCAD tablet overlay.
- Load the overlay CUI file.
- Start BricsCAD, then initialize the tablet.
- Configure the tablet: part of the tablet surface is used as the menu area, another part as the screen pointing area.
- Calibrate the tablet: a calibrated tablet can be used to trace the geometry of a paper drawing or image into a drawing.

To download the tablet overlay and the CUI files

Click here to download the file Tablet.zip. The ZIP file contains the following files:

- **overlay.png**: an image of the tablet overlay
- **overlay(A3).pdf**: to print the tablet overlay on a A3 paper size.
- **overlay(cm).dwg**: a DWG in which the **overlay.png** is attached as an image. The drawing contains two layouts to print the overlay using decimal units.
- **overlay(inch).dwg**: a DWG in which the **overlay.png** is attached as an image. The drawing contains two layouts to print the overlay using imperial units.
- **tablet.cui**: a partial cui file to be used with a tablet.
- **tablet(acadLike).cui**: an AutoCAD-like partial cui file to be used with a tablet.

To load the overlay CUI file

1. Extract the overlay CUI files to the **Support** folder of the **roamable root** folder (see BricsCAD User Files (1.13.2)). E.g. C:\Users\[user name]\AppData\Roaming\Bricsys\BricsCAD\V[version nummer]\en_US\Support\.
2. Load **Tablet.cui** as a partial CUI (see To load a partial CUI file (1.13.4)). The next time you start BricsCAD, **Tablet.cui** will be loaded automatically.

NOTES

- Please make sure the **overlay.png** image file is extracted to the same folder as the DWG's.
- It might be necessary to repeat the above procedure after upgrading BricsCAD.

To switch the tablet ON / OFF

Do one of the following:

- Click the **TABLET** field in the Status Bar (1.12.2).
- Press the **F4** function key to toggle the tablet mode.
- Choose **Tablet On** or **Tablet Off** in the **Settings** menu.
- Type **tablet** in the command bar, press Enter, then type **ON** or **OFF** ; or click **ON** or **OFF** in the prompt box.

Figure 1.296

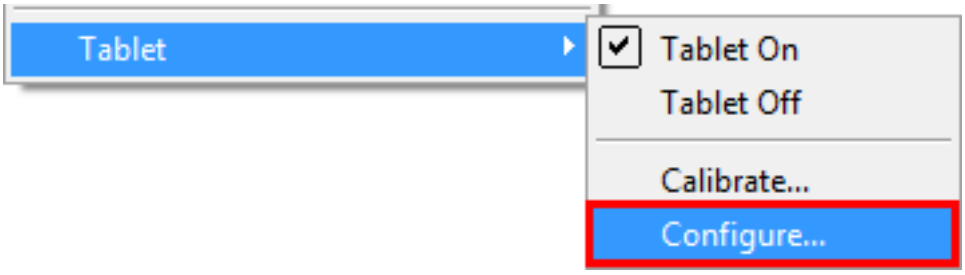
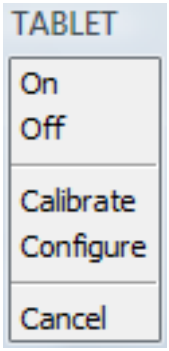


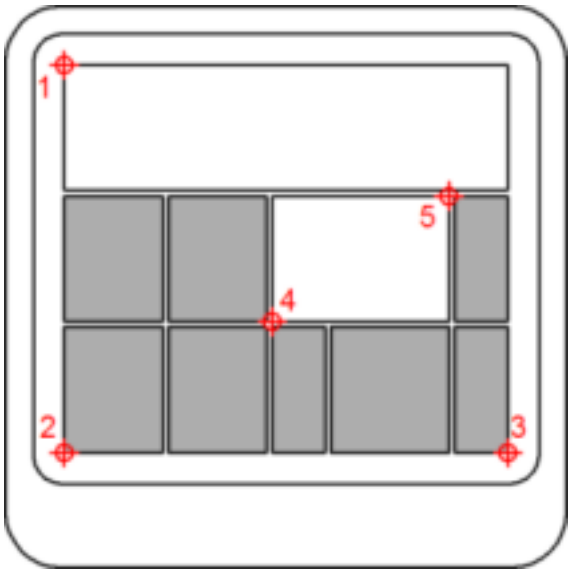
Figure 1.297



To configure the tablet

1. Do one of the following:
   - Choose **Tablet** in the **Settings** menu, then choose **Configure...** in the flyout menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/SettingsTabletConfig.png})
   - Type tablet in the command bar and press Enter. You are prompted: Tablet: ON/OFF/CALibrate/ConFiGure/<On>: a prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/T
     Type **cfg** in the command bar and press Enter or choose **Configure** in the prompt menu.
2. You are prompted: Do you want to align the tablet overlay? Yes/No/<No>. Type **Y** in the command bar and press Enter or choose **Yes** in the prompt menu. You are prompted: Digitize upper left corner of the overlay.
3. Click on the mark in the upper left corner (1) of the digitizing tablet overlay. (\vref {fig:C:/Dev/Task-2448/images/staticbric
   You are prompted: Digitize the lower left corner of the overlay.
4. Click on the mark in the lower left corner (2) of the digitizing tablet overlay. You are prompted: Digitize the lower right corner of the overlay.
5. Click on the mark in the lower right corner (3) of the digitizing tablet overlay. You are prompted: Digitize the lower left corner of the screen pointing area.
6. Click on the mark at position (4) on the of the digitizing tablet overlay. You are prompted: Digitize the upper right corner of the screen pointing area.
7. Click on the mark at position (5) on the of the digitizing tablet overlay. You are prompted: Tablet configured.

Figure 1.298



To calibrate the tablet

To calibrate the digitizing tablet you must specify at least two points. The more additional points you specify, the more accurate the digitizing process. Additional points are particularly recommended when digitizing a non-orthogonal image, such as an aerial photograph.

1. Do one of the following:
   - Choose **Tablet** in the **Settings** menu, then choose **Calibrate...** in the flyout menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/SettingsTabletCalibrate.png})
   - Type tablet in the command bar and press Enter. You are prompted: Tablet: ON/OFF/CALibrate/ConFiGure/<On>: a prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/T
     Type **cal** in the command bar and press Enter or choose **Calibrate** in the prompt menu.
2. You are prompted: Digitize point #1: Click a point on the digitizing tablet. You are prompted: Enter coordinates for point #1:
3. Do one of the following:
   - Click a point in the BricsCAD drawing window.
   - Enter the coordinates (x,y) in the command bar.
4. You are prompted: Digitize point #2: Repeat step 3 to specify the following calibration point. You are prompted: Digitize point #3 (or ENTER to end):
5. Do one of the following:
   - Repeat step 4 to specify an additional calibration point. (You can specify up to 10 calibration points).
   - Press Enter to conclude the calibration process. You are prompted: When 2 calibration points are specified: Tablet calibrated. When 3 or more than 4 calibration points are specified: Select transformation type Orthogonal/Affine: Wen 4 calibration points are specified: Select transformation type Orthogonal/Affine/Projective: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/Tab

Figure 1.299

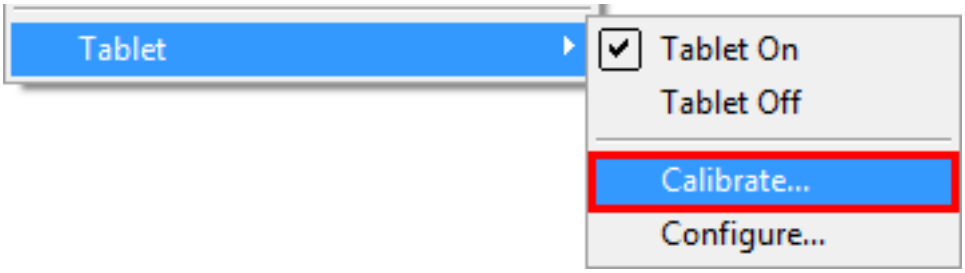


Figure 1.300

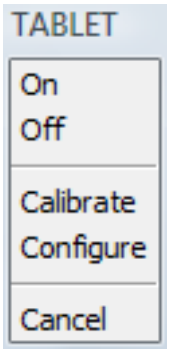


Figure 1.301

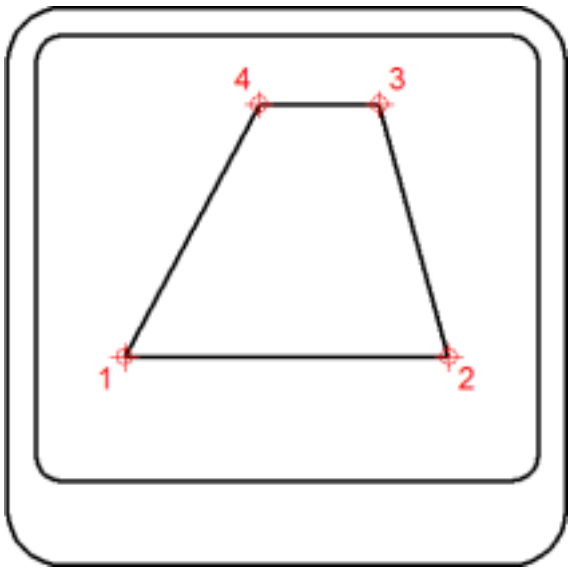


Figure 1.302



6. Select the transformation type. Transformation refers to the calculation of the points on the screen that correspond to points you digitize on the tablet. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/

   - **Orthogonal** (left): To be used with accurate paper drawings. Orthogonal transformation maintains all angles and preserves relative distances. If only two calibration points are specified orthogonal transformation is generated automatically.
   - **Affine** (middle): Maintains parallel lines, but not necessarily the angles between intersecting lines.
   - **Projective** (right): Does not maintain parallel lines, nor angles.

1.13 Customization

1.13.1 BricsCAD Startup Options

When starting up BricsCAD it is possible to add arguments or switches. E.g. the command line "BricsCAD.exe drawing1.dwg drawing2.dwg drawing3.dwg" will start the BricsCAD application and load the specified drawings while the command line "bricscad.exe /P MyProfile" will load BricsCAD using user profile MyProfile. "/P MyProfile" is called a command line switch. Both a slash or a hyphen are accepted to precede the switch, e.g. "-P MyProfile" will also work. The space is optional, e.g. "/PMyProfile" would also work, and the switch character is case insensitive, e.g. "-p MyProfile" is also fine.

Switch	Description
/P [userProfileName]	Starting without the /P option, BricsCAD re-uses the last used user profile (called current user profile), which is not necessarily the Default profile. If the user profile specified with the /P option doesn't exist at startup, a new empty user profile will be created with that name. If the /P argument is an .arg file the arg file is used to initialize the profile settings on the first run (See Starting BricsCAD with a specific profile (1.13.9)).
/B [scriptFileName.scr]	Runs a script (*.scr) or lisp (*.lsp) at startup. Opens all the file types that can be dragged and dropped on the application main frame window. If you pass filenames without a parameter, they are opened as if they had been passed with /B. Therefore, bricscad.exe mylisp.lsp is exactly the same as bricscad.exe /B mylisp.lsp, and bricscad.exe my.dwg is identical to bricscad.exe /B my.dwg.
/L	Suppresses the startup splash screen.
/LD [application.arx/brx/drx/lsp]	Loads an application on startup andon every new or opened drawingdepending on the file extension. If the file extension is .lsp/.des/.mnl, it is treated as a lisp file and loaded in every drawing. Other file types are loaded only once at startup.
/S [path]	Redefines the SRCHPATH system variable. e.g. BricsCAD.exe /S C:\BricsCAD\
/T [TemplateFileName]	Creates a new drawing based on the template drawing specified.
regserver unregserver	To register/unregister BricsCAD COM. Please note these arguments are to be used without a preceding "/".

NOTES

- Switch arguments containing spaces should be enclosed by double quotes, e.g. /P "My user profile"
- Switches can be combined, e.g. **/P MyProfile /L /S C:\BricsCAD** will start BricsCAD using the **MyProfile** user profile without the splash screen and sets the SRCHPATH system variable to **C:\BricsCAD** .
- Switches /B and /LD enable all sorts of files to be loaded, some maybe not expected. For example, .cui, .dsd, .txt, .pdf types are also processed.

Besides these switches, BricsCAD supports file names as command line arguments. Supported file types are:

- DWG - to load a drawing
- DXF - to load a drawing
- SCR - to load a script

- LSP - to load a LISP program
- DLL - to load an ADS/SDS or DRX/ARX program
- SLD - to load a slide file
- MNU - to load a menu file
- CUI - to load a menu file

If GETSTARTED = ON when you launch BricsCAD, the Get Started (1.10.6) dialog displays, which lets you choose the workspace and create a new drawing or open an existing drawing.

1.13.2 BricsCAD user files

Commands: SupportFolder (6.27.58), TemplateFolder (6.29.11) In BricsCAD the preferences of each user are stored in specific folders, which are called the **Local root** folder and the **Roamable root** folder. The **Local root** folder (Windows only) contains the **Templates** folder, where the drawing templates are saved. The **Roamable root** folder contains several subfolders:

- PlotConfig: where the plotter configuration files (*.pc3) are saved
- PlotStyles: where the plot style table files (*.stb) and plot color table files (*.ctb) are saved
- Support: where the hatch pattern files (*.pat), line style files (*.lin), alias files (*.pgp), unit files (*.unt), custom user interface files (*.cui), etc. are saved
- Templates: (Mac and Linux only) where the drawing templates are saved.

Platform Local / Roamable root folder Subfolders Windows C:\Users\[UserName]\AppData\Local \Bricsys\BricsCAD\V[Versio \Templates C:\Users\[UserName]\AppData\Roaming \Bricsys\BricsCAD\V[Version]\en_US \PlotConfig \PlotStyles \Support Mac /Users/[UserName]/Library/Preferences/Bricsys/BricsCAD/V[Version]/en_US /PlotConfig /PlotStyles /Support /Templates Linux /home/[UserName]/Bricsys/BricsCAD/V[Version]/en_US /PlotConfig /PlotStyles /Support /Templates

The BricsCAD user file manager

When you install an update of the software, the first time you launch BricsCAD after the update, the content of the **UserDataCache** folder (C:\Program Files\Bricsys\BricsCAD\UserDataCache) is compared with the content of the **Local root** folder and the **Roamable root** folder of the current user. If you have customized the files in these folders and if one or more of the corresponding files in the **UserDataCache** folder are more recent, the **BricsCAD user file manager** is launched asking you to keep your files or to overwrite them with the new version of these files. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/bricscaduserfilemanager.png})

1. Do one of the following:
   - Select a file in the file list, then click the **Yes** button to overwrite or click the **No** button to keep your file.
   - Repeat this procedure for each file in the list.
   - Click the **Yes to all** button to replace all your files with the more recent files from BricsCAD.
   - Click the **Stop Copying** button to keep all your files.
2. If you have decided not to update one or more files, an alert window displays. (\vref {fig:C:/Dev/Task-2448/images/staticb
3. Do one of the following:
   - Click the **Yes** button if you want to update one of these files later. In this case this procedure is restarted the next time you start BricsCAD.
   - Click the **No** button if you don't want to be reminded.

Figure 1.303

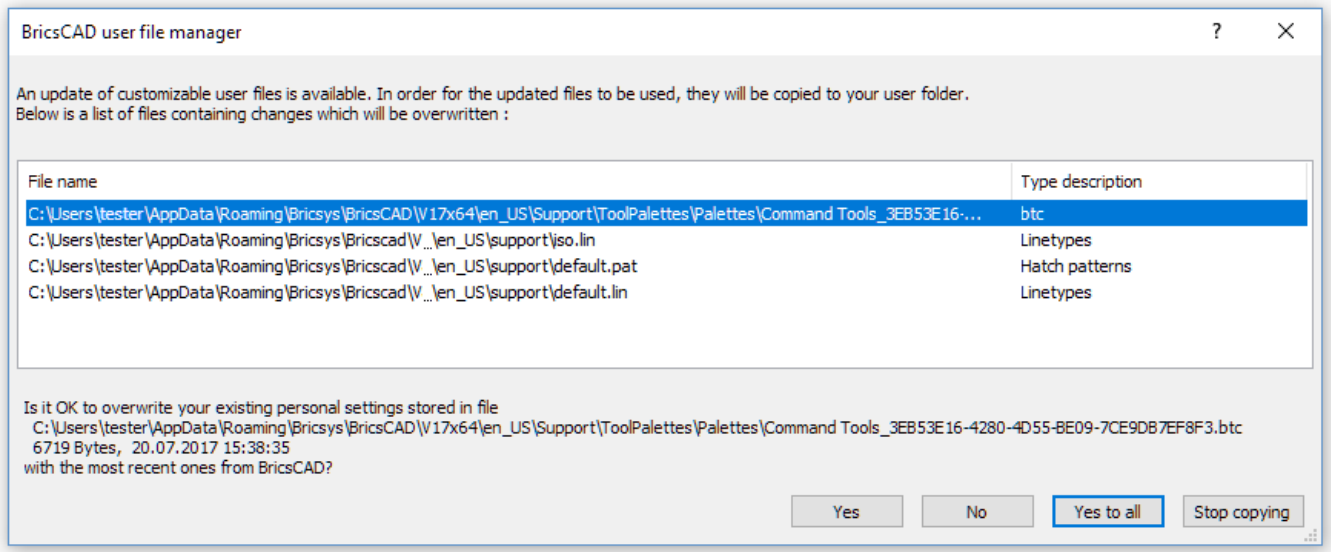
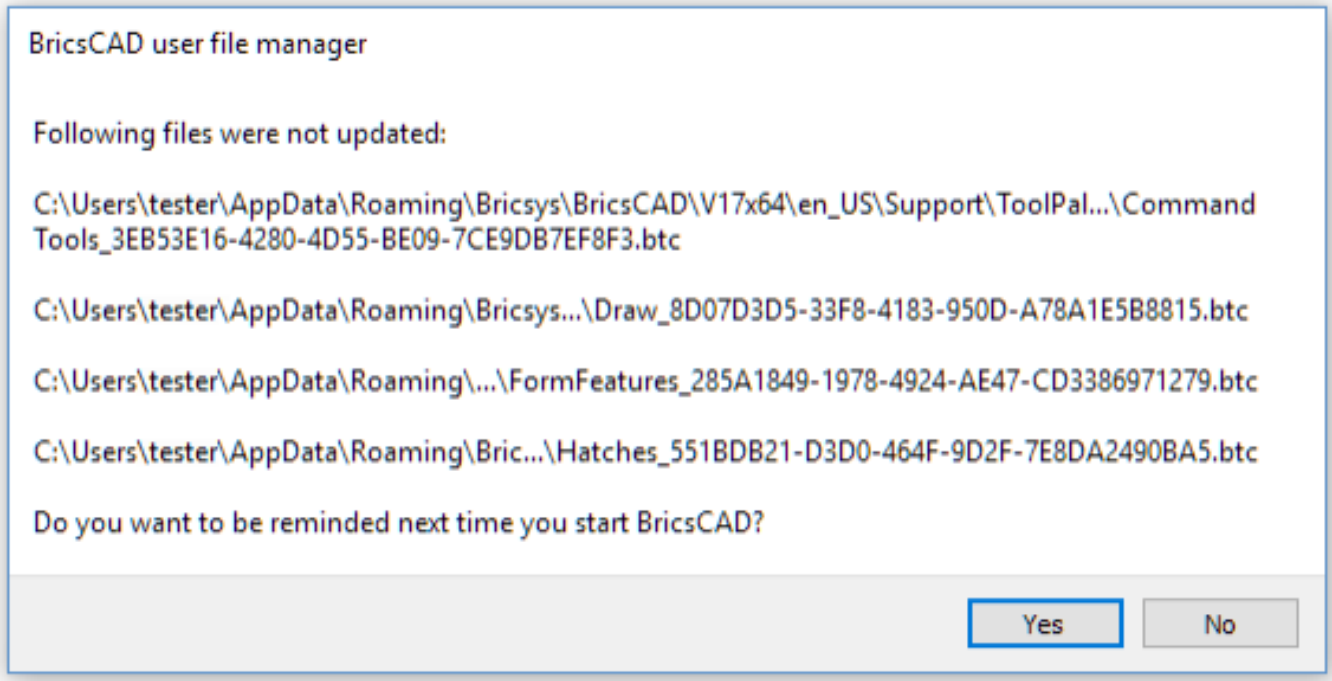


Figure 1.304



1.13.3 Understanding the Customize Dialog

Command: Customize (6.9.58)

What are the CUI files

Files with the CUI extension are Customize User Interface files for BricsCAD. These XML format files replace the previously used menu definition files (*.mnu *.mns). CUI files can be edited with the build-in interactive editor, launched by the Customize command. You can migrate CUI files between BricsCAD versions and combine them between different computers.

NOTE	The Load partial CUI file (1.13.4) **...** procedure accepts IntelliCAD menu files (*.mnu *.mns) and converts them to CUI files.

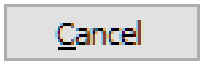
Opening the Customize dialog

To open the Customize dialog do one of the following:

- Choose **Customize** in the **Tools** menu
- In Windows and OS X: Hover over a toolbar or ribbon panel or a blank portion of the toolbar area, then right click and choose **Customize** in the context menu.
- In Linux: Hover over a blank portion of the menu bar, status bar or toolbar area, then right click and choose **Customize** in the context menu.
- Type **customize** or **cui** in the command window, then press Enter. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/

The components of the Customize dialog are:

1. File menu
2. Main customization file name and location
3. Browse for main customization file button
4. Search field: type a search string then click the looking glass icon, click again for the next hit.
5. Section tabs
6. Content of the currently selected section
7. Available tools tree
8. Properties grid, displaying the properties of the selected item
9. (\vref {fig:C:/help/enUS/V20/BCAD/Customization/01ManageyourcustomizationsButton.png}) Opens the customazation history. Here, settings can be reverted to defaults.
10. 

Saves changes and closes the Customize dialog. 

Closes the Customize dialog, without saving changes.

Tutorial: Saving Your Customizations with Partial CUI Files in BricsCAD Click here (1.3.11) to watch

Figure 1.305

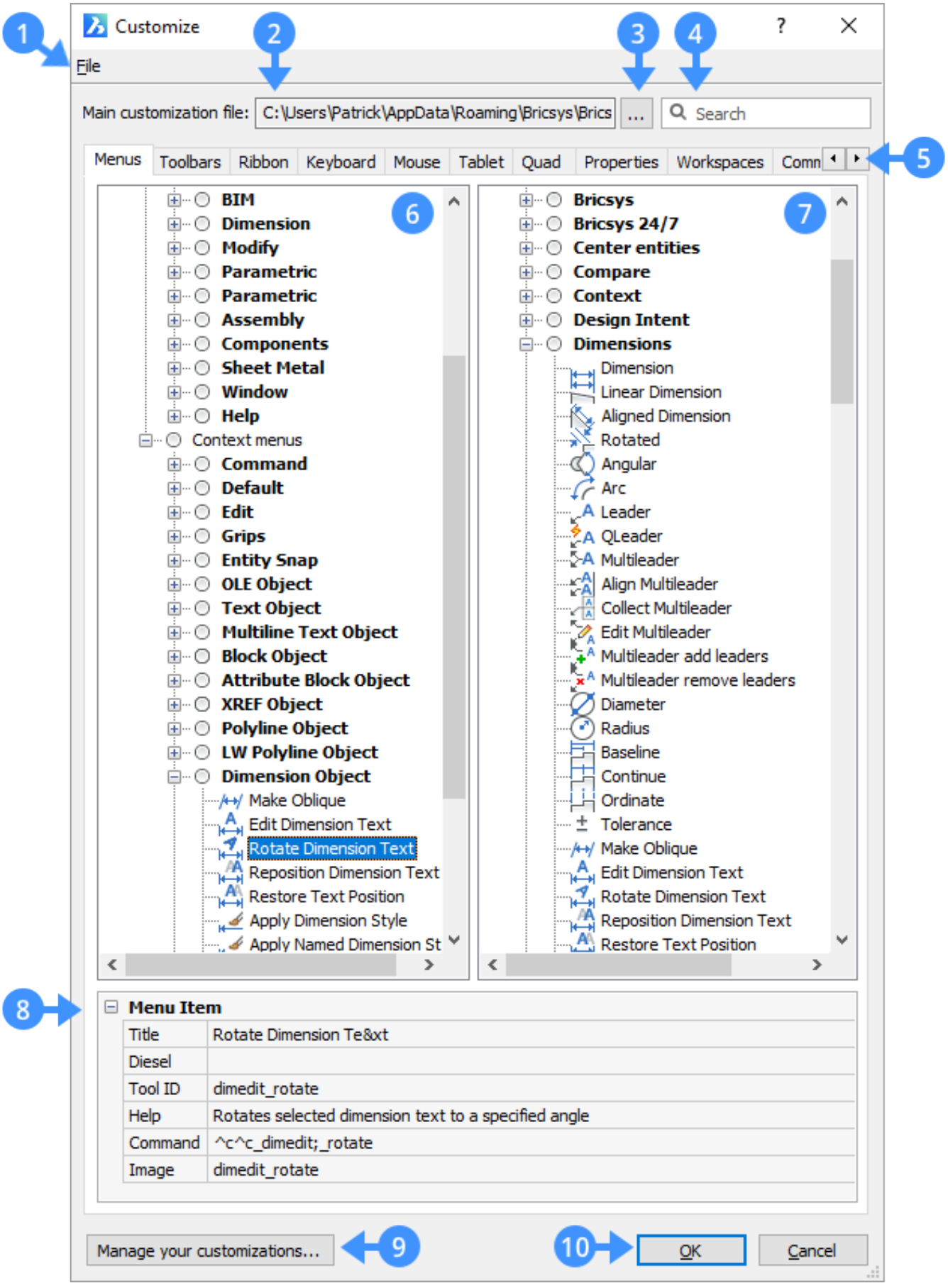
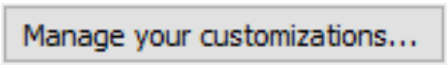

1
Customize
2
3
4
File
Main customization file:
C:\Users\Patrick\AppData\Roaming\Bricsys\Brics
...
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Properties
Workspaces
Comn
5
6
BIM
Dimension
Modify
Parametric
Parametric
Assembly
Components
Sheet Metal
Window
Help
Context menus
Command
Default
Edit
Grips
Entity Snap
OLE Object
Text Object
Multiline Text Object
Block Object
Attribute Block Object
XREF Object
Polyline Object
LW Polyline Object
Dimension Object
Make Oblique
Edit Dimension Text
Rotate Dimension Text
Reposition Dimension Text
Restore Text Position
Apply Dimension Style
Apply Named Dimension St
7
Bricsys
Bricsys 24/7
Center entities
Compare
Context
Design Intent
Dimensions
Dimension
Linear Dimension
Aligned Dimension
Rotated
Angular
Arc
Leader
QLeader
Multileader
Align Multileader
Collect Multileader
Edit Multileader
Multileader add leaders
Multileader remove leaders
Diameter
Radius
Baseline
Continue
Ordinate
Tolerance
Make Oblique
Edit Dimension Text
Rotate Dimension Text
Reposition Dimension Text
Restore Text Position
8
Menu Item
Title
Rotate Dimension Te&xt
Diesel
Tool ID
dimedit_rotate
Help
Rotates selected dimension text to a specified angle
Command
^c^c_dimedit;_rotate
Image
dimedit_rotate
Manage your customizations...
9
10
OK
Cancel

Figure 1.306



1.13.4 Loading and unloading CUI files

Commands: MenuLoad (6.21.14), CuiLoad (6.9.56), MenuUnload (6.21.15) and CuiUnload (6.9.57) When you install BricsCAD the default CUI files are copied to the **Support** folder of your Roamable root folder (1.13.2). (e.g. C:\Users\<user_name>\AppData\Roaming\Bricsys\Bricscad\Vxx.x\en_US\Support). When you launch BricsCAD the first time the **default.cui** file is loaded automatically. There are two procedures to load a CUI file:

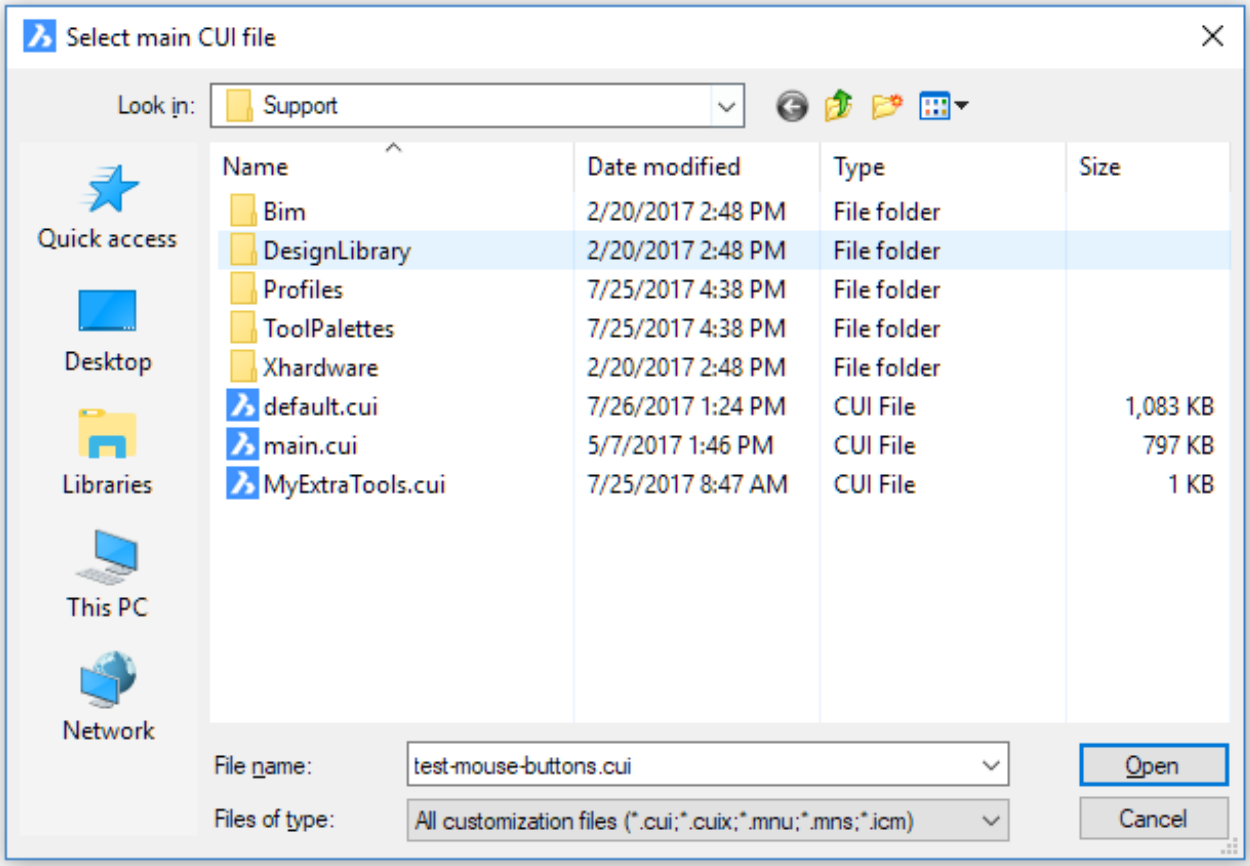
- Load main CUI file, which replaces all existing content with the content of the new CUI file.
- Load partial CUI file, which appends its content to the content of the currently loaded main CUI file.

NOTE	You can load an IntelliCAD menu file if you select **MNU files (*.mnu;*.mns)** in the **Files of Type** list box of the **Choose a Customization file** dialog in the Load main CUI file (1.13.4), Load partial CUI file (1.13.4) and Manage Customization Groups (1.13.4) procedures. The menu file will be converted to a CUI file automatically.

To load a main CUI file

1. Do one of the following:
   - Click the **Load Menu** tool button ( ) on the **Tools** toolbar.
   - Choose **Load Menu** in the **Tools** menu.
   - Type **menu** in the command bar, then press Enter.
   - Open the Customize dialog (1.13.3), then click the **Browse** button.
   - Open the Customize dialog (1.13.3), then choose **Load main CUI file ...** in the **File** menu.
2. The **Select main CUI file** dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/
   Do one of the following:
   - Double click a **CUI** file.
   - Select a **CUI** file, then click the **Open** button.
3. The selected **CUI** file is loaded. Close the **Customize** dialog.

Figure 1.307



To load a partial CUI file

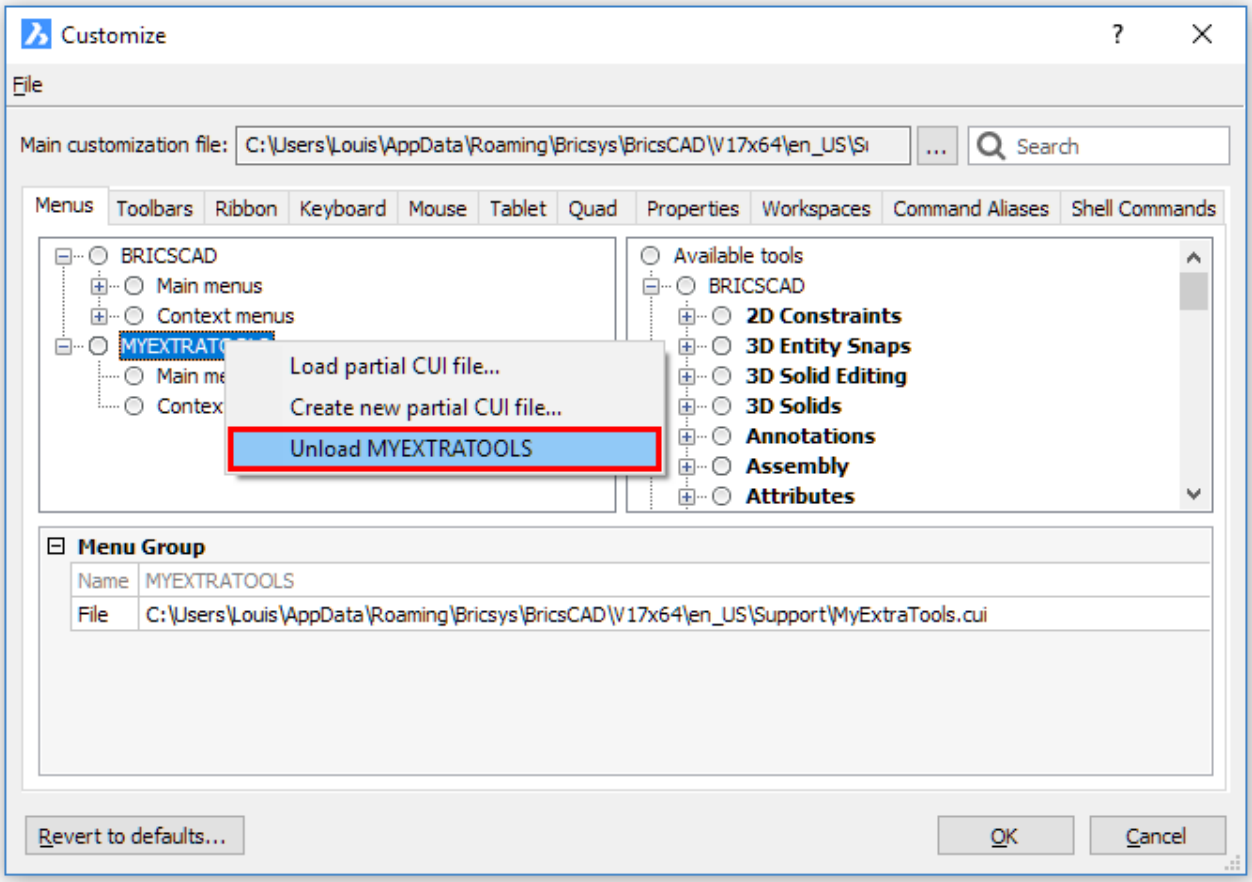
1. Open the Customize dialog (1.13.3), then choose **Load partial CUI file ...** in the **File** menu. The **Choose a Customization file** dialog displays.
2. Do one of the following:
   - Double click a CUI file.
   - Select a CUI file, then click the **Open** button.
3. **Optional**Repeat steps 1 and 2 to load another partial CUI file.
4. Do one of the following:
   - Click the **OK** button (OK) to confirm the loading of the partial CUI file(s) and close the **Customize** dialog.
   - Click the **Cancel** button to (Cancel) close the **Customize** dialog and unload the partial CUI file(s).

NOTE	If a CUIX file is selected in step 2 it will be converted to a CUI file on-the-fly. When a CUIX file contains bitmaps, they are copied into a .RESZ resource file, which are supported as an alternative to resource DLL (zip file containing tool bitmaps; works on all platforms). The following image formats are supported: BMP, JPG, PNG, ICO, GIF, TIFF, XBM, XPM.

To unload a partial CUI file

1. Open the Customize dialog (1.13.3).
2. In the **Menus** , **Toolbars** or **Keyboard** section select the partial CUI file you want to unload. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/02UnloadPartialCui.png})

Figure 1.308

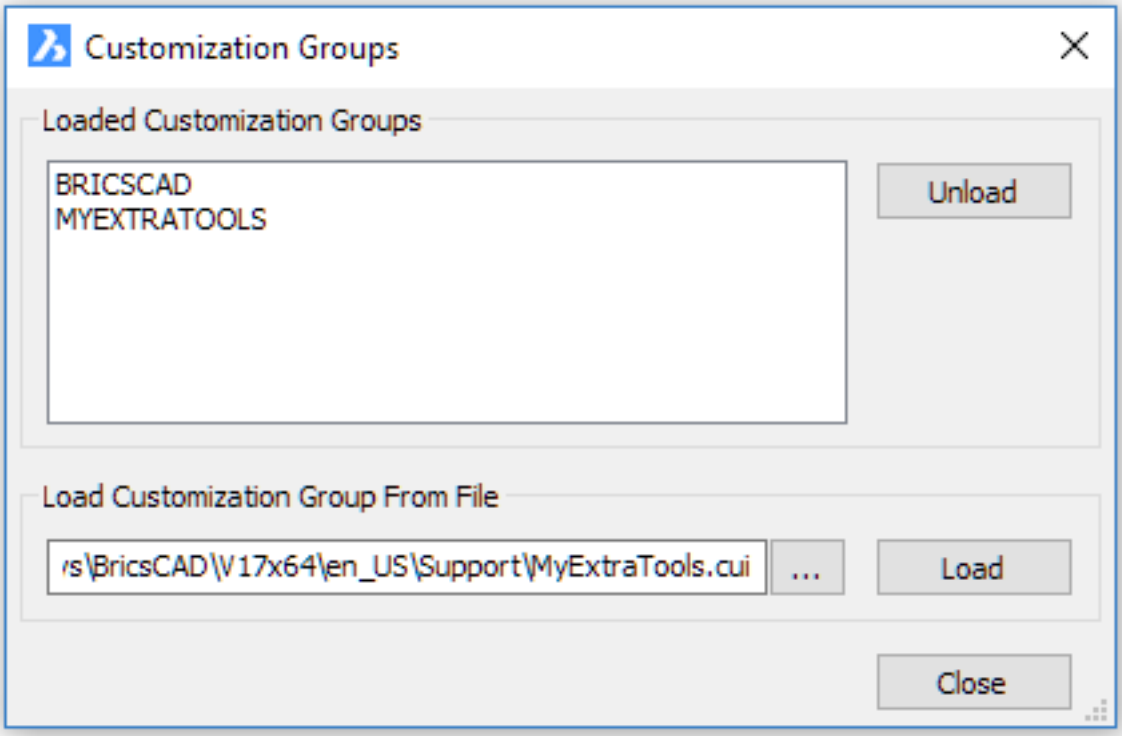

Customize
File
Main customization file:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V17x64\en_US\Si
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Properties
Workspaces
Command Aliases
Shell Commands
BRICSCAD
Main menus
Context menus
Load partial CUI file...
Create new partial CUI file...
Unload MYEXTRATOOLS
Available tools
BRICSCAD
2D Constraints
3D Entity Snaps
3D Solid Editing
3D Solids
Annotations
Assembly
Attributes
Menu Group
Name
MYEXTRATOOLS
File
C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V17x64\en_US\Support\MyExtraTools.cui
Revert to defaults...
OK
Cancel

Managing customization groups

1. Type **Menuload** in the command bar, then press Enter.
2. The **Customization groups** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/
3. **Optional**To load a partial CUI file, click the browse button (…) to select a partial CUI file, then click the **Load** button.
4. **Optional**To unload a partial CUI file, select the partial CUI file in the **Loaded Customization Groups** list, then click the **Unload** button.
5. Click the **Close** button to stop.

Tutorial: Saving Your Customizations with Partial CUI Files in BricsCAD Click here (1.3.8) to watch

Figure 1.309


Customization Groups
Loaded Customization Groups
BRICSCAD
MYEXTRATOOLS
Unload
Load Customization Group From File
rs\BricsCAD\V17x64\en_US\Support\MyExtraTools.cui
...
Load
Close

1.13.5 Creating custom tools

General procedure to create custom tools

1. Create a Partial CUI file.
2. Optional:Add toolbox(es) in a Menu Group.
3. Optional:Add tools in a toolbox
4. Optional:Add main menu(s)
5. Optional:Add sub menu(s)
6. Optional:Add context menu(s)
7. Optional:Add toolbar(s)
8. Optional:Add flyout(s) to toolbar(s)
9. Optional:Add control(s) to toolbar(s)
10. Optional:Add keyboard shortcuts
11. Optional:Add existing tools to menus, toolbars or shortcuts
12. Optional:Create new tools in menus, toolbars or shortcuts
13. Optional:Change the tool order in menus and toolbars

Tutorial: Easy Menu Customization Click here (1.3.10) to watch

Tutorial: Advanced Menu Customization Click here (1.3.12) to watch

To create a Partial CUI file

1. In the Customize dialog (1.13.3) do one of the following:
   - In the **File** menu, choose **Create new partial CUI file ...**
   - Place the cursor on Menu group, then right click an choose **Create new partial CUI file ...** in the context menu.
2. The **Create a Customization File** dialog opens. By default the new partial CUI file will be created in the **Support** folder of the Roamable root folder (1.13.2).B **Optional** select a different folder to save the CUI file.
3. Type a name for the new CUI file in the **File name** field.
4. Click the **Save** button. The newly created **Partial CUI file** is loaded and a **Menu Group** of the same name is created in the **Menus** tree, **Toolbars** tree, **Keyboards** tree and **Available Tools** tree.

To add a toolbox in a menu group

1. In the Customize dialog (1.13.3) select a menu group.
2. Right click and choose **Append toolbox** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
3. Type a name for the new **toolbox** in the **Title** field of the **Add toolbox** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbri
4. Click the **OK** button. The toolbox is added in the **Available Tools** tree.
5. Click the **Apply** button to save the definition.

Figure 1.310

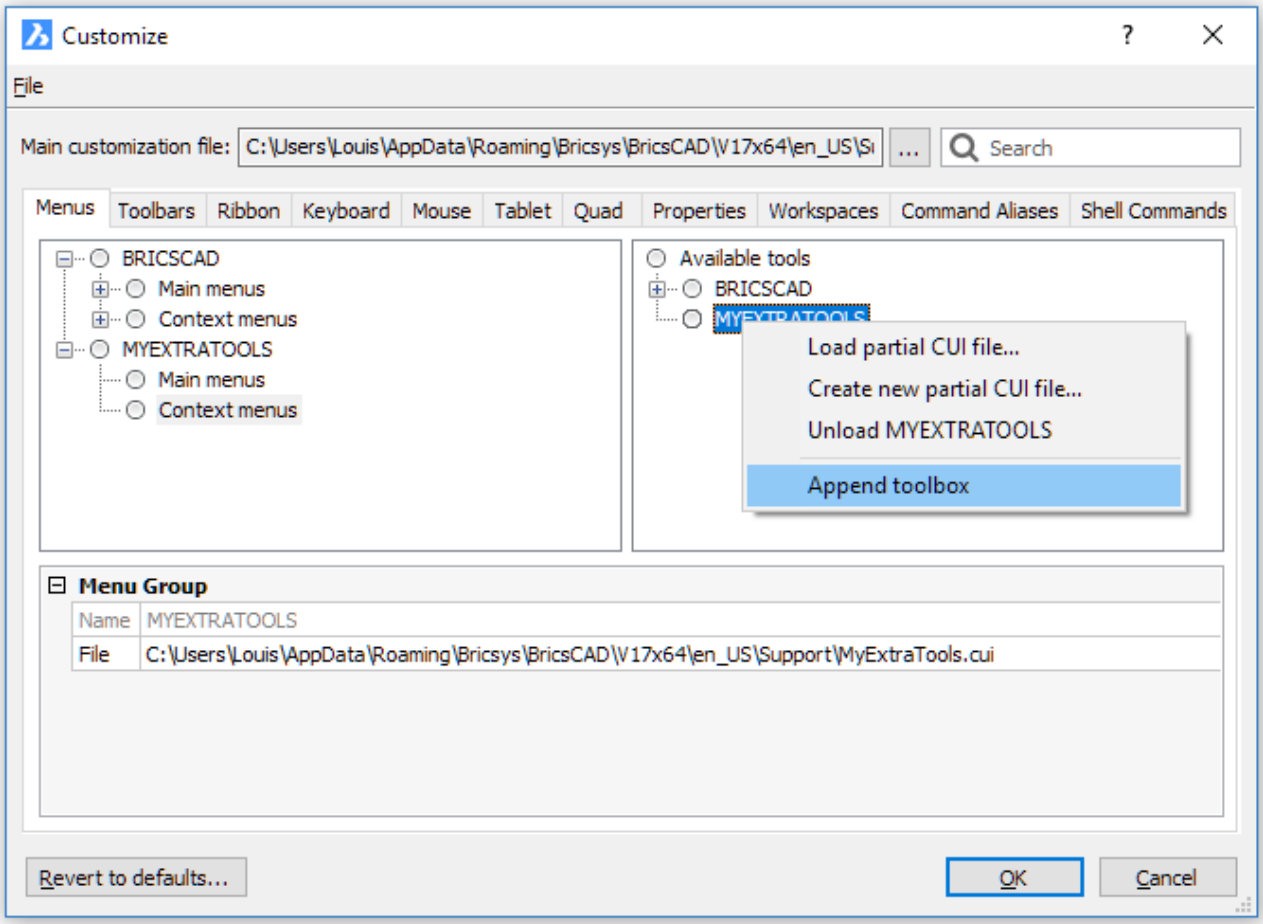
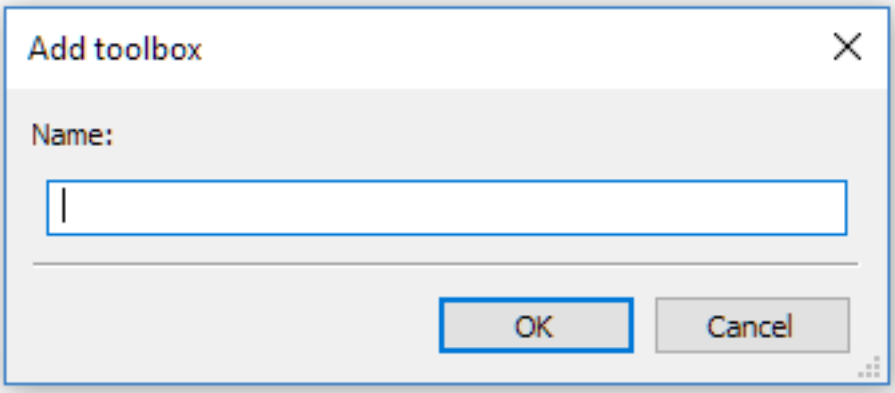


Figure 1.311



To add a tool in a toolbox

1. In the Customize dialog (1.13.3) select a toolbox in the **Available Tools** tree.
2. Right click and choose **Append tool** in the context menu. The **Add tool** dialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/03AddTool.png})
3. Type a name for the new tool in the **Title** field of the **Add tool** dialog.
4. **Optional**Type a help string in the **Help** field of the properties grid.
5. Type a command or command macro in the **Command** field of the properties grid. e.g.: ^c^c_-layer;m;InteriorWalls;;Mline In the above example a macro is created which makes the **InteriorWalls** layer current, then launches the **Mline** command. If the layer does not exist yet, it is created first.
6. Click the **Browse** button (…), then select an image for the new tool. Do one of the following:
   - Select **Builtin** , then choose one of the built-in icons. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
   Select **Bitmap file** , and choose a bmp, gif, jpg, jpeg or png image file.
   - Click the **OK** button.
7. The tool is added in the toolbox. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customi

NOTE

Syntax to create macros:

- **^c^c** is the equivalent of pressing the Esc key twice, which first ends a running command if necessary.
- an underscore (_) in front of a command name ensures that the macro still works in a non-English version of BRICSCAD.
- a point (.) in front of a command name ensures that the macro still works if this command is temporarily undefined (see the UNDEFINE (6.30.4) and REDEFINE (6.26.7) commands).
- a minus sign (-) in front of a command launches the command line version of a command, thus preventing the command dialog window to open.
- a semicolon (;) is the equivalent of pressing the Enter key
- a backslash (\) temporarily pauses the macro to allow user input, e.g. specifying a point.

Figure 1.312

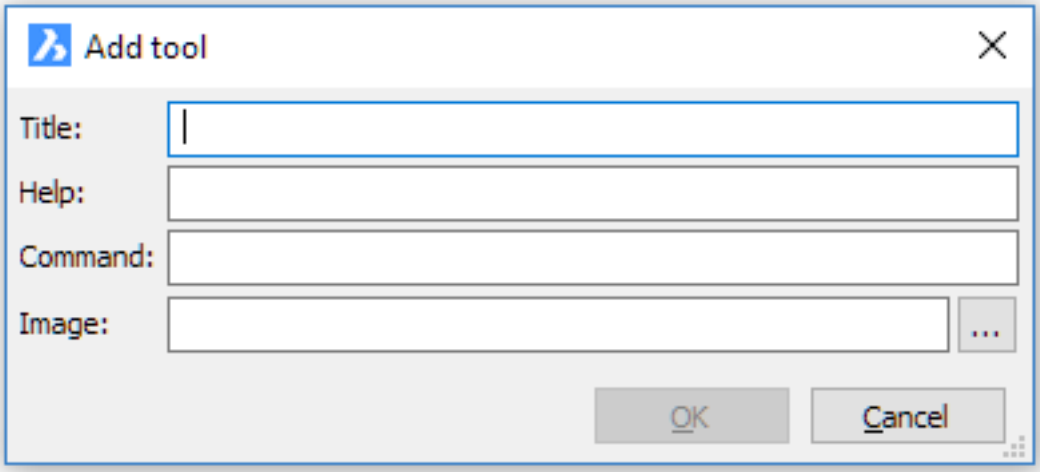


Figure 1.313

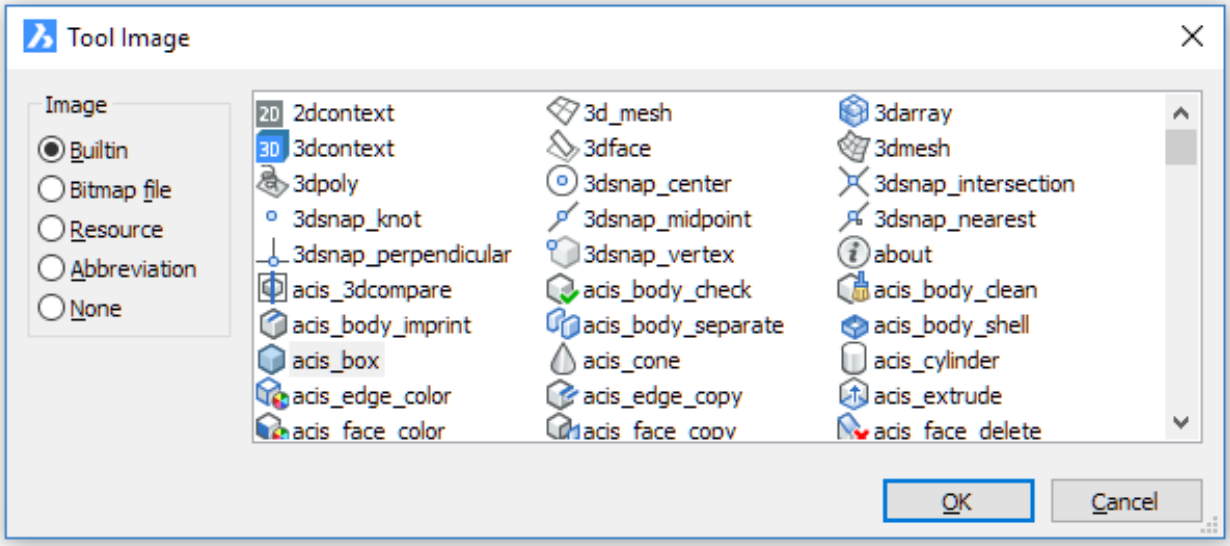
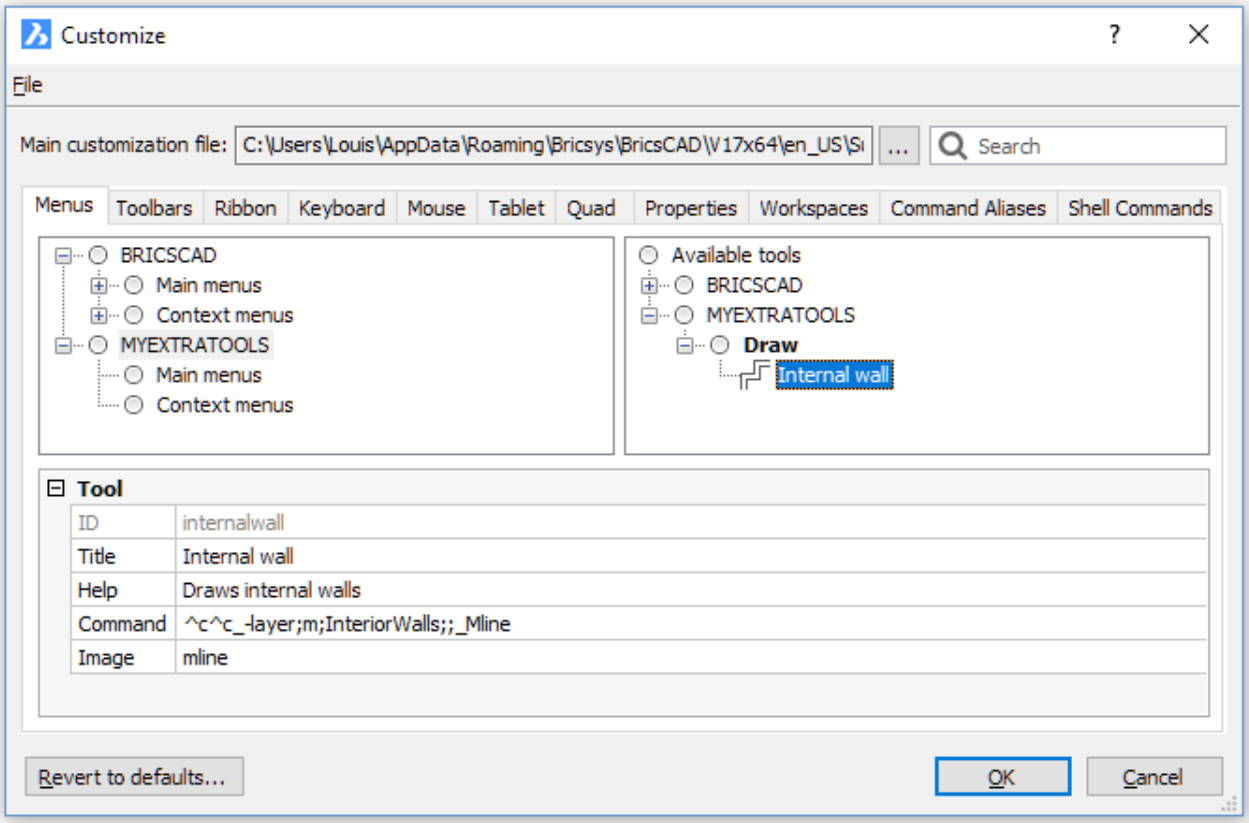


Figure 1.314



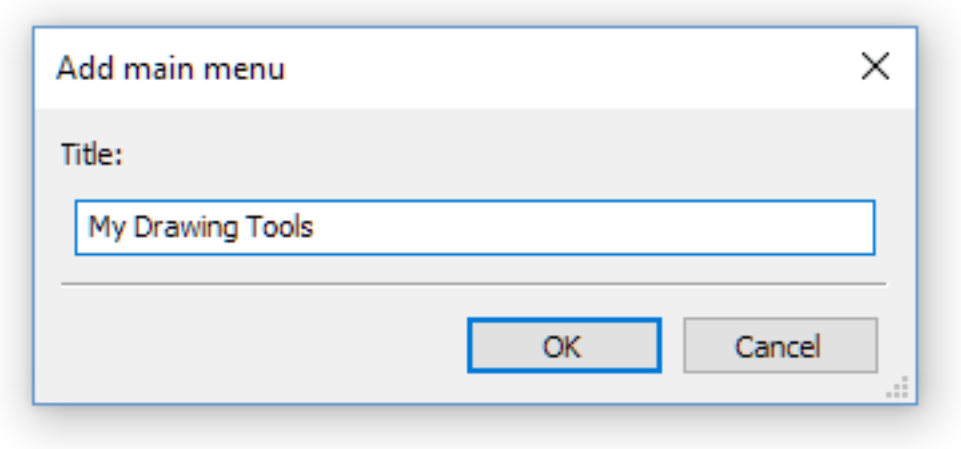
To add a main menu

New menus are added above the selected existing menu or as the first menu if you select the **Main Menus** subgroup.

1. In the Customize dialog (1.13.3) click the **Menus** tab.
2. Do one of the following:
   - Click the **Main Menus** subgroup of the menu group, then right click and choose **Append main menu** in the context menu.
   - Select an existing menu in the **Main Menus** subgroup, then right click and choose **Insert main menu** in the context menu.
3. Type a name in the **Title** field of the **Add main menu** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
4. Click the **OK** button.
5. **Optional** Repeat steps 2 through 5 to add more menus.
6. Click the **Apply** button to save the definition(s).

NOTE	The new main menu is not added to the **Menu bar** until at least one menu items is added to it.

Figure 1.315



To add a submenu to a menu

New submenus are added above the selected existing item or as the first item if you select the menu.

1. In the Customize dialog (1.13.3) click the **Menus** tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. If collapsed, expand the **Main menus** group of the menu group.
4. Do one of the following:
   - Select the menu, then right click and choose **Append submenu** in the context menu.
   - Select an existing item in the menu, then right click and choose **Insert submenu** in the context menu.
5. Type a name in the **Title** field of the **Add submenu** dialog.
6. Click the **OK** button.

To add a context menu

New context menus are added above the selected existing context menu or as the first context menu if you select the **Context Menus** subgroup.

1. In the Customize dialog (1.13.3) click the **Menus** tab.
2. Do one of the following:
   - Select the **Context Menus** subgroup of the menu group, then right click and choose **Append context menu** in the context menu.
   - Select an existing context menu, then right click and choose **Insert context** menu in the context menu.
3. Select a menu in the **menu** list of the **Add context menu** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
4. **Optional:**Select **As regular context menu**, then click the **Add Context...** button. The **Add context alias** dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/03AddCont Choose:
   - CMCOMMAND: to replace the default **Command** context menu.
   - CMDEFAULT: to replace the default **Default** context menu.
   - CMEDT: to replace the default **Edit** context menu.
   - GRIPS: to replace the default **Grips** context menu.
   - SNAP: to replace the default **Entity Snap** context menu.
5. Click the **OK** button to confirm.
6. **Optional** repeat the previous step to add another alias. Select an alias, then click the **Remove** button to remove the alias. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/03AddContextM
7. **Optional**Select **As context menu on specific entities:** , then click the **Add Entity Type(s)...** button. The **Add Entity Alias** dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customi

Figure 1.316

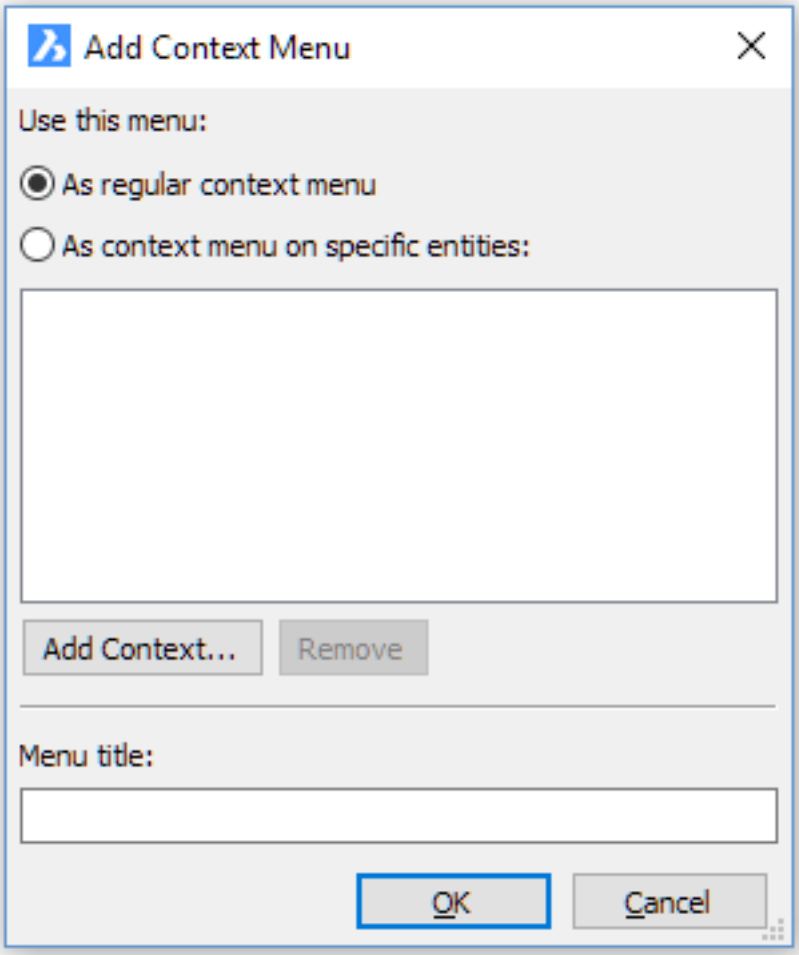


Figure 1.317

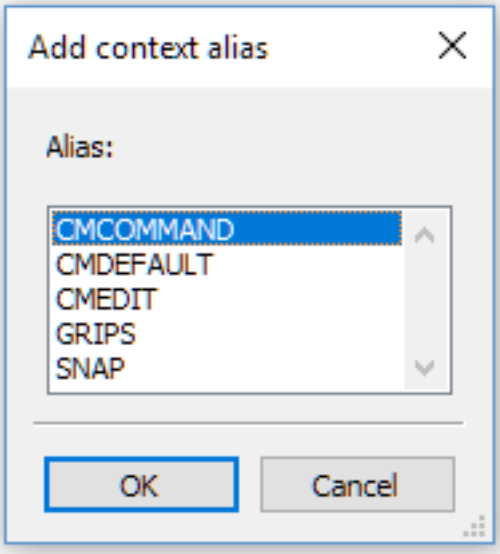
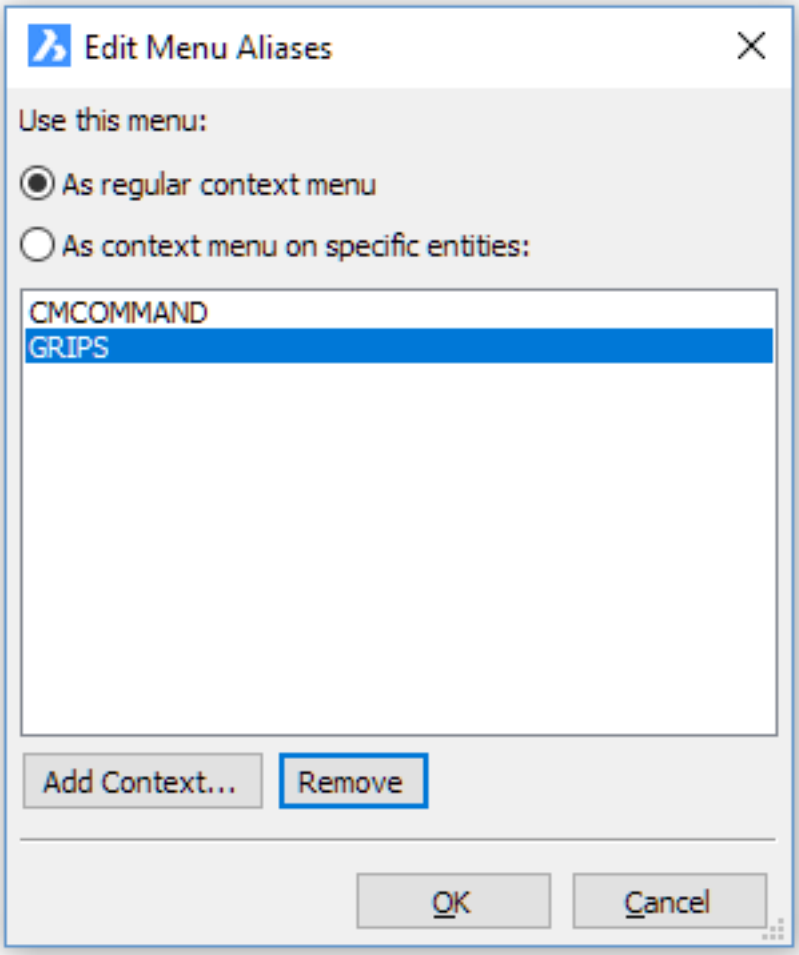


Figure 1.318

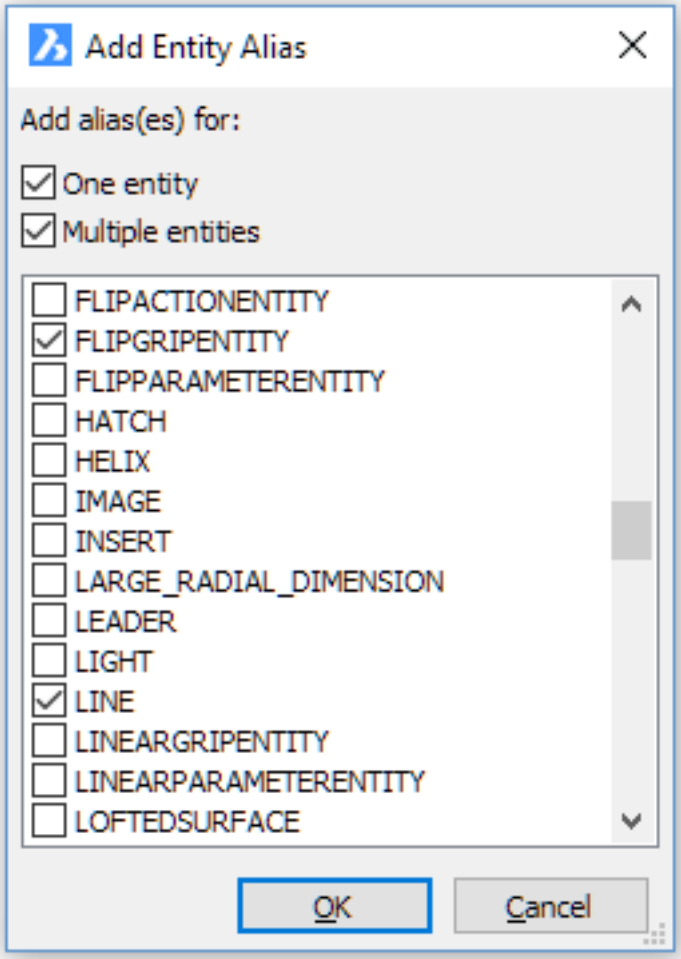

Edit Menu Aliases
Use this menu:
As regular context menu
As context menu on specific entities:
CMCOMMAND
GRIPS
Add Context...
Remove
OK
Cancel

Select one or more entity types in the list. Do one of the following, then click the **OK** button to confirm.

- Check the **One entity** option: the context menu applies if one entity is selected only.
- Check **Multiple entities** : the context menu applies in multiple entities are selected only.
- Check both options: the context menu applies if one or more entities are selected.

8. Type a name in the Menu title field.
9. Click the **OK** button to save the context menu.
10. **Optional:** Repeat steps 2 through 5 to add more context menus.

Figure 1.319


Add Entity Alias
Add alias(es) for:
One entity
Multiple entities
FLIPACTIONENTITY
FLIPGRIPENTITY
FLIPPARAMETERENTITY
HATCH
HELIX
IMAGE
INSERT
LARGE_RADIAL_DIMENSION
LEADER
LIGHT
LINE
LINEARGRIPENTITY
LINEARPARAMETERENTITY
LOFTEDSURFACE
OK
Cancel

To add a toolbar

New toolbars are added above the selected existing toolbar or as the first toolbar if you select the menu group.

1. In the Customize dialog (1.13.3) click the **Toolbars** tab.
2. Do one of the following:
   - Select a menu group, then right click and choose **Append toolbar** in the context menu.
   - Select an existing toolbar in a menu group, then right click and choose **Insert toolbar** in the context menu.
3. Type a name in the **Title** field of the **Add toolbar** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
4. Click the **OK** button.
5. **Optional:**Adjust the Toolbar properties. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/C
   - **Title**: The toolbar name must include alphanumeric characters with no punctuation other than a dash (–) or an underscore (_). The **Title** along with the **Alias** enables the toolbar to be referenced programmatically.
   - **Position**: Defines the display position of the toolbar when the CUI file is loaded. The options are: **Floating**, **Top**, **Left**, **Bottom** or **Right**. The latter four refer to the window edge where the toolbar will be docked.
   - **Display**: Defines whether the toolbar displays or not when the CUI file is loaded.
   - **Rows**: Specifies the number of rows when the toolbar position is floating. View bars with controls cannot have multiple rows.
   - **Xval**: When the position is Floating, specifies the X coordinate in pixels, measured from the left edge of the screen to the left side of the toolbar.
   - **Yval**: When the position is Floating, specifies the Y coordinate in pixels, measured from the top edge of the screen to the top of the toolbar.
6. Click the **Apply** button to save the definition(s).

NOTE	The new toolbar is loaded only if the **Display** property is set to **Show**. See Opening a Toolbar (1.12.3) for more information on opening toolbars.

Figure 1.320

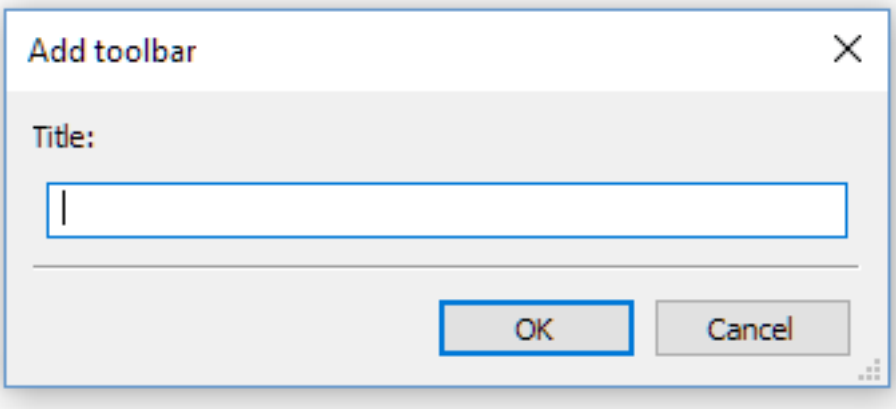


Figure 1.321

Toolbar	
Title	My Drawing Tools
Alias	TB_MyDrawingTools
Position	Floating
Visible	Hide
Rows	1
Xval	
Yval	

To add a flyout to a toolbar

1. In the Customize dialog (1.13.3) click the **Toolbars** tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. Do one of the following
   - Select a toolbar, then right click and choose **Append Flyout** in the context menu, to add the flyout at the end of the toolbar.
   - Expand the toolbar, right click a tool then choose **Insert flyout** in the context menu, to add the flyout above the selected tool.
4. The **Add flyout** dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Custo
   Type a name in the **Title** field of the **Add flyout** dialog, then click the **OK** button. A new empty toolbar is created and inserted as a flyout in the selected toolbar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/B
5. Do one of the following:
   - Click the **Target Alias** field, then click the settings field and select an existing toolbar in the drop-down list.
   - Add tools to the new toolbar.

Figure 1.322

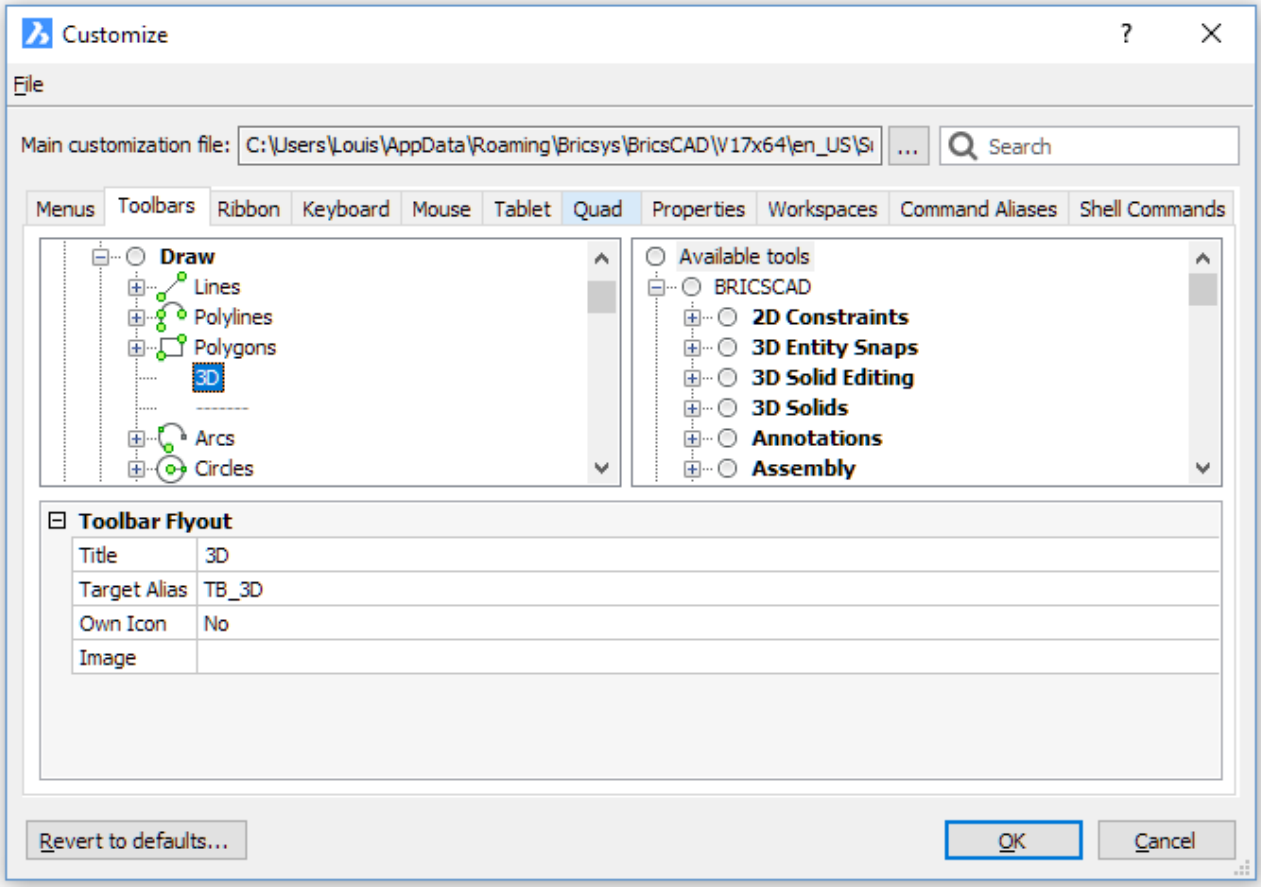
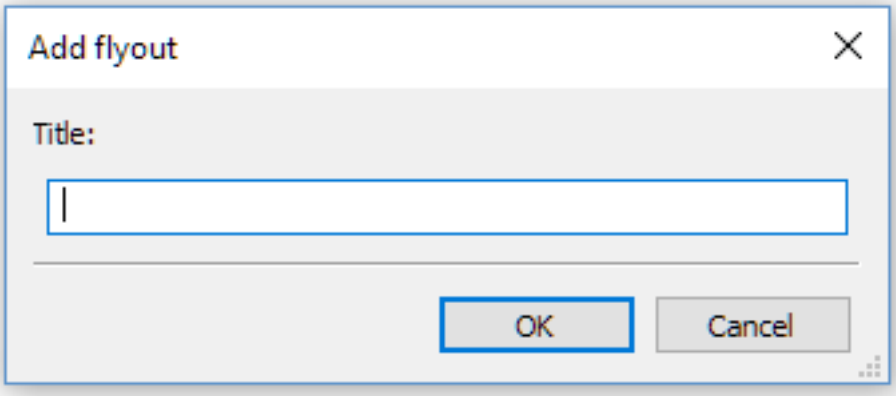


Figure 1.323



To add a control to a toolbar

1. In the Customize dialog (1.13.3) click the **Toolbars** tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. Do one of the following:
   - Select the toolbar, then right click and choose **Append control** in the context menu. The new control will be added as the first item in the toolbar.
   - Select an existing item in the toolbar, then right click and choose **Insert control** in the context menu. The new control will be added above the selected existing item.
4. Select a control in the **Add control** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/
5. Click the **OK** button.
6. To specify the width of the control, do one of the following:
   - Accept the **Default** width.
   - Type the width in pixels in the **Width** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/B

NOTE	Type **Default** or **0** (zero) in the **Width** field to restore the default width of the control field.

Figure 1.324

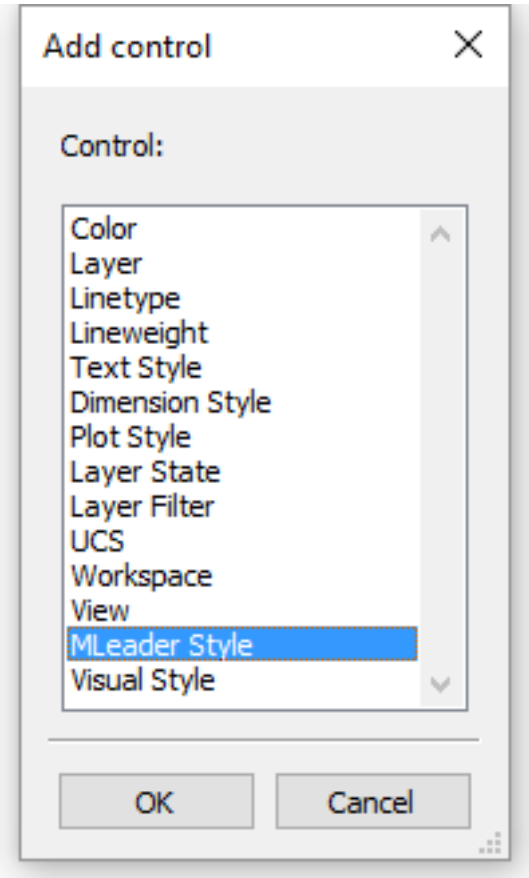
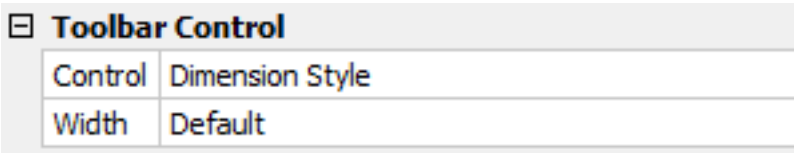


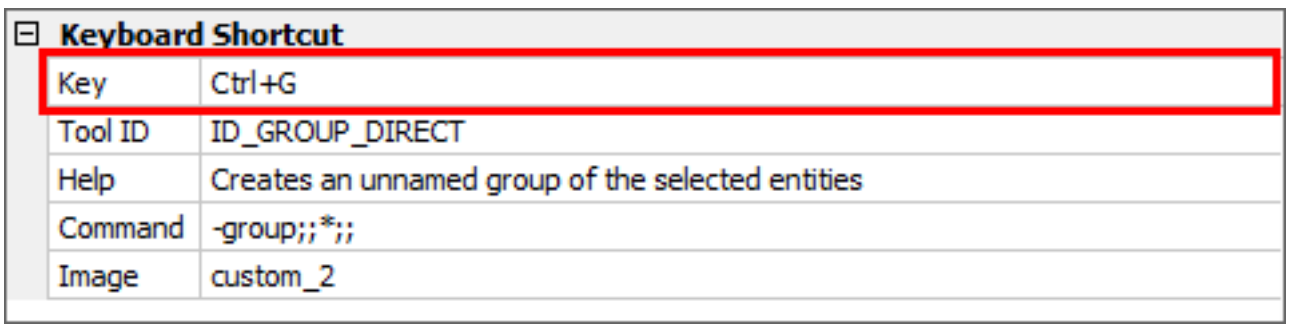
Figure 1.325



To add a keyboard shortcut

1. In the Customize dialog (1.13.3) click the **Keyboard** tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. Do one of the following:
   - Select the menu group, then right click and choose **Append shortcut** in the context menu. The new shortcut will be added at the end of the shortcut list.
   - Select an existing shortcut in the toolbar, then right click and choose **Insert shortcut** in the context menu. The new shortcut will be added above the selected existing shortcut.
4. Do one of the following:
   - Proceed with the **Add an existing tool** procedure.
   - Proceed with the **Add a new tool** procedure.
5. In the **Properties** grid of the new shortcut select the **Key** field, then press the key combination you want to assign to the shortcut. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/03K

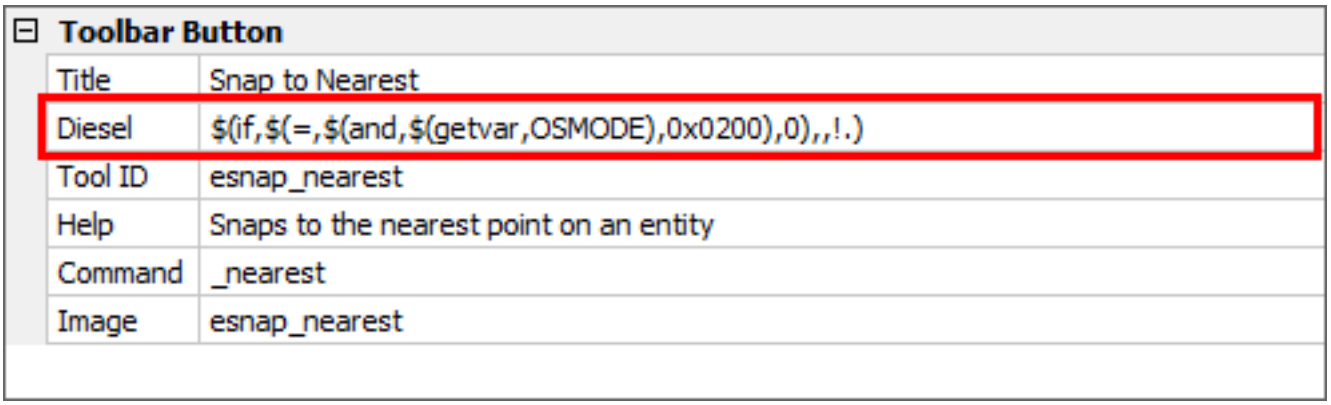
Figure 1.326



To add an existing tool to a menu, toolbar or shortcut

1. In the Customize dialog (1.13.3) select the item you want to add an existing tool to.
2. In the **Available Tools** pane, expand the toolbox that contains the tool you want to add.
3. Drag the tool to the target item:
   - Drop the tool on top of the item name to append the tool.
   - Drop the tool on top of another tool to insert the new tool above the existing tool.
4. **Optional:**Edit the tool in the settings grid, e.g. to add a diesel expression in the **Diesel** field. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/03ToolbarBu

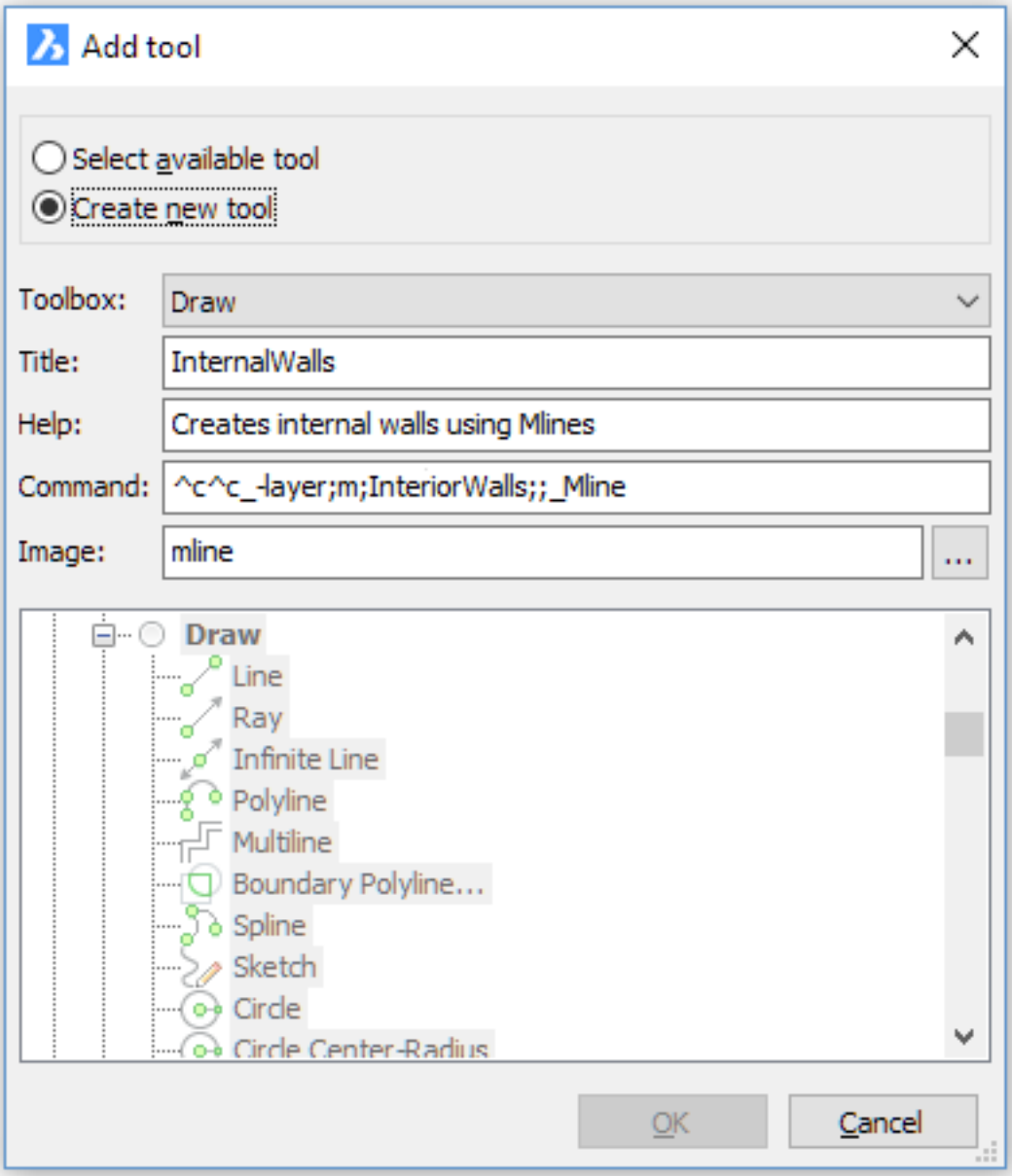
Figure 1.327



To create a new tool

1. In the **Add Tool** dialog choose **Create new tool.**
2. Select the **Toolbox** in which you want to create the new tool.
3. Type a name for the new tool in the **Title** field of the **Add tool** dialog.
4. **Optional:**Type a help string in the **Help** field of the properties grid.
5. Type a command or command macro in the **Command** field of the properties grid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/03AddNewToolDialog.png}) In the screenshot above the **Command** field contains a macro that makes the **InteriorWalls** layer current, then launches the **Mline** command.If the layer does not exist yet, it is created first.
6. Click the **Browse** button (), then select an image for the new tool.
7. Do one of the following:
   - Select **Builtin**, then choose one of the built-in icons. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en

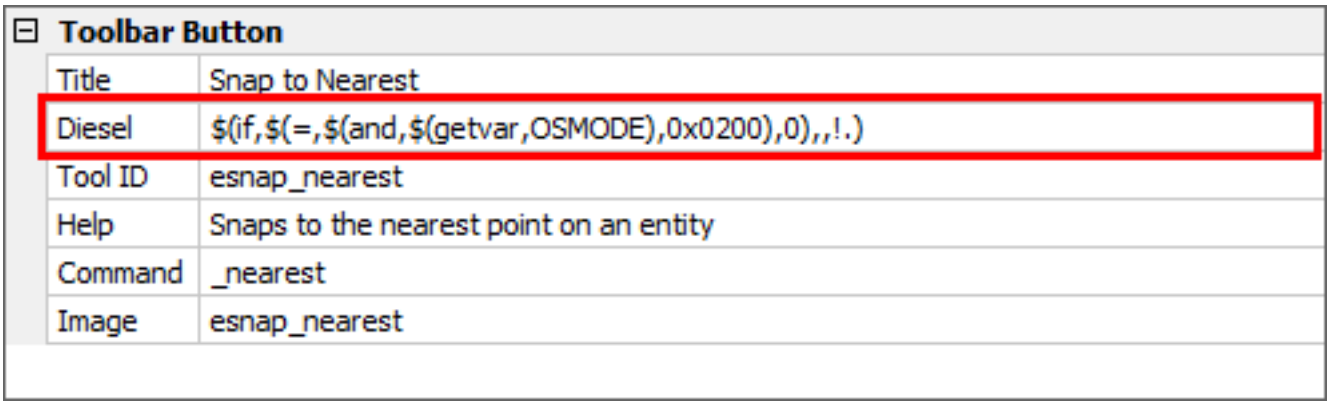
Figure 1.328


Add tool
Select available tool
Create new tool
Toolbox:
Draw
Title:
InternalWalls
Help:
Creates internal walls using Mlines
Command:
^c^c_-layer;m;InteriorWalls;;_Mline
Image:
mline
...
Draw
Line
Ray
Infinite Line
Polyline
Multiline
Boundary Polyline...
Spline
Sketch
Circle
Circle Center-Radius
OK
Cancel

- Select **Bitmap file** , and choose a bmp, gif, jpg, jpeg or png image file.

8. Click the **OK** button to create the new tool.

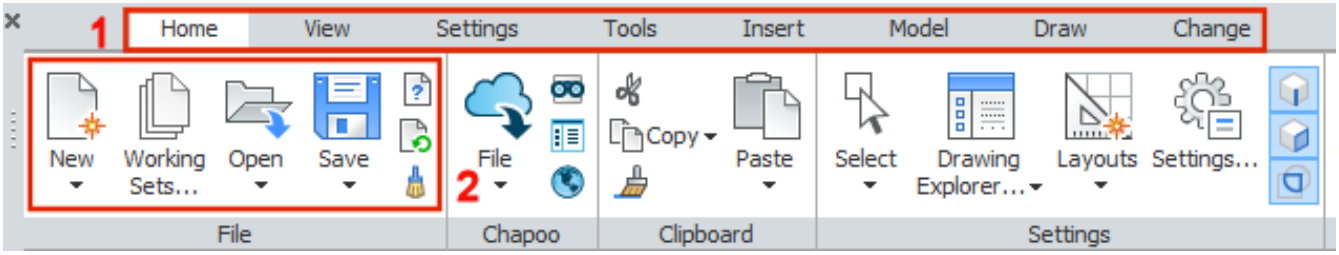
Figure 1.329



To change the order of the tools in a menu or toolbar

1. In the Customize dialog (1.13.3) select the menu or toolbar.
2. Drag the tool to its new position:
   - Drop the tool on top of the item name to move it to the last position.
   - Drop the tool on top of another tool to move the tool above this tool.

Figure 1.330



1.13.6 Ribbon

Commands: Ribbon (6.26.31) and RibbonClose (6.26.33) A ribbon organizes tools in a series of **Panels** (2) which are grouped in different **Tabs** (1). The number of tabs and their order depends on the current workspace and is defined in the **Ribbon** node of the workspace (see Editing a Workspace (1.13.10)). (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/RibbonGeneral.pn
The RibbonDockedHeight (1.13.11) system variable stores the preferred height of the Ribbon bar. Values between 0 and 500 are accepted. Set the value to 0 (zero) for automatic height calculation.

Ribbon Panels

A ribbon panel is composed of one or more rows. In a row you can add one or more command buttons or row panels, which in turn can hold one or more command buttons.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/RibbonPanel.png
Ribbon Panel
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/RibbonPanelStru
Structure of the Ribbon Panel in the Customize dialog

The **Entity Snaps** ribbon panel above is composed of:

- a launcher (1)
- a single row, holding three command buttons (2), a row panel (3) with three rows of four command buttons each, two command buttons (4) and a row panel (5) with three rows of one command each.

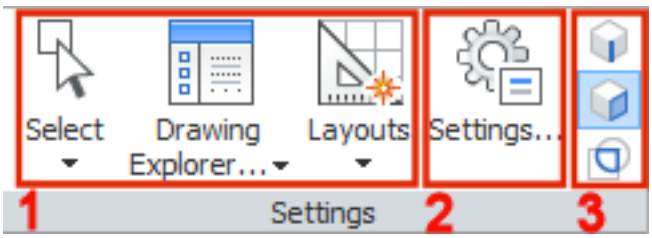
Ribbon Buttons

The following button types are available:

- Command button (2): launches a command
- Split button (1): groups a series of command buttons, much like a flyout toolbar.
- Toggle button (3): toggles a setting; shows the current status of setting

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/

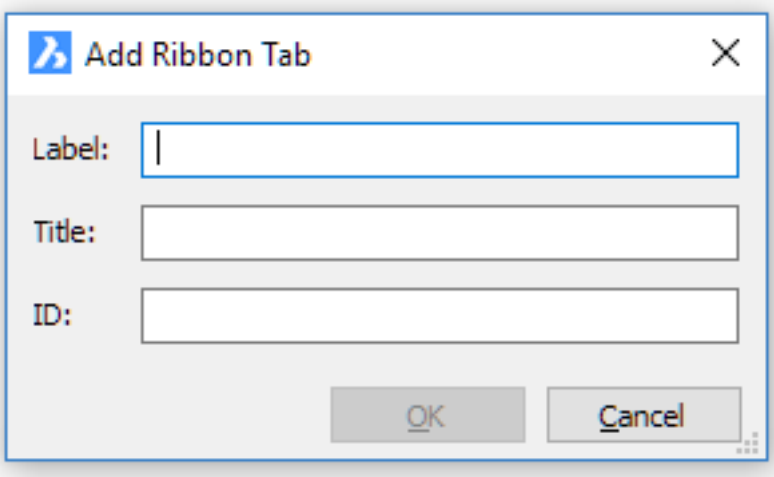
Figure 1.331



Creating a ribbon tab

1. Open the Customize dialog (1.13.3).
2. Click the **Ribbon** tab on the **Customize** dialog.
3. Right click the **Ribbon tabs** group and choose **Append ribbon tab** in the context menu.
4. Fill out the fields in the **Add Ribbon Tab** dialog box: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/
Label: Caption on the ribbon tab. Title: Used in the **Select Ribbon Tab** dialog when selecting ribbon tabs for a workspace (see Editing a Workspace (1.13.10)). ID: A unique name to identify the ribbon tab.
5. Click the **OK** button.

Figure 1.332



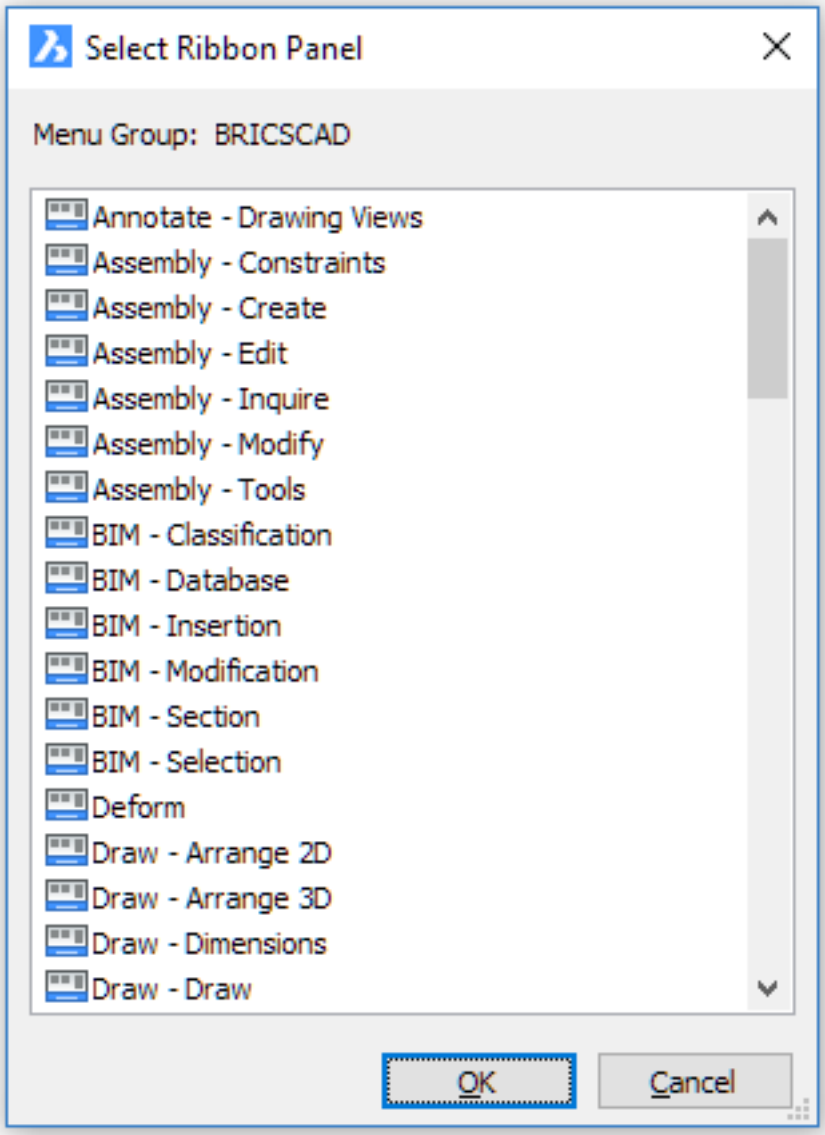
Editing a ribbon tab

1. Open the Customize dialog (1.13.3).
2. Click the **Ribbon** tab on the **Customize** dialog.
3. Expand the **Ribbon tabs** group.

To add a panel to a ribbon tab

1. Do one of the following:
   - Right click the ribbon tab and choose **Append panel** in the context menu, to add a panel at the end.
   - Right click an existing panel and choose **Insert panel** in the context menu, to ad a panel above the existing panel.
2. Select the panel in the **Select Ribbon Panel** dialog, then click the **OK** button. (\vref {fig:C:/Dev/Task-2448/images/wwwbri
3. **Optional:**Drag the panel to a different position.

Figure 1.333


Select Ribbon Panel
Menu Group: BRICSCAD
Annotate - Drawing Views
Assembly - Constraints
Assembly - Create
Assembly - Edit
Assembly - Inquire
Assembly - Modify
Assembly - Tools
BIM - Classification
BIM - Database
BIM - Insertion
BIM - Modification
BIM - Section
BIM - Selection
Deform
Draw - Arrange 2D
Draw - Arrange 3D
Draw - Dimensions
Draw - Draw
OK
Cancel

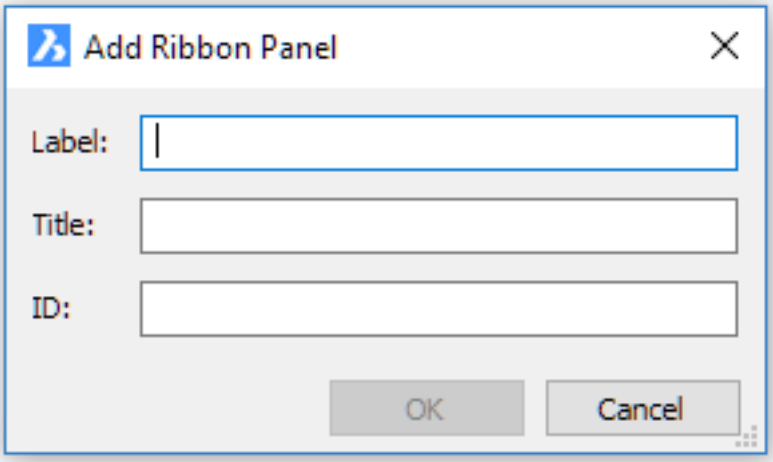
To remove a panel from a ribbon tab

1. Select the panel.
2. Do one of the following:
3. Press the **Delete** key.
4. Right click and choose **Remove** in the context menu.

Creating a ribbon panel

1. Open the Customize dialog (1.13.3).
2. Click the **Ribbon** tab on the **Customize** dialog.
3. Right click the **Ribbon Panels** group and choose **Append ribbon panel** in the context menu.
4. Fill out the fields in the Add Ribbon Panel dialog box: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/hel
   Label: Caption of the panel in the ribbon. Title: Used in the **Select Ribbon Panel** dialog when selecting the panels for a ribbon tab ID: A unique name to identify the ribbon panel.
5. Click the **OK** button.

Figure 1.334



Editing a ribbon panel

1. Open the Customize dialog (1.13.3).
2. Click the **Ribbon** tab on the **Customize** dialog.
3. Expand the **Ribbon Panels** group.
4. Expand the ribbon panel.

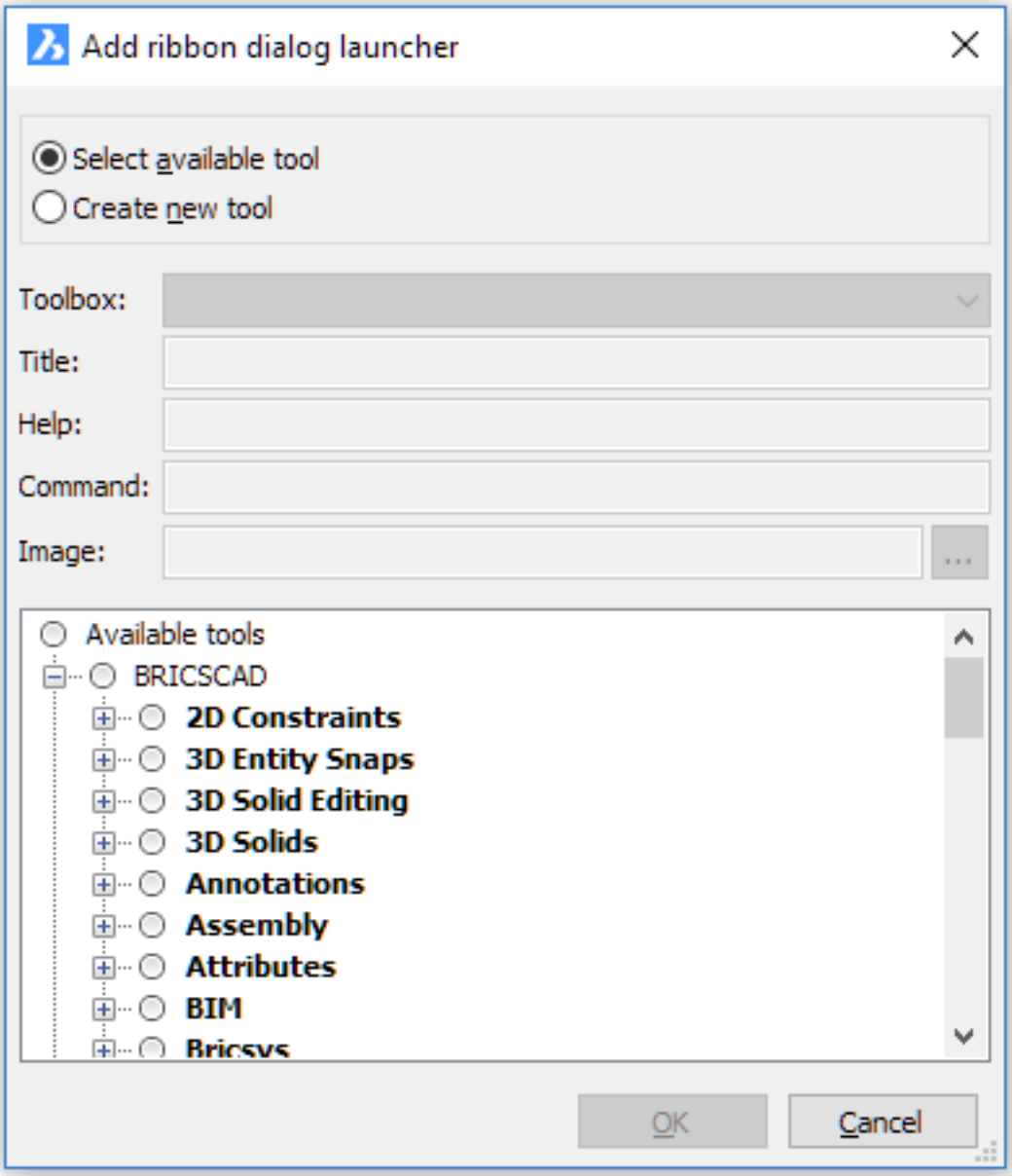
To add a row to a ribbon panel Do one of the following:

- Right click the ribbon panel and choose **Append Row** in the context menu. The row is added at the bottom of the ribbon panel tree.
- Right click an existing row and choose **Insert Ribbon Row** in the context menu. The row is added above the existing row.
- **Optional:** Drag the row to a different location in the ribbon panel tree.

To add a launcher to a ribbon panel

1. Right click the ribbon panel and choose **Add launcher** in the context menu.
2. Either select an available tool or Create a New Tool (1.13.5), then click the **OK** button. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/AddRibbonD

Figure 1.335


Add ribbon dialog launcher
Select available tool
Create new tool
Toolbox:
Title:
Help:
Command:
Image:
Available tools
BRICSCAD
2D Constraints
3D Entity Snaps
3D Solid Editing
3D Solids
Annotations
Assembly
Attributes
BIM
Bricsys
OK
Cancel

Editing a panel row

1. Open the Customize dialog (1.13.3).
2. Click the **Ribbon** tab on the **Customize** dialog.
3. Expand the **Ribbon panels** group.
4. Expand the ribbon panel.
5. Expand the ribbon rows and ribbon row panels.

To add a row panel to a row Do one of the following:

- Right click the row and choose **Append Row Panel** in the context menu. The row panel is added at the bottom of the row tree.
- Right click an existing row panel and choose **Insert Row Panel** in the context menu. The row panel is inserted above the existing row panel.

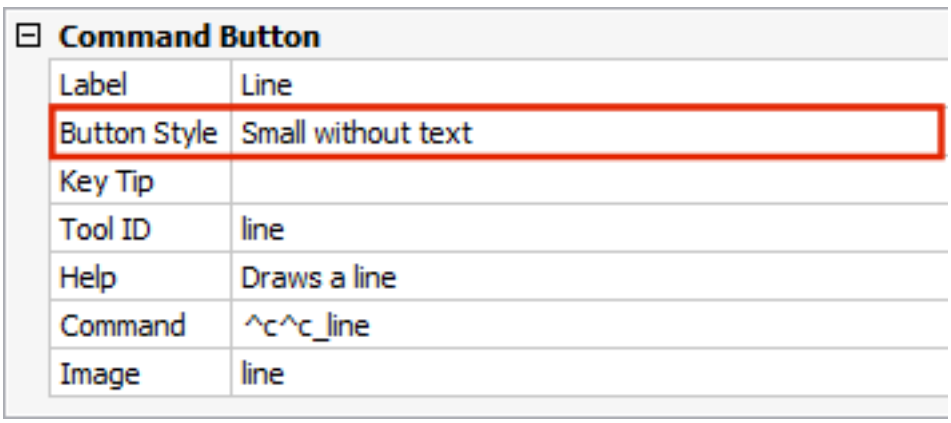
To add a command button to a row

1. Drag and drop a tool from the **Available Tools** list onto the row. Drop the tool on the row to add a command button at the bottom of the row tree. Drop the tool on an existing command button to add a command button above an existing command button.
2. Set the **Button Style** in the **Command Button** grid. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/
The options are:

Icon	Style
	Small
Polyline	Small with text
	Large
Spline	Large with text

3. **Optional:** Drag the command button to a different position in the same row or onto a different row.

Figure 1.336



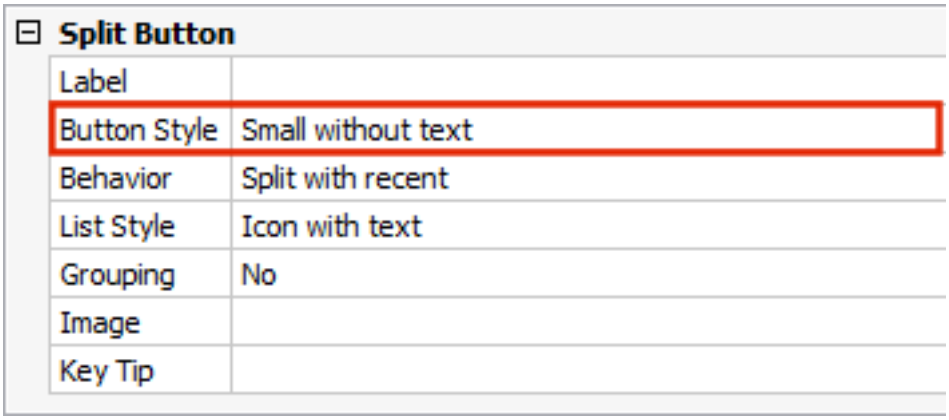
To add a split button to a row

1. Select the row, then right click and choose **Append split button** in the context menu. The split button is added at the bottom of the row tree.
2. Drag a series of related tools from the **Available Tools** list onto the split button.
3. Set the **Button Style** in the **Split Button** grid. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/
The options are:

Icon	Style
	Small
Polyline	Small with text
	Large
Spline	Large with text

1. Type a label text in the **Label** field.
2. **Optional:** Set the **Behavior** to **Split with recent (static text)** : always display the split button label, instead of the label of the most recently used tool.
3. **Optional:**Drag the split button to a different position in the same row or onto a different row.
4. Drag and drop tools from the **Available Tools** list onto the split button.

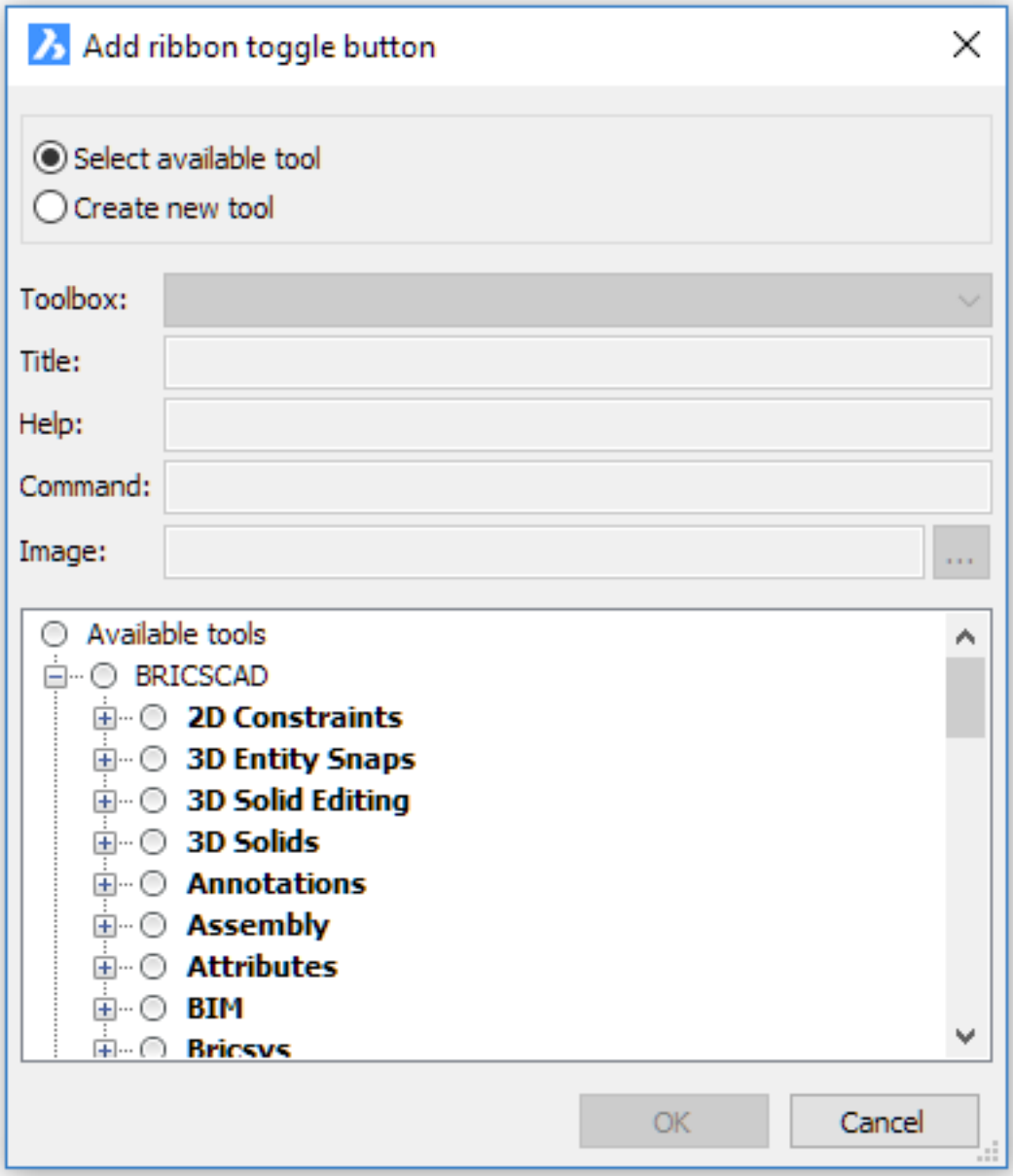
Figure 1.337



To add a toggle button to a row

1. Select the row, then right click and choose **Append Toggle Button** in the context menu.
2. Either select an available tool or Create a New Tool (1.13.5), then click the **OK** button. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/AddRibbonT
3. The toggle button is added at the bottom of the row tree.
4. **Optional:**Drag the toggle button to a different position in the same row or onto a different row.

Figure 1.338


Add ribbon toggle button
Select available tool
Create new tool
Toolbox:
Title:
Help:
Command:
Image:
...
Available tools
BRICSCAD
2D Constraints
3D Entity Snaps
3D Solid Editing
3D Solids
Annotations
Assembly
Attributes
BIM
Bricsys
OK
Cancel

To add a control to a row

1. Drag a control from the controls list onto the row.
2. Drop the control on the row to add it at the bottom of the row tree.
3. Drop the control on a existing command button to add it above the command button.

Figure 1.339

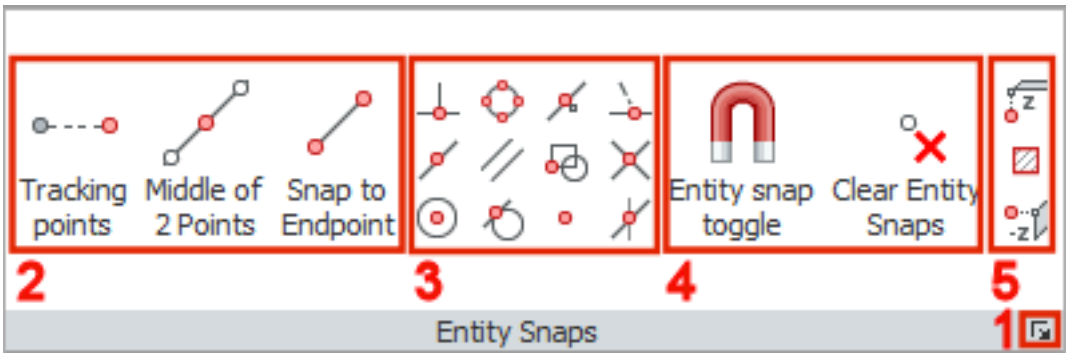
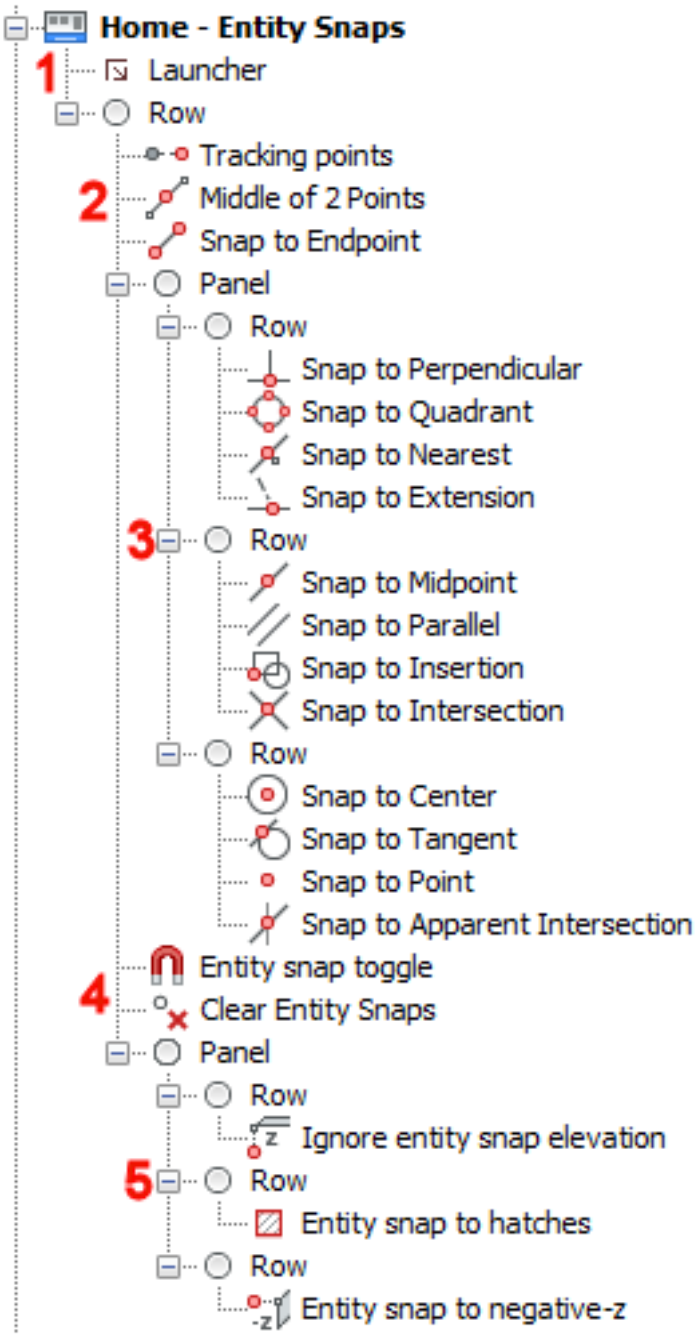


Figure 1.340


Home - Entity Snaps
1
Launcher
Row
Tracking points
2
Middle of 2 Points
Snap to Endpoint
Panel
Row
Snap to Perpendicular
Snap to Quadrant
Snap to Nearest
Snap to Extension
3
Row
Snap to Midpoint
Snap to Parallel
Snap to Insertion
Snap to Intersection
Row
Snap to Center
Snap to Tangent
Snap to Point
Snap to Apparent Intersection
Entity snap toggle
4
Clear Entity Snaps
Panel
Row
Ignore entity snap elevation
5
Row
Entity snap to hatches
Row
Entity snap to negative-z

1.13.7 Tool Palettes

Command: TOOLPALETTES (6.29.19), TOOLPALETTESCLOSE (6.29.20), TPNAVIGATE (6.29.23) The **ToolPalettes** command opens the Tool Palettes bar. The **ToolPalettesClose** command closes the Tool Palettes bar. The **TpNavigate** command displays a tool palette or a tool palette group. On a tool palette you can add blocks, hatches and commands in a tabbed window. Drag a tool from the palette to the graphic screen to launch the related tool.

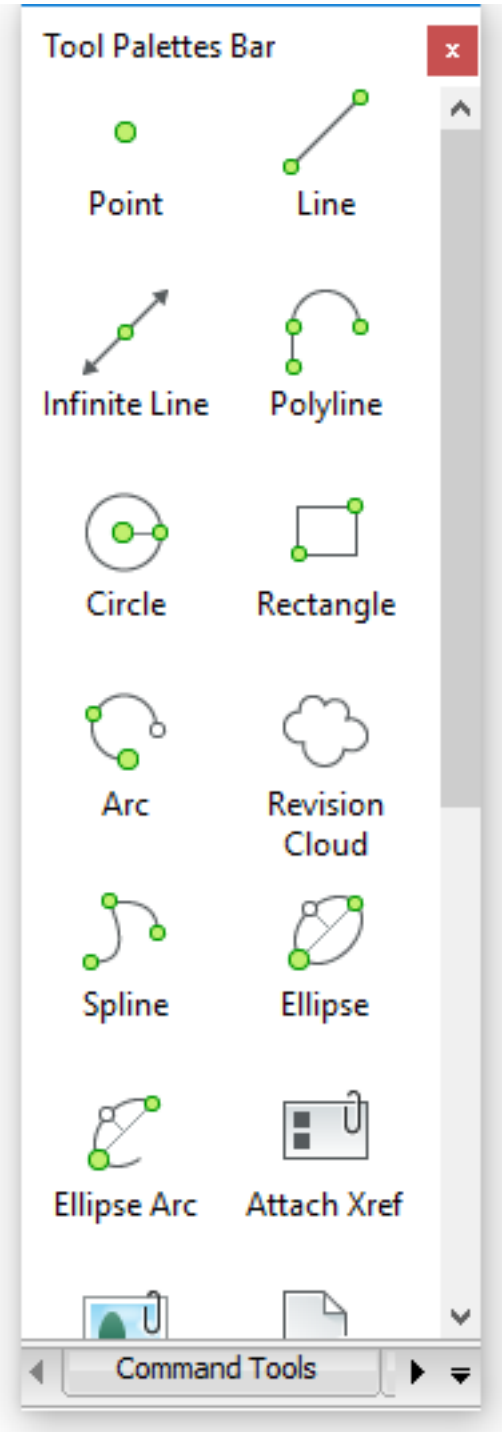
To open the Tool Palettes Bar

Do one of the following:

- Hover over a toolbar (Windows) or an empty toolbar area (Windows and Linux), then right click and choose **Tool Palettes Bar** in the context menu.
- Type **toolpalettes** in the command bar, then press Enter.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/05T

Figure 1.341

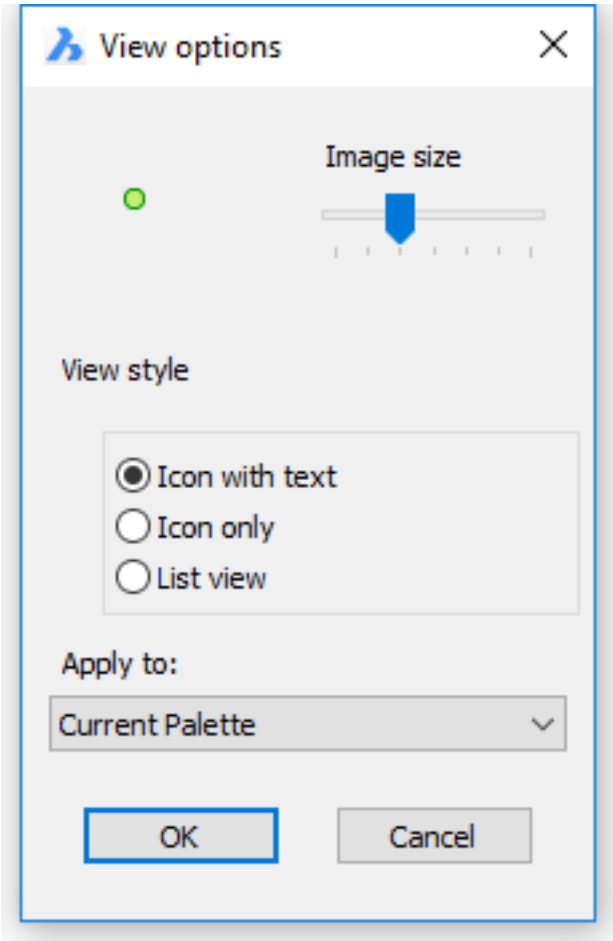

Tool Palettes Bar
Point
Line
Infinite Line
Polyline
Circle
Rectangle
Arc
Revision Cloud
Spline
Ellipse
Ellipse Arc
Attach Xref
Command Tools

To set the view options

1. Right click in an empty space of the icon area on the **Toolpalettes Bar** and choose **View Options...** in the context menu. The **View Options** dialog displays. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enU

2. Drag the **Image Size** slider to adjust the image size.
3. Select either **Icon with text**, **Icon only** or **List view.**
4. Choose either **Current Palette** or **All Palettes** from the **Apply to** list button.
5. Click the **OK** button to confirm.

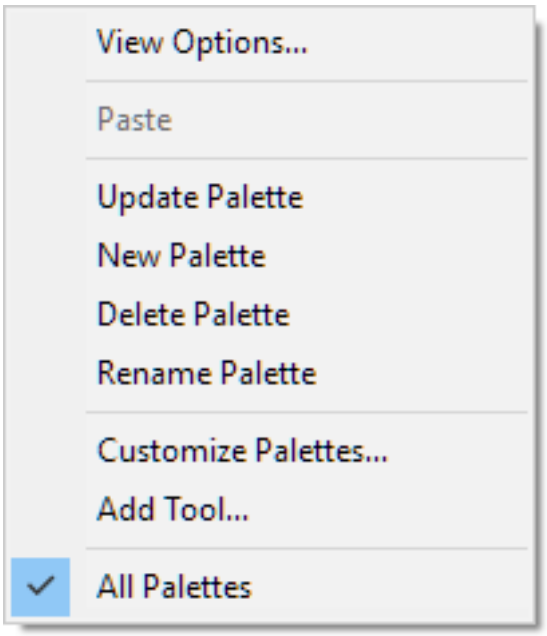
Figure 1.342



To add a new tool palette

1. Move the cursor to the icon area of the **Tool Palettes Bar**, then right click. A context menu displays: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/A
2. Choose **New Palette** in the context menu. A new tab is added.
3. Type a name for the new palette in the **Name** field.

Figure 1.343



To open the customize dialog box

1. Move the cursor to the icon area of the **Tool Palettes Bar**, then right click. A context menu displays: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/

2. Choose **Customize Palettes** in the context menu. The **Customize** dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/05Customiz

Figure 1.344

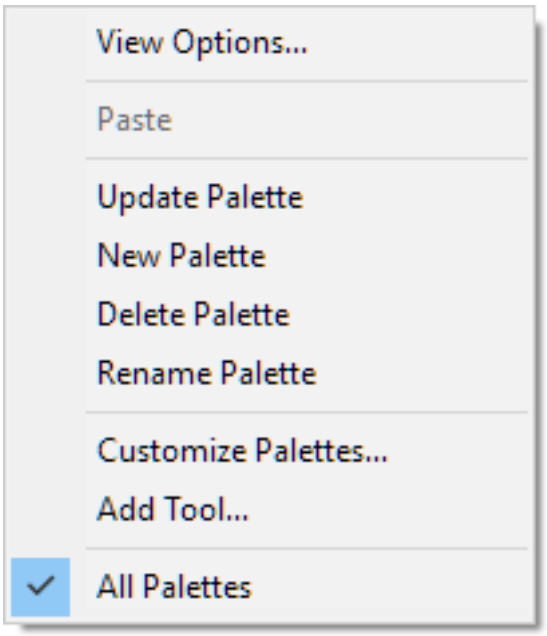
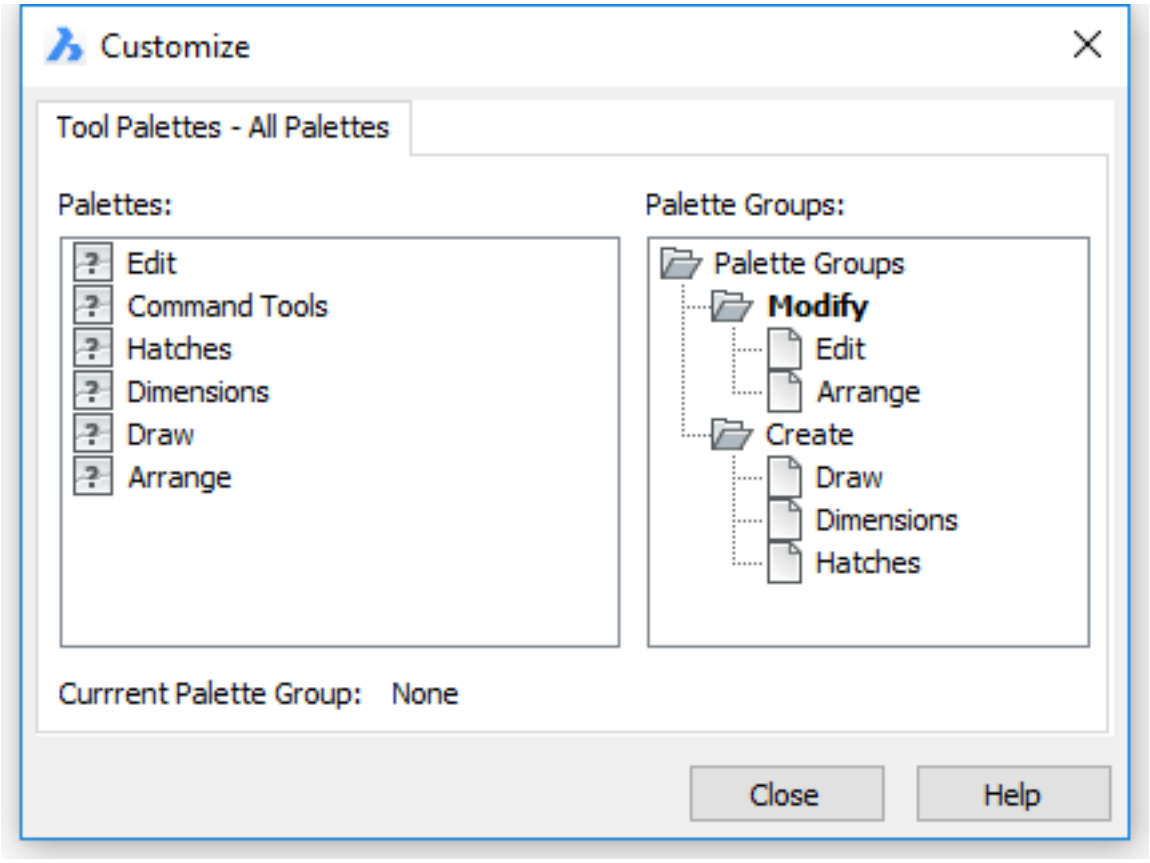


Figure 1.345



To import a tool palette

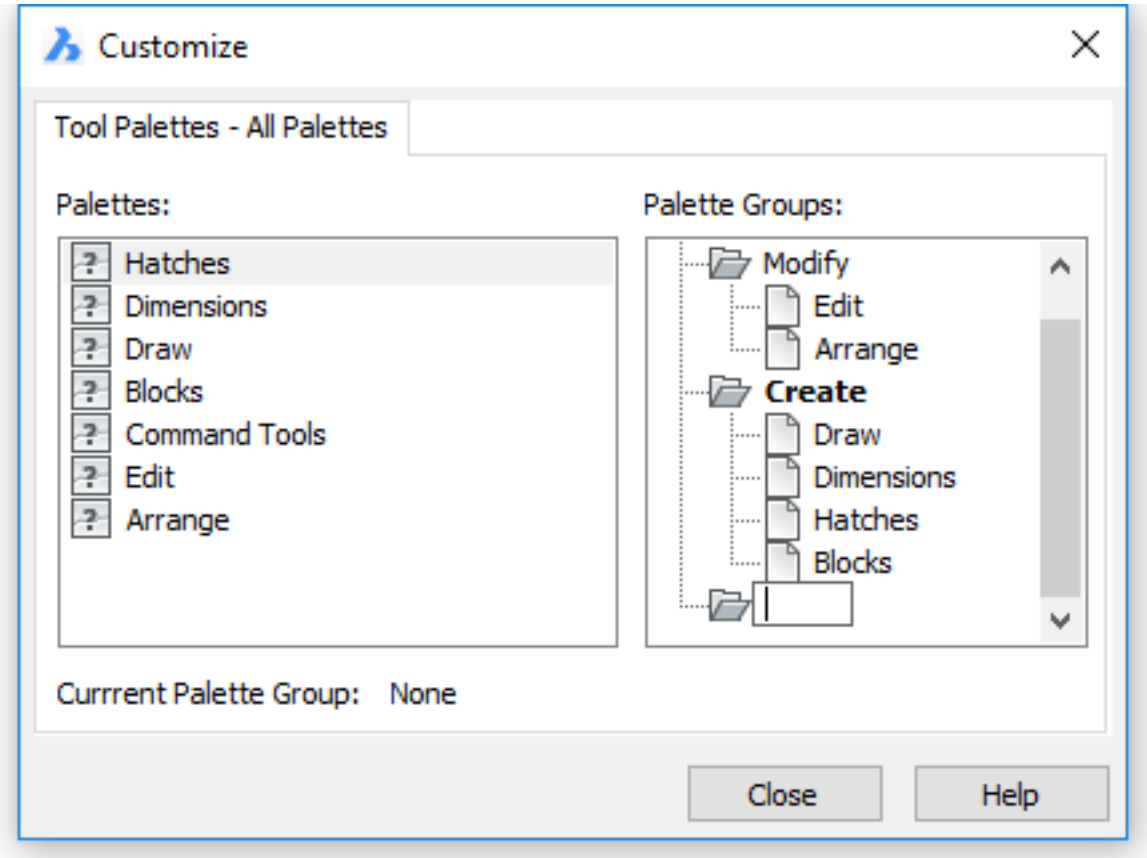
1. Open the Customize dialog box.
2. In the **Palettes** pane on the **Customize** dialog box, right click and choose **Import** in the context menu.
3. On the **Import Palette** dialog, select a folder, then select a palette file (*.btc or *.xtp).
4. Click the **Open** button or double click to import the palette. The palette is added in the **Palettes** list.
5. Click the **Close** button to close the **Customize** dialog box.

Working with palette groups

To create a palette group

1. Open the Customize dialog box.
2. Place the cursor on the **Palette Groups** root folder name.
3. Right click and choose **New Palette** in the context menu. A new group folder is added at the bottom of the palette groups tree. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02
4. Type a name for the new group.

Figure 1.346



To create a palette subgroup

1. Open the Customize dialog box.
2. Place the cursor on an existing group name.
3. Right click and choose **New Palette** in the context menu. A new group folder is added as a subgroup of the selected group.
4. Type a name for the new group.

To add a palette to a group

1. Open the Customize dialog box.
2. Place the cursor on a palette.
3. Press and hold the left mouse button, then drag the palette onto a palette group name.
4. Release the left mouse button to add the palette to the group.

To remove a palette from a group

1. Open the Customize dialog box.
2. Select the palette in the group.
3. Right click and choose **Remove** in the context menu.

To set a palette group current

1. Open the Customize dialog box.
2. Select the palette group in the palette groups tree.
3. Right click and choose **Set Current** in the context menu.

To delete a palette group

1. Open the Customize dialog box.
2. Select the palette group in the palette groups tree.
3. Right click and choose **Delete** in the context menu.

To rename a tool palette

1. Right click tool palette tab and choose **Rename Palette** in the context menu.
2. Type a new name for the palette in the **Name** field.
3. Press Enter to confirm.

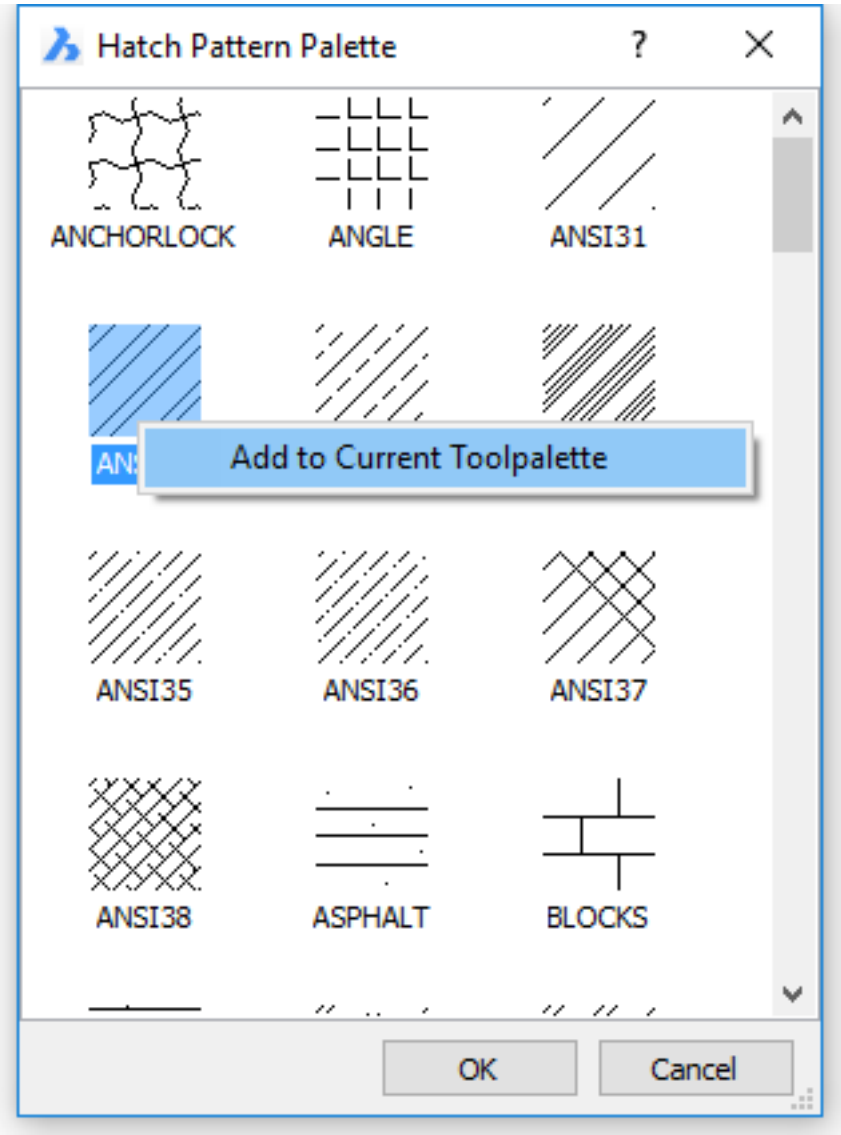
To delete a tool palette

1. Right click tool palette tab and choose **Delete Palette** in the context menu.
2. Click the **OK** button on the **Confirm Palette Deletion** box.

Adding a hatch pattern

1. Select the palette you want to add a hatch pattern to.
2. Open the Hatch and Gradient (1.17.28) dialog box.
3. Select either **Predefined** or **Custom** from the **Type** list button.
4. Double click the **Swatch** field on the **Hatch and Gradient** dialog box. The **Hatch Pattern Palette** displays.
5. Select the hatch pattern you want to add, then right click and Choose **Add to Current Tool Palette**. The hatch pattern is added on the current tool palette. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/
6. (option) Repeat steps 5 and 6 to add more hatch patterns.
7. (option) Drag-and-drop patterns to change their order on the palette.

Figure 1.347


Hatch Pattern Palette
?
ANCHORLOCK
ANGLE
ANSI31
Add to Current Toolpalette
ANSI35
ANSI36
ANSI37
ANSI38
ASPHALT
BLOCKS
OK
Cancel

Adding blocks

Blocks can be added from the current drawing or from any drawing.

Adding blocks from the current drawing:

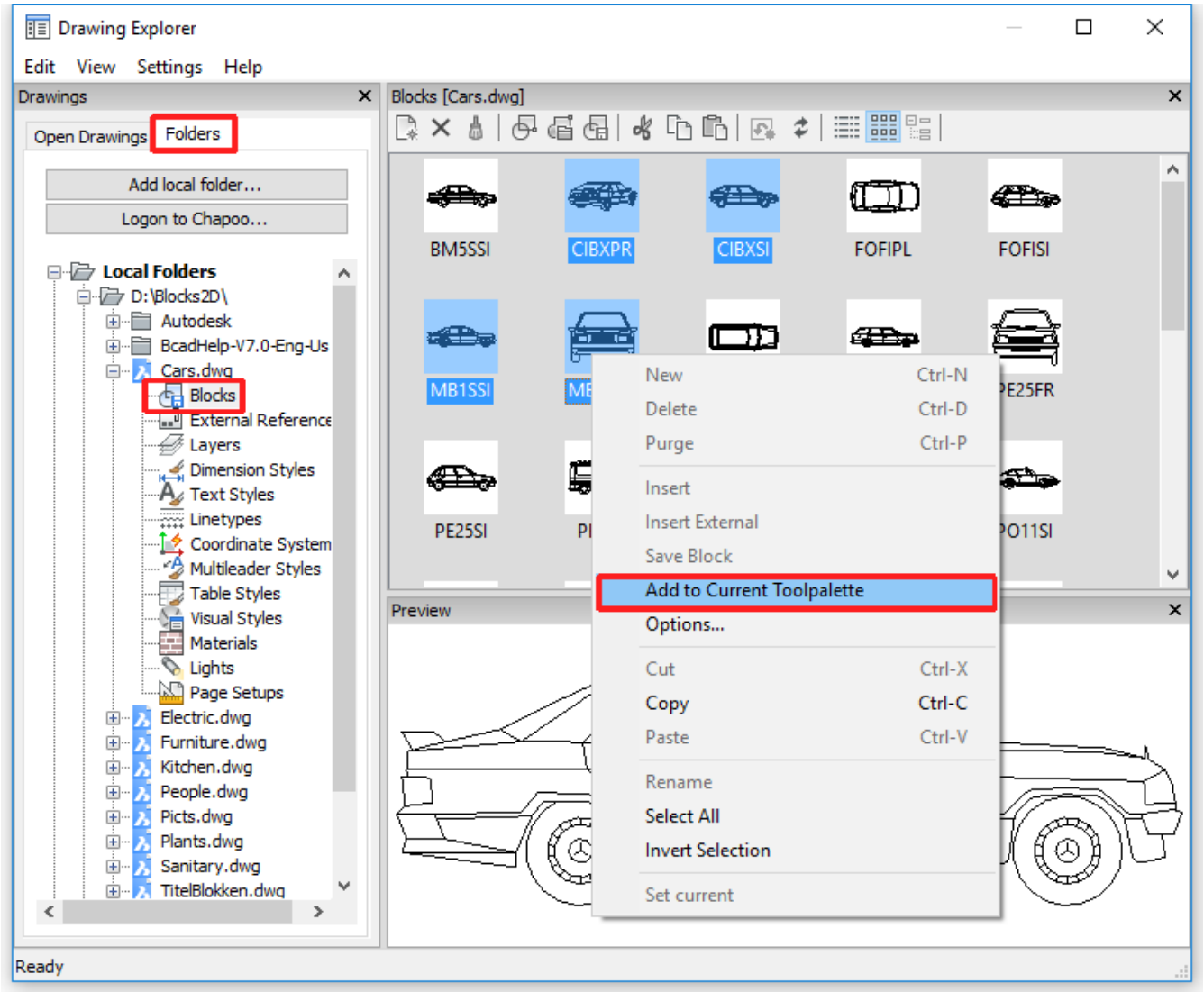
1. Choose **Drawing Explorer - Blocks** in the **Tools** menu.
2. On the **Drawing Explorer - Blocks** dialog box select the block or blocks you want to add.
3. Right click and choose **Add to Current Tool Palette** in the context menu.

Adding blocks from any drawing:

1. If not already available, add the containing folder to the Folders (6.13.14) list on the **Drawing Explorer - Drawings** dialog box.
2. Select the drawing in the containing folder on the **Drawing Explorer - Drawings** dialog box.
3. Click the **Expand** icon (+) of the containing drawing. The **Blocks** icon of the drawing displays.
4. Click the **Blocks** icon of the containing drawing. The blocks in the drawing display in the **Details** pane.
5. Select the block or blocks you want to add. Press and hold the Ctrl key to select multiple blocks.
6. Right click and choose **Add to Current Tool Palette** in the context menu.
7. (option) Drag-and-drop blocks to change their order on the palette.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/

Figure 1.348

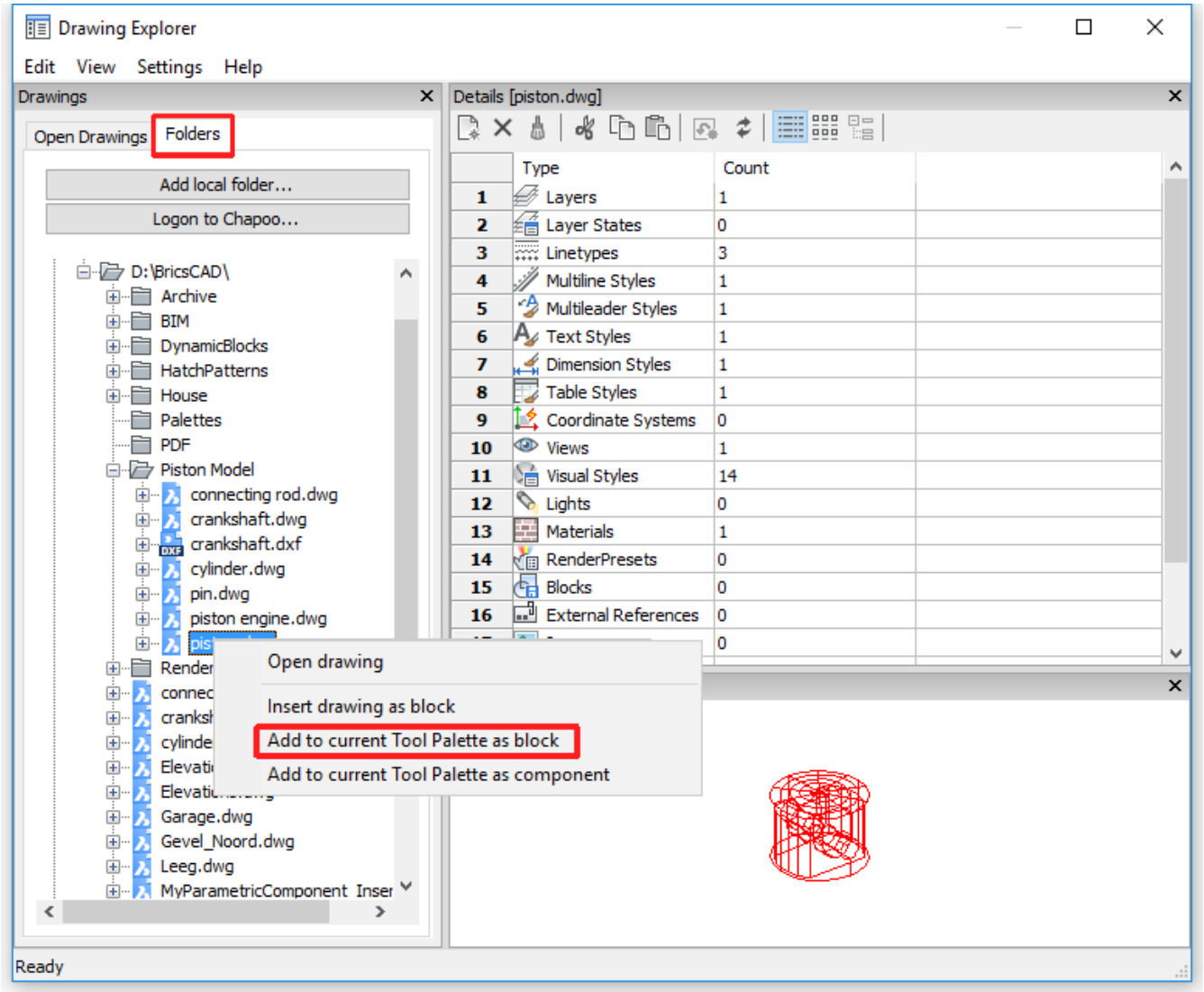

Drawing Explorer
Edit View Settings Help
Drawings
Open Drawings
Folders
Add local folder...
Logon to Chapoo...
Local Folders
D:\Blocks2D\
Autodesk
BcadHelp-V7.0-Eng-Us
Cars.dwg
Blocks
External Reference
Layers
Dimension Styles
Text Styles
Linetypes
Coordinate System
Multileader Styles
Table Styles
Visual Styles
Materials
Lights
Page Setups
Electric.dwg
Furniture.dwg
Kitchen.dwg
People.dwg
Picts.dwg
Plants.dwg
Sanitary.dwg
TitelBlokken.dwg
Blocks [Cars.dwg]
BM5SSI
CIBXPR
CIBXSI
FOFIPL
FOFISI
MB1SSI
PE25SI
New Ctrl-N
Delete Ctrl-D
Purge Ctrl-P
Insert
Insert External
Save Block
Add to Current Toolpalette
Options...
Cut Ctrl-X
Copy Ctrl-C
Paste Ctrl-V
Rename
Select All
Invert Selection
Set current
Preview
Ready

Adding a drawing as a block:

1. If not already available, add the containing folder to the Folders (6.13.14) list on the **Drawing Explorer - Drawings** dialog box.
2. Select the drawing in the containing folder on the **Drawing Explorer - Drawings** dialog box.
3. Right click and choose **Add to Current Toolpalette as block** in the context menu.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/
When placing a block from a toolpalette the -Insert (6.17.14) command is launched.

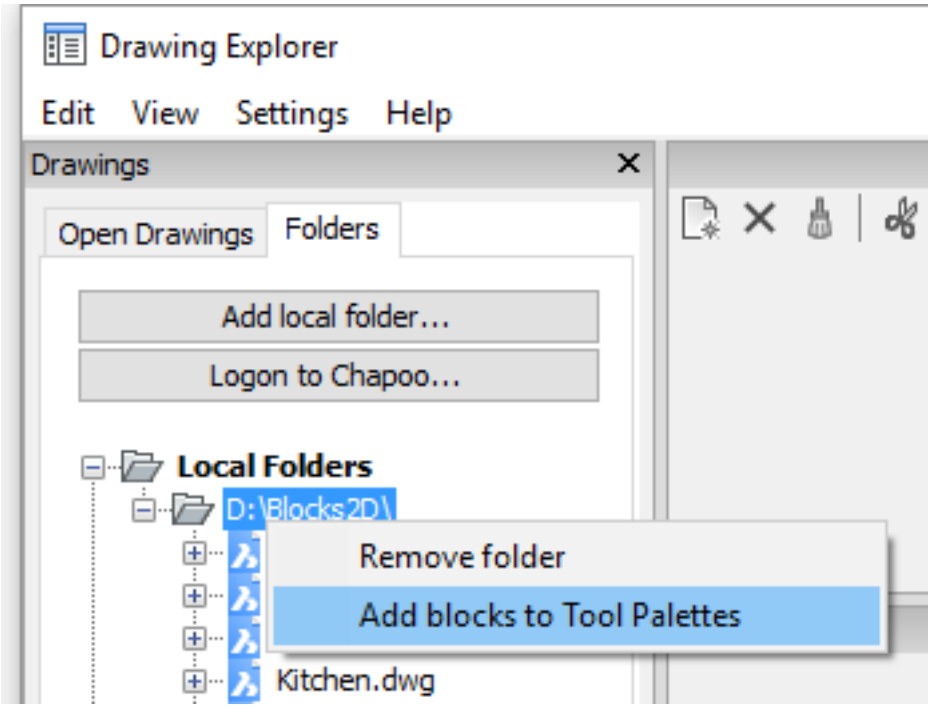
Figure 1.349


Drawing Explorer
Edit View Settings Help
Drawings
Open Drawings
Folders
Add local folder...
Logon to Chapoo...
D:\BricsCAD\
Archive
BIM
DynamicBlocks
HatchPatterns
House
Palettes
PDF
Piston Model
connecting rod.dwg
crankshaft.dwg
crankshaft.dxf
cylinder.dwg
pin.dwg
piston engine.dwg
Open drawing
Insert drawing as block
Add to current Tool Palette as block
Add to current Tool Palette as component
Garage.dwg
Gevel_Noord.dwg
Leeg.dwg
Details [piston.dwg]
Type
Count
Layers 1
Layer States 0
Linetypes 3
Multiline Styles 1
Multileader Styles 1
Text Styles 1
Dimension Styles 1
Table Styles 1
Coordinate Systems 0
Views 1
Visual Styles 14
Lights 0
Materials 1
RenderPresets 0
Blocks 0
External References 0
Ready

Adding all blocks from multiple drawings. This procedure collects all blocks in each of the drawings of the selected folder and add them to a palette of the drawing name. New palettes are created if necessary.

1. If not already available, add the containing folder to the Folders (6.13.14) list on the **Drawing Explorer - Drawings** dialog box.
2. Select the folder, then right click and choose **Add blocks to Tool Palettes** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/05AddBlock

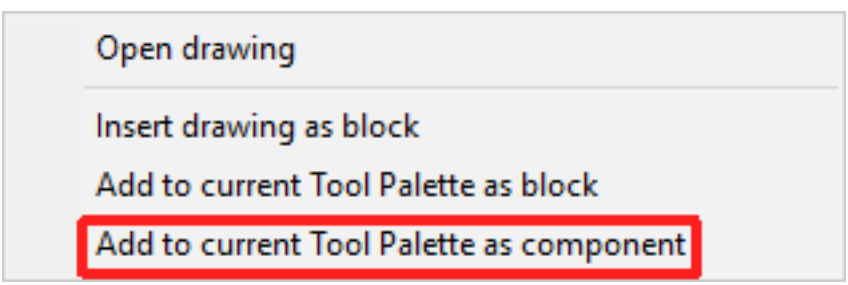
Figure 1.350



To add a drawing as a component.

1. If not already available, add the containing folder to the Folders (6.13.14) list on the **Drawing Explorer - Drawings** dialog box.
2. Select the drawing in the containing folder on the **Drawing Explorer - Drawings** dialog box.
3. Right click and choose **Add to Current Toolpalette as component** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/05FoldersDr
When placing a component from the tool palette, the **-BmInsert (6.8.12)** command is launched.

Figure 1.351

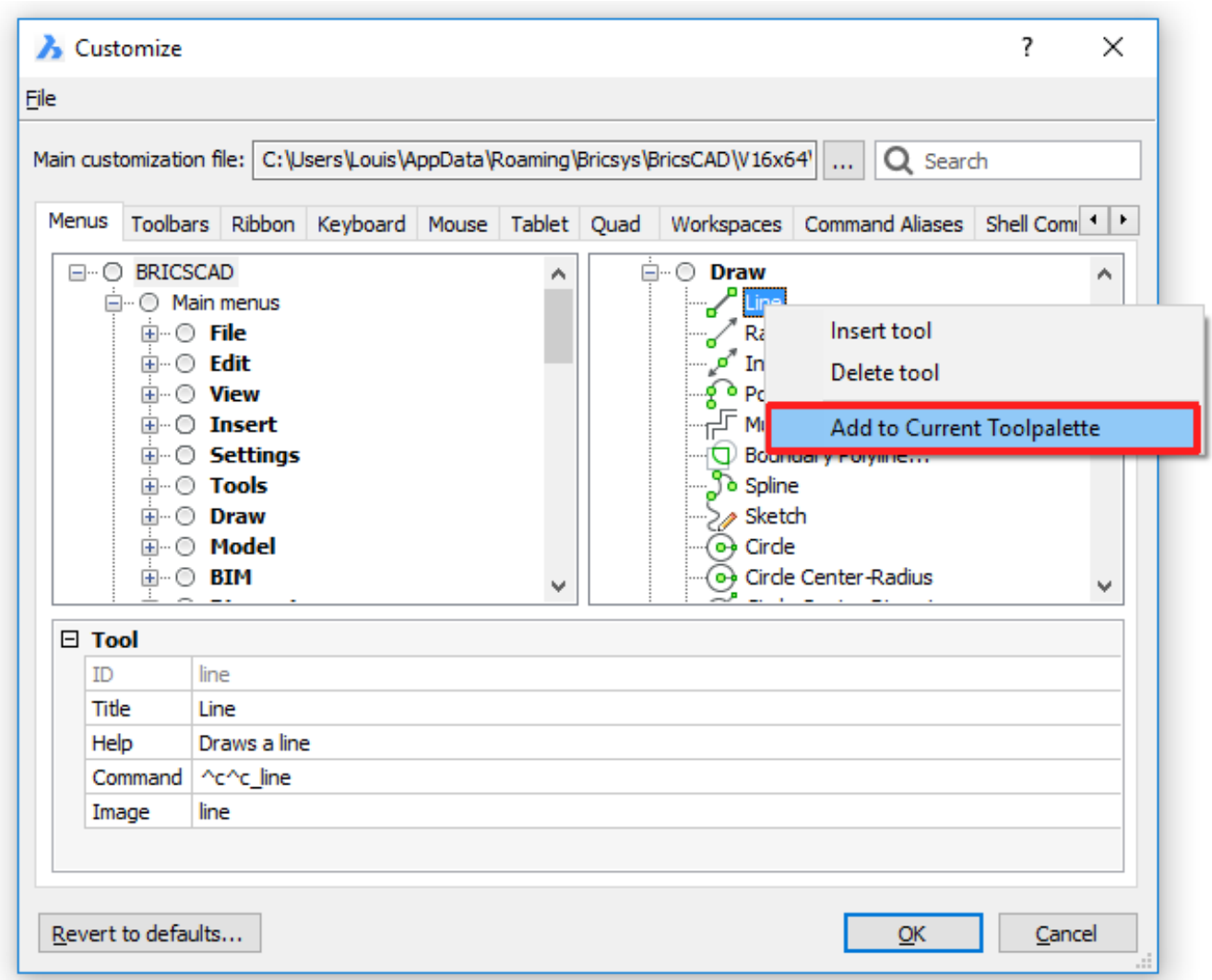


Adding a command

1. Choose **Customize** in the **Tools** menu.
2. Under **Available Tools** on the **Customize** dialog, expand the containing toolbox, then select the command in the commands list.
3. Right click and choose **Add to Current Tool Palette** in the context menu.
4. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourprefere
5. (option) Drag-and-drop commands to change their order on the palette.

NOTE	See the topic Creating Custom Tools (1.13.5) to create your own tools, which you can then add to a tool palette.

Figure 1.352


Customize
File
Main customization file:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V16x64\
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Workspaces
Command Aliases
BRICSCAD
Main menus
File
Edit
View
Insert
Settings
Tools
Draw
Model
BIM
Draw
Insert tool
Delete tool
Add to Current Toolpalette
Spline
Sketch
Circle
Circle Center-Radius
Tool
ID
line
Title
Line
Help
Draws a line
Command
^c^c_line
Image
line
Revert to defaults...
OK
Cancel

Editing a tool

General procedure to edit a tool on a tool palette:

1. Right click the tool, then choose **Properties** in the context menu. The **Tool Properties** dialog box displays. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/02Definingyourprefere
Hatch tool properties (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/0
Block tool properties (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/0
Command tool properties
2. Edit the tool properties.
3. Click the **OK** button to confirm.

Figure 1.353

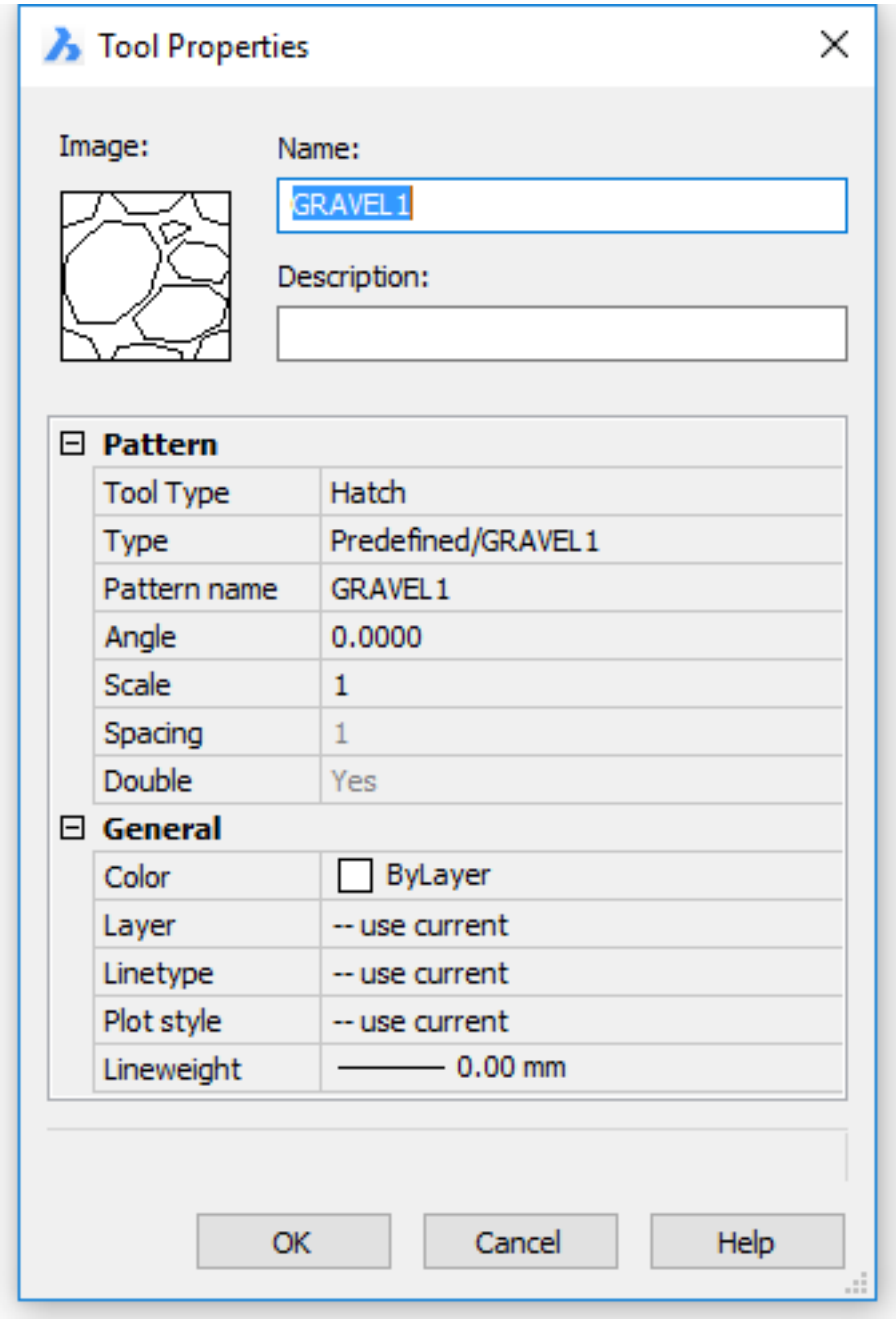

Tool Properties
Image:
Name:
GRAVEL1
Description:
Pattern
Tool Type
Hatch
Type
Predefined/GRAVEL1
Pattern name
GRAVEL1
Angle
0.0000
Scale
1
Spacing
1
Double
Yes
General
Color
ByLayer
Layer
-- use current
Linetype
-- use current
Plot style
-- use current
Lineweight
0.00 mm
OK
Cancel
Help

Figure 1.354

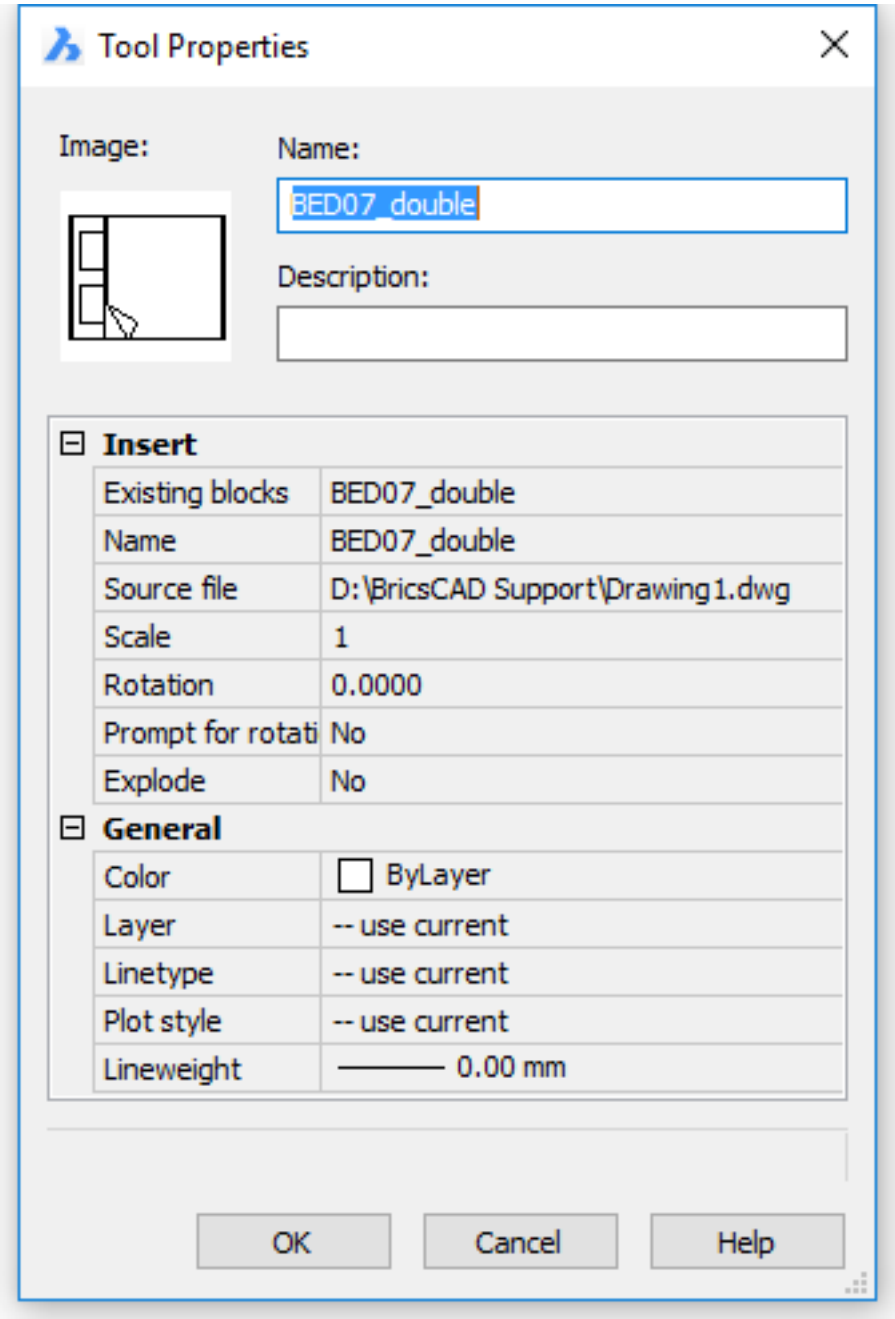
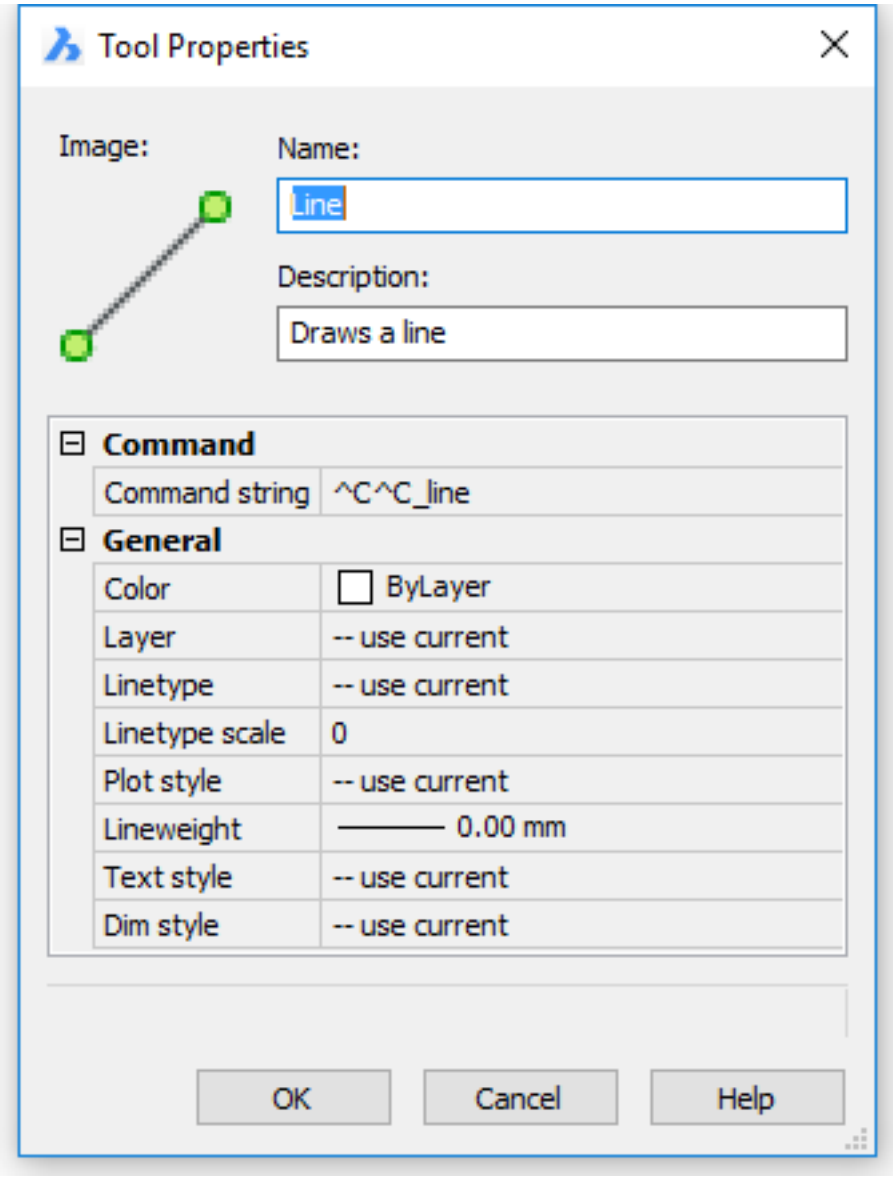

Tool Properties
Image:
Name:
BED07_double
Description:
Insert
Existing blocks
BED07_double
Name
BED07_double
Source file
D:\BricsCAD Support\Drawing1.dwg
Scale
1
Rotation
0.0000
Prompt for rotati
No
Explode
No
General
Color
ByLayer
Layer
-- use current
Linetype
-- use current
Plot style
-- use current
Lineweight
0.00 mm
OK
Cancel
Help

Figure 1.355


Tool Properties
Image:
Name:
Line
Description:
Draws a line
Command
Command string
^C^C_line
General
Color
ByLayer
Layer
-- use current
Linetype
-- use current
Linetype scale
0
Plot style
-- use current
Lineweight
0.00 mm
Text style
-- use current
Dim style
-- use current
OK
Cancel
Help

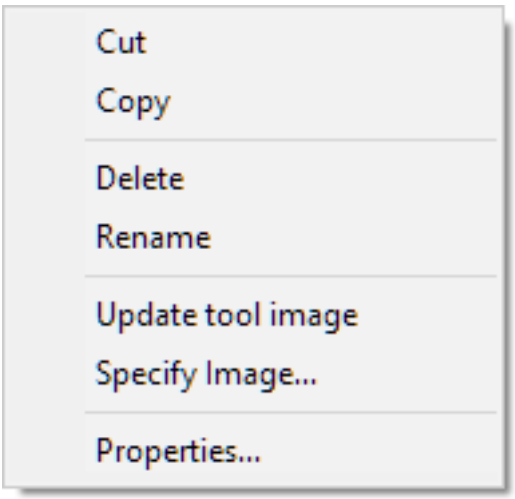
Managing tools

To manage the tools on a tool palette do the following:

1. Right click the tool. A context menu displays: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS

2. Choose an option in the context menu.

Figure 1.356



To specify a tool image

1. Right click the tool and choose **Specify Image** in the context menu. A file dialog box displays.
2. Select the image file, then click the **Open** button.

NOTE	To restore the original tool image: Right click the tool and choose **Remove Image** in the context menu.

To move a tool Drag-and-drop tools to re-arrange tools on a palette: drag the tool to the desired location and drop the tool on top of the existing tool. To move a tool from one palette to another, do the following:

1. Right click the tool on the source palette and choose **Cut** in the context menu.
2. Select the target palette.
3. Right click and choose **Paste** in the context menu. The tool is added at the bottom of the palette.

To copy a tool To copy a tool from one palette to another, do the following:

1. Right click the tool on the source palette and choose **Copy** in the context menu.
2. Select the target palette.
3. Right click and choose **Paste** in the context menu. The tool is added at the bottom of the palette.

Tutorial: Customizing Commands in the Tool Palette Click here (1.3.13) to watch

1.13.8 Command Aliases

Command aliases are alternative names for BricsCAD commands. You can launch the LINE command by entering L at the command prompt. There is no need to type the complete name. After typing the command alias at the command prompt the command is executed. Some commands have more than one alias: e.g. REC, RECT and RECTANGLE are all aliases for Rectang (6.26.6), whereas the last alias is even longer than the command name. In general, command aliases can increase your speed in entering the commands at keyboard. Command aliases are saved in a *.pgp file (P roG ram P arameters file). By default the **default.pgp** file in the **Support** folder of the Roamable Root folder (1.13.2) is used. If a command can be executed transparently (= while another command is being executed), the alias can also be used transparently. To execute a command or an alias transparently, precede the command name or alias with an apostrophe (').

NOTES

- Unlike commands, aliases are only local and cannot be used as global (preceded by a underscore (_).
- Command aliases cannot be used in scripts (*.scr).

To edit the alias file in a text editor

In order to modify existing aliases or to create new ones, the **default.pgp** file can be edited using an ASCII text editor (e.g. Notepad). The first section of the **default.pgp** file defines the **Shell** commands. The second section of the **default.pgp** file defines command aliases. The following syntax is used to define a command alias: **alias, *command alias** = the character string to be entered at the command prompt. The command alias must be followed by a comma (,). **command** = the BricsCAD command. The command name must be preceded by an asterisk (*). Comment lines must be preceded by a semicolon (;). Comment lines can be used to add additional information, e.g. when the file was last edited and by whom. Some commands, such as Hatch (6.16.1) or Boundary (6.6.15), can be preceded by a hyphen (-) to access the command line version of the command instead of the dialog box version. Command aliases can be created for these commands also, e.g.: **vi, *-view** Execute the REINIT (6.26.19) command to reload the alias file when it was externally edited.

Using the Customize dialog to edit the alias file

1. Open the Customize dialog (1.13.3).
2. Click the **Aliases** tab on the **Customize** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/U
3. **Optional** To create a new alias:
   - Click the **Add** button. The **Add Alias** dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel
   - Click the down arrow at the right hand side of the **Command** field, then select a command. You can start typing the command name in the **Command** field to easily find the command in the list.
   - Type the command alias for the command in the **Alias** field.
   - Click the **OK** button.
4. **Optional** To edit an existing alias:
   - Select the alias in the list.
   - Click the **Edit** button. The **Edit Alias** dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel
   - Type a new alias in the Alias field. If the alias already exists you are prompted to overwrite the existing alias.
   - Click the **OK** button.
5. **Optional** To delete an alias:
   - Select the alias in the list.
   - Click the **Delete** button. A **Confirm** box displays.
   - Click the **OK** button on the Confirm box to delete the alias.

Figure 1.357

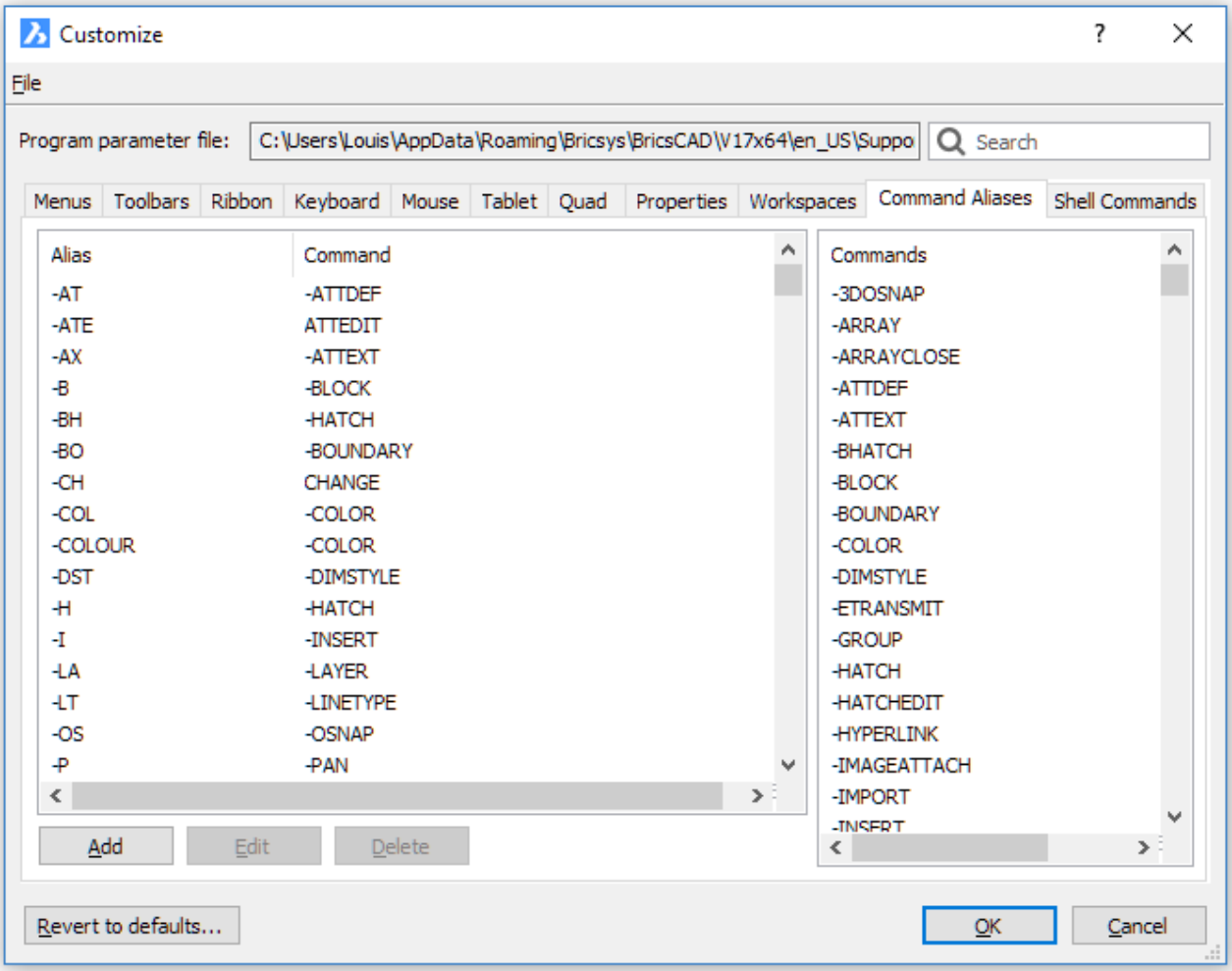


Figure 1.358

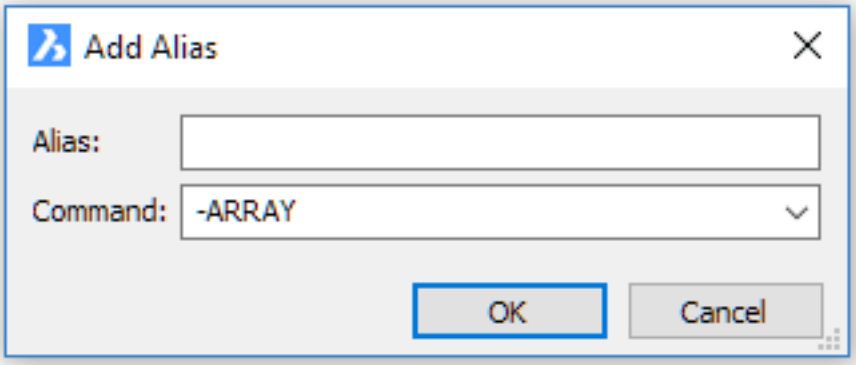
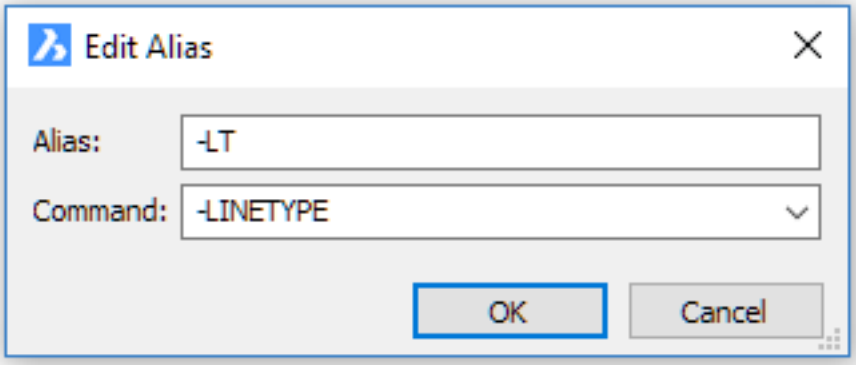


Figure 1.359



6. Click the **OK** button on the **Customize** dialog to stop.

NOTE	If you use the **Customize** dialog to edit the alias file, it is not necessary to execute the ReInit (6.26.19) command.

Tutorial: Abbreviating Command Input with Aliases Click here (1.3.21) to watch

1.13.9 User Profile Manager

Command: ProfileManager (6.24.47) In BricsCAD you can customize the settings that control your drawing environment, which can then be saved in a **user profile** . You can create multiple user profiles, e.g. a profile to run BricsCAD with a white background and a limited set of toolbars and another profile with no command bar and all toolbars and snap settings needed to draw comfortably in 3D. User profiles can also be used when multiple users share the same computer and each user likes to create his personal drawing environment. User profiles can be saved to a file, which can then be imported on a different computer. This allows you to copy your settings to a new computer or to install the same settings on multiple computers. If you use a third party applications on top of BricsCAD, you might want to create a user profile to use with each of these applications. The **User Profile Manager** is a standalone tool that lists the existing profiles and provides all the tools needed to Load, Create, Set Current, Copy, Rename, Delete, Export and Import user profiles.

Content of a user profile

User profiles save many settings that control the drawing environment. Once you start BricsCAD using a user profile, it automatically tracks and stores changes that you make to your drawing environment.

- Main Cui file.
- Current Workspace: BricsCAD has five standard workspaces: 2D Drafting, 3D Modeling, Mechanical, Sheet Metal and BIM.
- Configuration settings: all system variables which are saved in the Registry () and user preferences.
- Default plot settings.
- Settings in dialog windows, e.g. search options in the **Settings** dialog.
- Project (1.14.11) settings
- Recent paths
- Status bar (1.12.2) settings
- Properties Bar (1.12.4) settings
- Toolbar and menu configuration
- Worksets (6.33.14).

To Launch the User Profile Manager

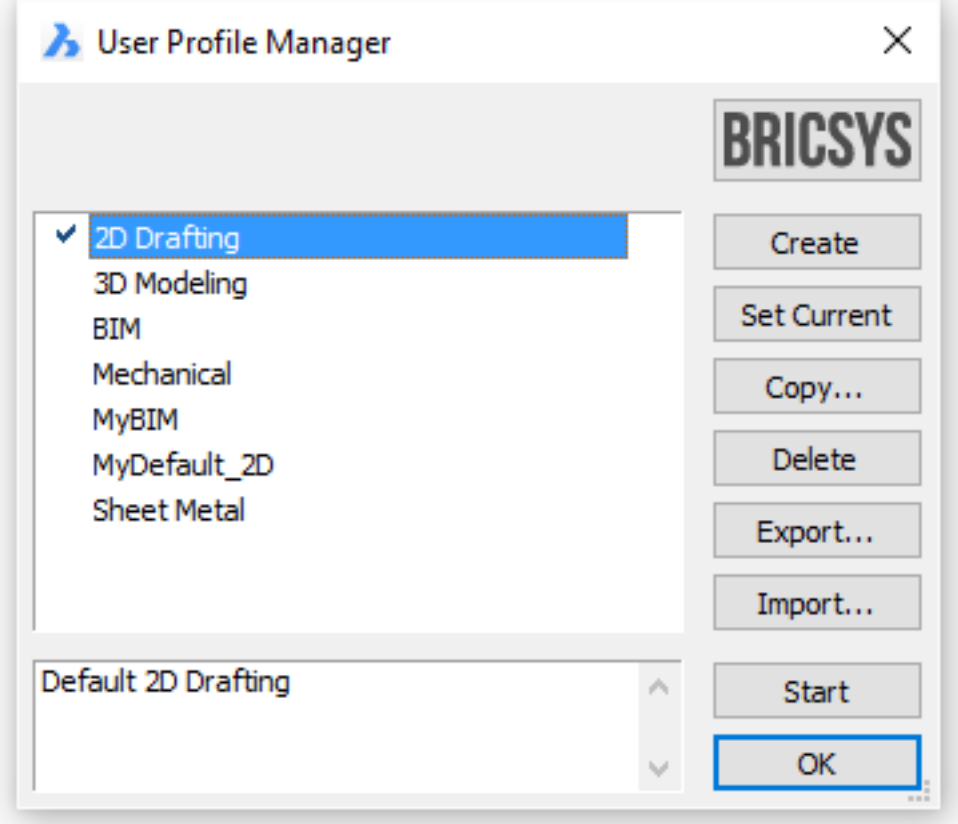
Do one of the following:

- When BricsCAD is running: choose **User Profile Manager** in the **Tools** menu.https://static.bricsys.com/
- From the **Windows Start menu** , choose **All Programs > Bricsys > BricsCAD > User Profile Manager**.
- Double click **userprofilemanager.exe** in the BricsCAD program folder (e.g. C:\Program Files\Bricsys\BricsCAD Vx). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/02Definingyourpreferences

In the **User Profile Manager** dialog:

- The **Current Profile** is marked.
- **Create:**Creates a new profile, using default settings.
- **Set Current:**Sets the selected profile as the current profile, which will be used the next time you start BricsCAD.
- **Copy:**Click to make a copy of the selected profile.
- **Delete:**Click to delete the selected profile.
- **Export...:**Click to save the selected profile to a file on your computer, disk, or network. The file is saved with an **.arg** extension.
- **Import...:**Click to open a profile that has been saved to **.arg** file).
- **Start:**Starts BricsCAD with the selected user profile.
- **OK:** Closes the **User Profile Manager** dialog
- Double click a profile in the list of the available profiles to start BricsCAD using this profile.

Figure 1.360



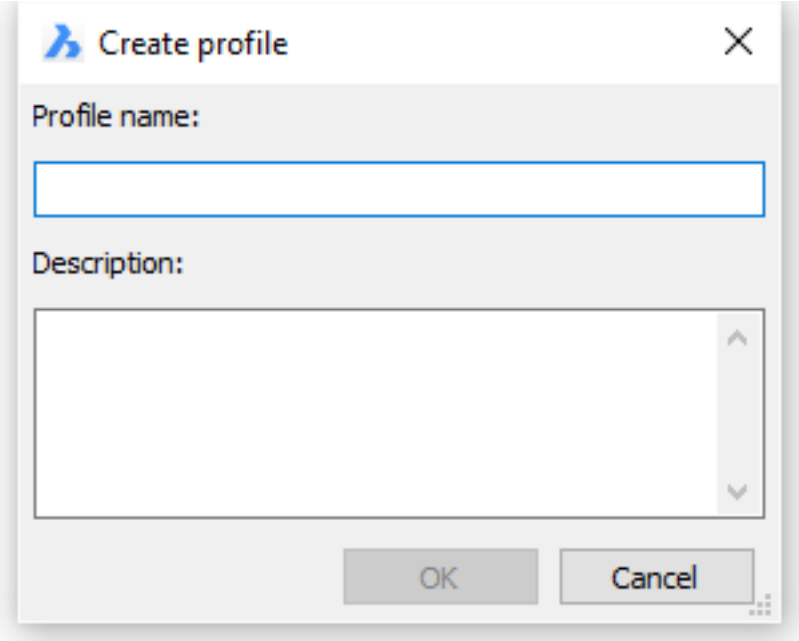
Creating user profiles

To create a new user profile:

1. Launch the User Profile Manager.
2. Click the **Create** button on the **User Profile Manager** dialog window. The **Create Profile** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/10C
3. Type a name for the new profile in the **Profile name** field.
4. **Optional**Type a description for the new profile in the **Description** field.
5. Click the **OK** button to create the profile. The new profile is added in the available profiles list. The new profile is created using default hard-coded values for all settings.

NOTE	Use the Copy (6.9.50) option to create a new user profile as a copy of an existing profile.

Figure 1.361



Starting BricsCAD with a specific user profile

When you start BricsCAD, the user profile that was current when you closed the previous BricsCAD session is used. If the GETSTARTED (1.13.11) system variable is ON, the **GET STARTED** page of the Welcome dialog box (1.14.1) displays on startup. Here you can select the user profile you want to use. To create a desktop shortcut to launch BricsCAD using a specific user profile, do one of the following:

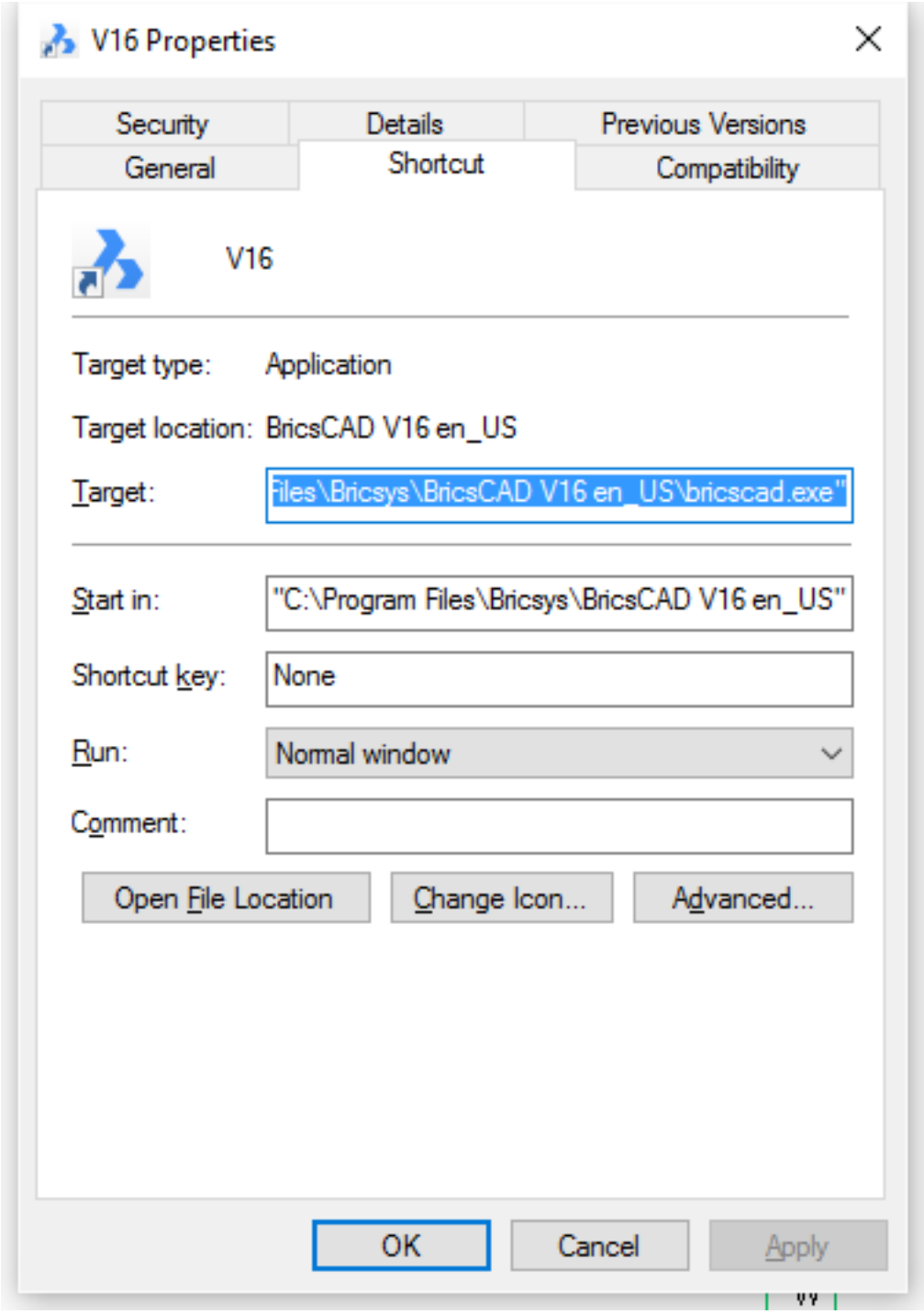
- Add the **/P option** to the **Target** settings of the BricsCAD shortcut. To open the shortcut settings dialog: right click the shortcut icon on the desktop and choose **Properties** in the context menu. A typical BricsCAD shortcut could be **"C:\Program Files\Bricsys\BricsCAD Vx\bricscad.exe" /P <UserProfileName>** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/10ShortcutProperties

- Launch the >User Profile Manager, then either double click a user profile or select a user profile and click the **Start** button.

NOTE

- The **/P** option is case insensitive.
- If you load a user profile when BricsCAD is already running a second instance of BricsCAD is launched, at least on condition that the **SINGLETON** variable is OFF. This variable controls whether multiple BricsCAD sessions can run simultaneously. See the Settings dialog (1.13.11) for more information about variables and settings.
- If the user profile in the argument of the **/P** option does not exist, the user profile is created automatically using default hard-coded values for all settings.
- If the user profile name contains spaces, use double quotes: e.g. /P "background white".

Figure 1.362


V16 Properties
Security
Details
Previous Versions
General
Shortcut
Compatibility
V16
Target type: Application
Target location: BricsCAD V16 en_US
Target: Files\Bricsys\BricsCAD V16 en_US\bricscad.exe"
Start in: "C:\Program Files\Bricsys\BricsCAD V16 en_US"
Shortcut key: None
Run: Normal window
Comment:
Open File Location
Change Icon...
Advanced...
OK
Cancel
Apply

Restarting BricsCAD using a different user profile

1. Choose **User Profile Manager** in the **Tools** menu.
2. Select the user profile you want to start up with.
3. Click the **Set Current** button on the **User Profile Manager** dialog.
4. Close and restart BricsCAD.

NOTE	If the GETSTARTED (1.13.11) system variable is ON, the **GET STARTED** page of the Welcome dialog box (1.14.1) displays on startup. Here you can select the user profile you want to use.

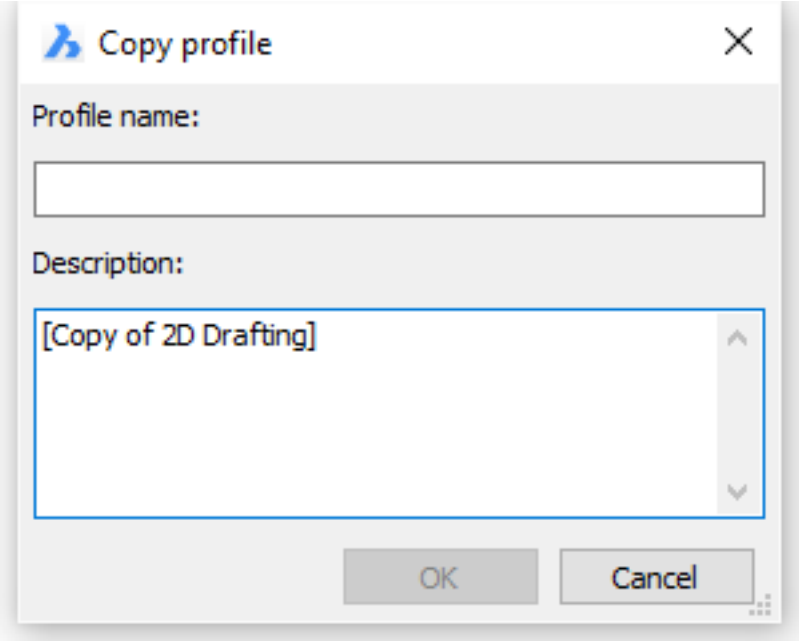
Managing user profiles

Once you start using user profiles, you may need to copy, rename or delete them. Copying a profile is a quick way to create a new profile based on an existing profile.

To copy a user profile:

1. Launch the User Profile Manager.
2. Select the user profile you want to copy.
3. Click the **Copy** button. The **Copy Profile** dialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
4. Type a name for the new profile in the **Profile name** field.
5. **Optional** Type a description for the new profile in the **Description** field.
6. Click the **OK** button to create the profile.

Figure 1.363



To rename a user profile:

1. Launch the User Profile Manager.
2. Select the user profile you want to rename.
3. Click the name of the selected user profile. The **Profile name** field is active now. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/10RenameUserProfile
4. Type a new name in the **Profile name** field.
5. Click outside the **Profile name** field.

Figure 1.364



To edit the Description field:

1. Select the profile you want to edit the description of.
2. Edit the **Description** field.

To delete a user profile:

1. Launch the User Profile Manager.
2. Select the user profile you want to delete.
3. Click the **Delete** button.
4. Click the **Yes** button on the confirmation box to delete the profile.

NOTE	The **current profile** cannot be deleted.

To export a user profile

1. Launch the User Profile Manager.
2. Select the profile you want to export in the profile list.
3. Click the **Export** button. The **Save As** dialog displays.
4. Select a folder to save the user profile in.
5. **Optional** Edit the name of the profile in the **File name** field.
6. Click the **Save** button. The file is saved with an ***.arg** extension.

To import a user profile

1. Launch the User Profile Manager.
2. Click the **Import** button. The **Open** dialog displays.
3. Select a user profile file (*.arg).
4. Click the **Open** button. The **Import Profile** dialog displays.
5. **Optional**Type a different name in the **Profile name** field.
6. **Optional** Edit the profile description.
7. Click the OK button on the **Import Profile** dialog. The profile is added to the profile list.

To import a user profile from another user

When you import a user profile of another user all user dependent search paths (e. g. the **local root** and **roamable root** folders - see BricsCAD User Files (1.13.2)) still refer to the other user, as a result the folders cannot be found and error messages such as ”Unable to open program parameters file: default.pgp” might pop up when you start BricsCAD. To correct this problem do the following:

1. Open the exported user profile (*.arg) in a text editor such as Notepad.
2. Replace the user name dependent references with an environment variable. On a Windows XP computer replace **C:\\Documents and Settings\\username** with **%USERPROFILE%.** On a Windows Vista or Windows 7 computer replace **C:\\Users\\username** with **%USERPROFILE%.**
3. Use the **Find/Replace** procedure to quickly replace all references.
4. Save the user profile (*.arg)
5. Import the user profile.

Start BricsCAD using the imported profile. The environment variable **%USERPROFILE%** will be replaced with: **C:\\Users\\curren**

NOTE	Environment variables in profiles, using the %<variable_name>% format, are supported in V10.3.11. and higher.

1.13.10 Workspaces

Commands: Workspace (6.33.15), WsSave (6.33.17) The **Workspace** command sets the current workspace and lets you create, modify and save workspaces. The **WsSave** command saves the current workspace under a different name, thus creating a new workspace. Workspaces define settings and preferences regarding a specific working environment. The following workspaces are available:

- Drafting
- Drafting (toolbars)
- Modeling
- Mechanical
- BIM

The name of the workspace is stored in the WSCURRENT system variable of the current User Profile (1.13.9). If the GETSTARTED (1.13.11) system variable is ON, the **GET STARTED** page of the Welcome dialog box (1.14.1) displays on startup. Here you can select the user profile you want to use.

Managing workspaces

Workspaces are defined in the **Workspace** tab page on the **Customize** dialog. Do one of the following:

- Open the Customize dialog (1.13.3), then select the **Workspace** tab.
- Right click the **Current Workspace** field in the Status Bar (1.12.2), then choose **Customize Workspaces...** in the context menu.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/13C
A workspace definition is composed of:

- Menus : defines which menus are available.
- Toolbars : defines which toolbars display by default.
- Palettes : controls the visibility of various palettes. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
  Right click a palette then choose an option in the context menu:
  - Show: 
  - Hide: 
  - Don't change: 
- Ribbon : defines the available ribbon tabs.
- Quad Group Order : defines the order and availability of the Quad command groups.
- On Switch : defines the system variables and user preferences to be set and the commands to be executed when switching to the workspace.

Figure 1.365

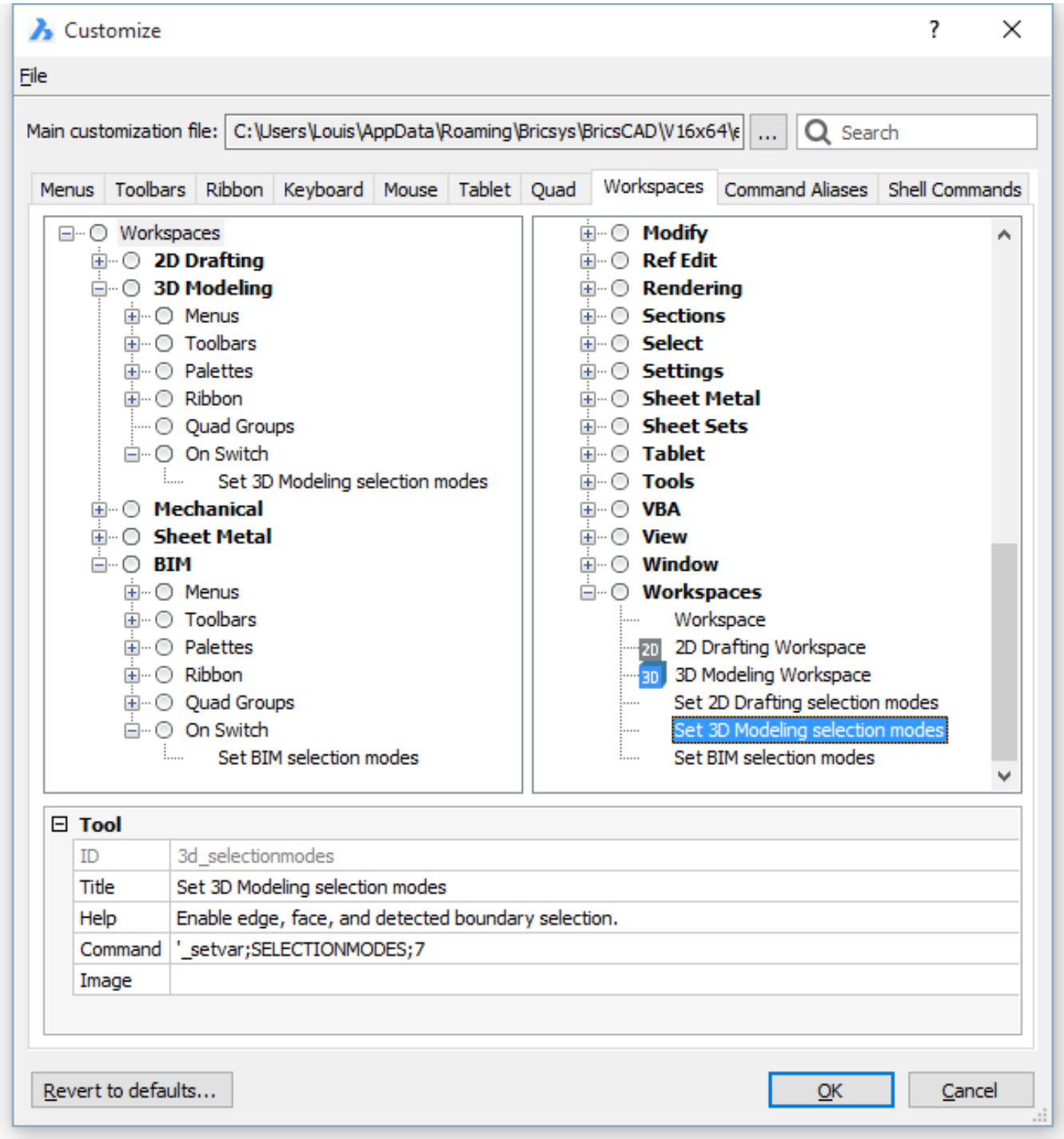
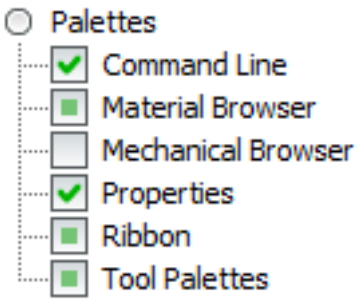


Figure 1.366



Setting the current workspace

Do one of the following:

- Right click the **Current Workspace** field in the Status Bar (1.12.2), then select a workspace in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/02Definingyourpreferenc
- Choose a workspace in the **Workspaces** toolbar. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18
- Type **wscurrent** at the command prompt. You are prompted: New value for WSCURRENT, or . for none/<"2d DRAFTING">: Type the name of the new current workspace, then press enter.

NOTES

- If the WSAUTOSAVE (1.13.11) system variable is On, upon switching to another workspace, the toolbar positions of the current workspace are saved in the CUI file. The default value for WSAUTOSAVE is ON.
- It is not recommended to alter the workspace if the name of the user profile refers to a workspace.

Figure 1.367

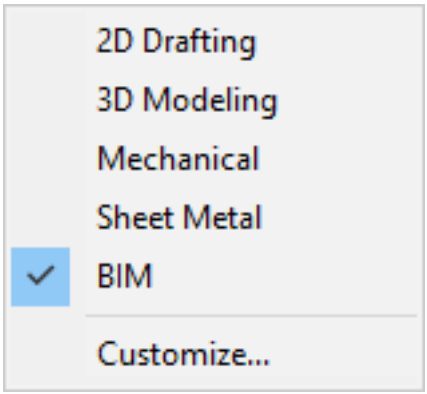
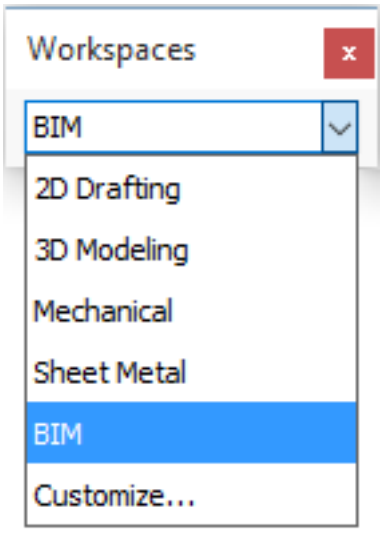


Figure 1.368



Creating a workspace

1. Right click the **Current Workspace** field in the Status Bar (1.12.2), then choose **Customize Workspaces...** in the context menu.
2. Do one of the following:
   - Select **Workspaces** , then right click and choose **Append Workspace** in the context menu.
   - Select an existing workspace, then right click and choose **Insert Workspace** in the context menu.
3. Type a name for the new workspace, then click the **OK** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel

When appending a workspace, the new workspace is created below the last existing workspace. When inserting a workspace, the new workspace is created above the selected workspace. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco
A newly created workspace contains the following items:

- A standard menu set
- A standard toolbar set
- All palettes set to **Don't change**
- An empty ribbon
- A default Quad group order
- An empty **On Switch** sequence

Figure 1.369

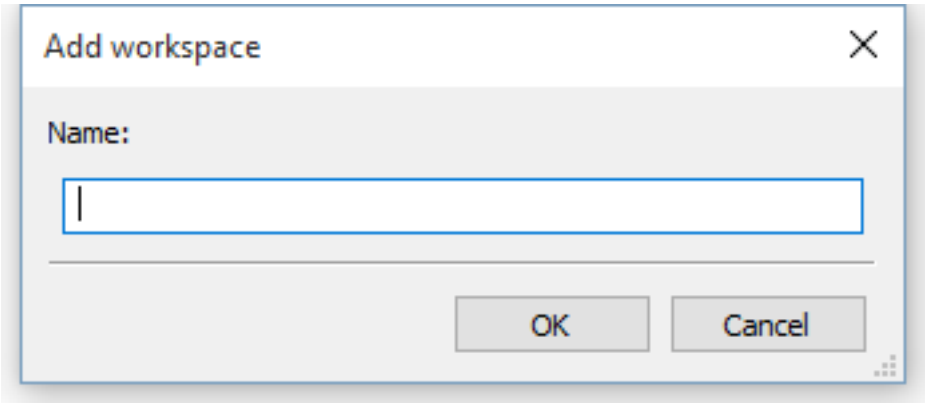
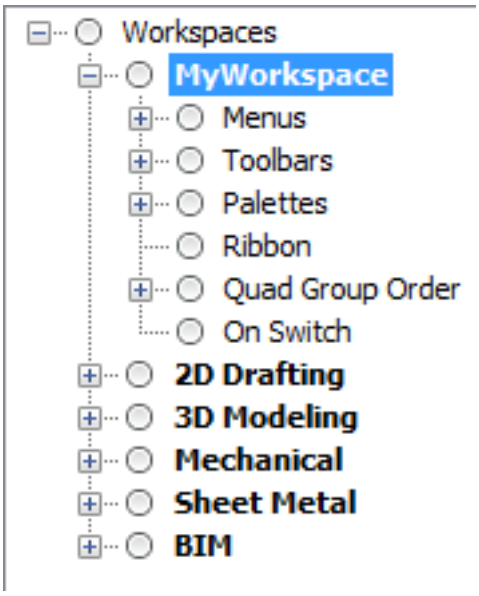


Figure 1.370



Editing a workspace

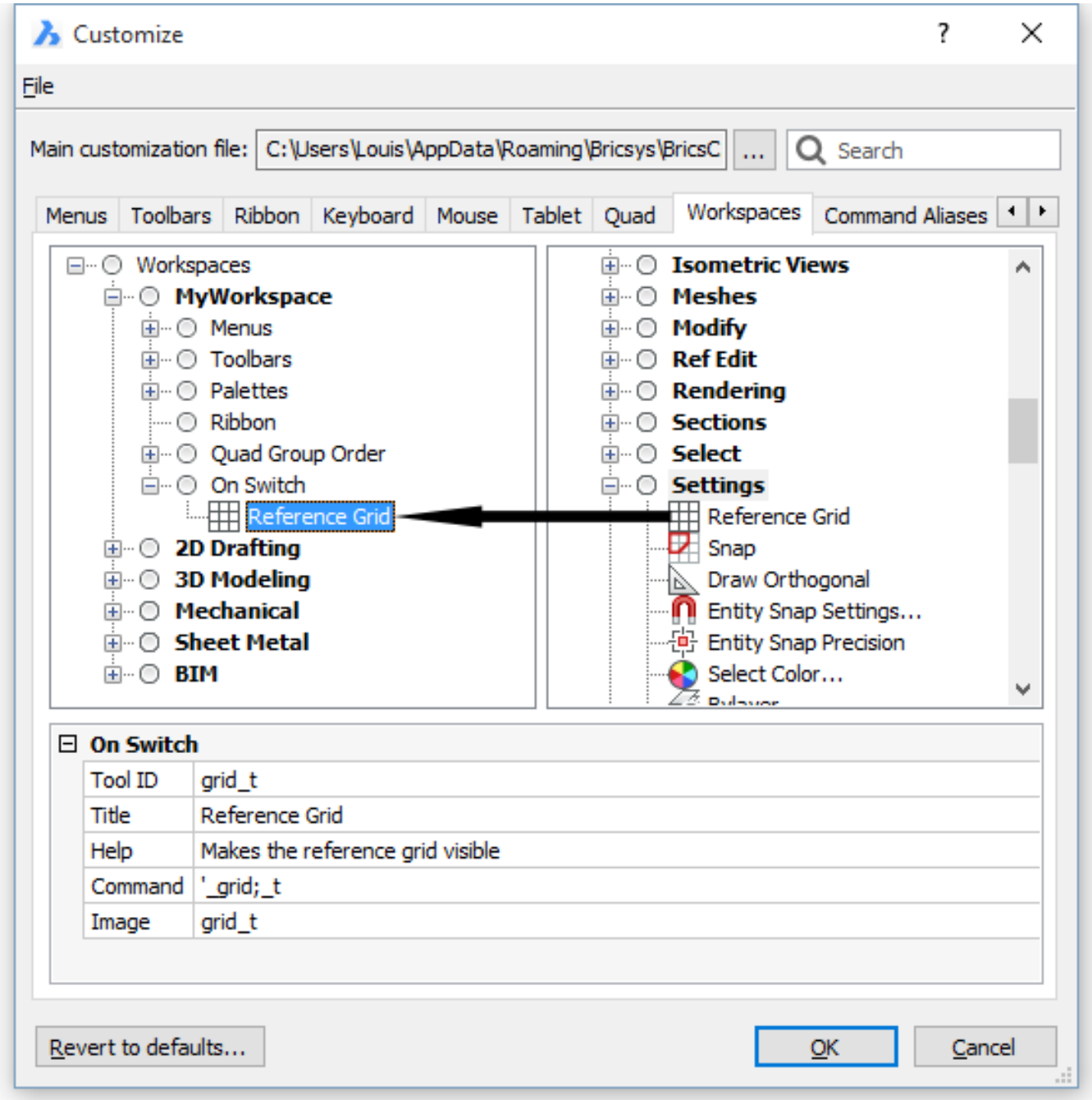
1. Right click the **Current Workspace** field in the Status Bar (1.12.2), then choose **Customize...** in the context menu.
2. Click the **Expand** button (+) of the workspace.
3. Edit the menu set:
   - Change the order: Drag a menu to another location.
   - Delete a menu: select the menu, then right click and choose **Remove** in the context menu.
   - Insert a menu:
     - Right click a menu in the list and choose **Insert menu** in the context menu. The **File** menu is inserted by default.
     - In the **Workspace Menu** grid, select the **ID** field and select the desired menu in the list.
4. Edit the toolbar set:
   - Change the order: Drag a toolbar to another location.
   - Delete a toolbar: select the toolbar, then right click and choose **Remove** in the context menu.
   - Insert a toolbar:
     - Right click a toolbar in the list, and choose **Insert toolbar** in the context menu. The **Standard** toolbar is inserted by default.
     - In the **Workspace Toolbar** grid, select the **ID** field and select the desired toolbar in the list.
5. Edit the visibility of the palettes:
   - Select the palette.
   - Click the **Display** property in the **Workspace Palette** grid, then set the desired 'on switch' behavior: **Show** , **Hide** or **Don't change** .
6. Edit the composition of the ribbon:
   - To add a tab: right click an existing tab, then choose **Insert tab** in the context menu and select the tab in the **Select Ribbon Tab** dialog.
   - To remove a tab: right click the tab and choose **Remove** in the context menu.
   - To change the position of a tab: press and hold the left mouse button, then drag the tab to the desired place.
7. Edit the Quad settings:
   - To change the order of the Quad command groups: press and hold the left mouse button, then drag a command group to its new position.
   - Right click a Quad command group, then choose **Switch off** in the context menu to remove the command group from the Quad cursor menu.

- Right click a currently switched off Quad command group then choose **Switch on** in the context menu to add the command group to the Quad cursor menu.

8. Edit the **On Switch** procedure: To add a command to the **On Switch** sequence: select the command in the available commands pane and drag it to the **On Switch** node. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help

NOTE	See To create a new tool (1.13.5) if you want to add a custom tool to the **On Switch** sequence.

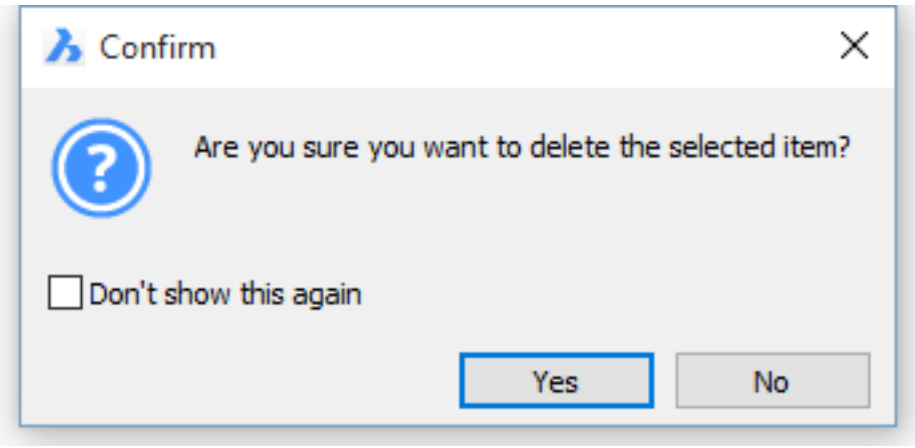
Figure 1.371


Customize
File
Main customization file:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsC
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Workspaces
Command Aliases
Workspaces
MyWorkspace
Menus
Toolbars
Palettes
Ribbon
Quad Group Order
On Switch
Reference Grid
2D Drafting
3D Modeling
Mechanical
Sheet Metal
BIM
Isometric Views
Meshes
Modify
Ref Edit
Rendering
Sections
Select
Settings
Reference Grid
Snap
Draw Orthogonal
Entity Snap Settings...
Entity Snap Precision
Select Color...
On Switch
Tool ID grid_t
Title Reference Grid
Help Makes the reference grid visible
Command '_grid;_t
Image grid_t
Revert to defaults...
OK
Cancel

Deleting a workspace

1. Right click the **Current Workspace** field in the Status Bar (1.12.2), then choose **Customize Workspaces...** in the context menu.
2. Right click the workspace, then choose Delete in the context menu. A Confirm dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/13C
3. Press the **Yes** button to delete the selected workspace.

Figure 1.372

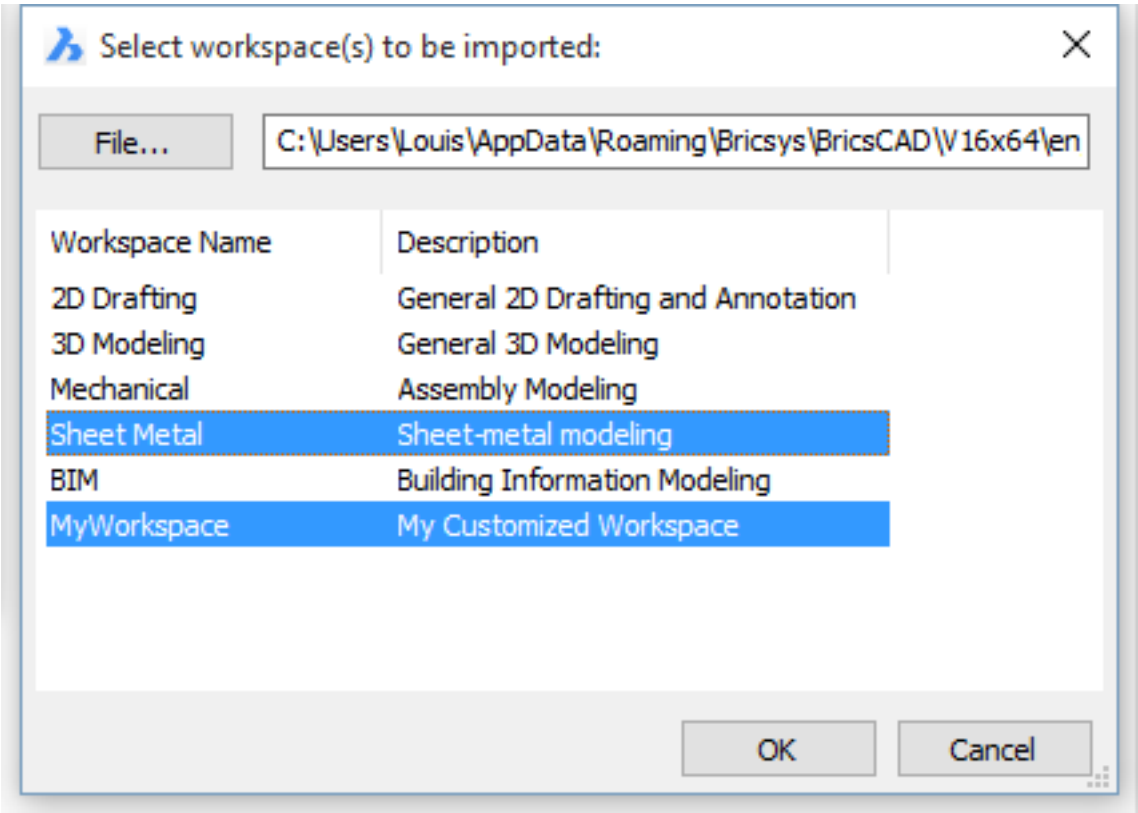


Importing a workspace

The option in the **File** menu on the **Customize** dialog box can import workspaces from another CUI file into the current main CUI.

1. Open the Customize (1.13.3) dialog.
2. In the **File** menu choose **Import Workspace(s)...** The **Choose a Customization File** dialog displays.
3. Browse to the folder where the CUI file, you want to import the workspace from, is saved. By default BricsCAD saves CUI files in the **Support** folder of the Roamable Root folder (1.13.2). e.g. **C:\Users\<user_name>\AppData\Roamin**
4. Select the CUI file. A dialog box, listing all workspaces in the selected CUI file, displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/13ImportWorkspaceD
5. Select the workspace to import in the list, then click the **OK** button. Press and hold the Ctrl-button to select multiple workspaces.

Figure 1.373


Select workspace(s) to be imported:
File...
C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V16x64\en
Workspace Name
Description
2D Drafting
General 2D Drafting and Annotation
3D Modeling
General 3D Modeling
Mechanical
Assembly Modeling
Sheet Metal
Sheet-metal modeling
BIM
Building Information Modeling
MyWorkspace
My Customized Workspace
OK
Cancel

1.13.11 Settings dialog box

Command: Settings (6.27.23)

Tutorial: Settings is Better Than Options and Here's Why Click here (1.3.6) to watch

Access the Settings dialog box

BricsCAD centralizes all system variables and user preferences in a single dialog box called Settings. If you like to type, you can access the Settings dialog box with the Options (6.23.16) or DSettings (6.10.46) commands. If you prefer graphical access, you'll find the **Settings** tool in the **General** command group on the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/SettingsQuad.png}) Tools in the Settings dialog box enable you to find and edit the 900+ system variables and user preferences in BricsCAD. You can even export all them and their values to a CSV file. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
The tools at the top of the Settings dialog box, allow you to view the settings in different ways:

- : Categorized
- : Alphabetical
- : Differences compared to the defaults

Below those tools is the list of settings. And, at the bottom, you'll find useful information about the selected setting. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/SettingsDetailDialog.png})

1. Selected setting
2. Name
   - System variable names are fully capitalized: INSUNITS
   - User preference names are mixed case: TemplatePath
3. Title
4. Description
5. Type and save location
   - : Editable setting
   - : Not-editable setting
   - : Saved in the drawing = can be defined differently in each drawing
   - : Saved in the registry = applies to all drawings opened in the current user profile
   - : Setting cannot be saved
   - BricsCAD-only : Most settings correspond to AutoCAD variables. There are, however, a few that are unique to BricsCAD. They are clearly indicated by an additional icon.

Figure 1.374

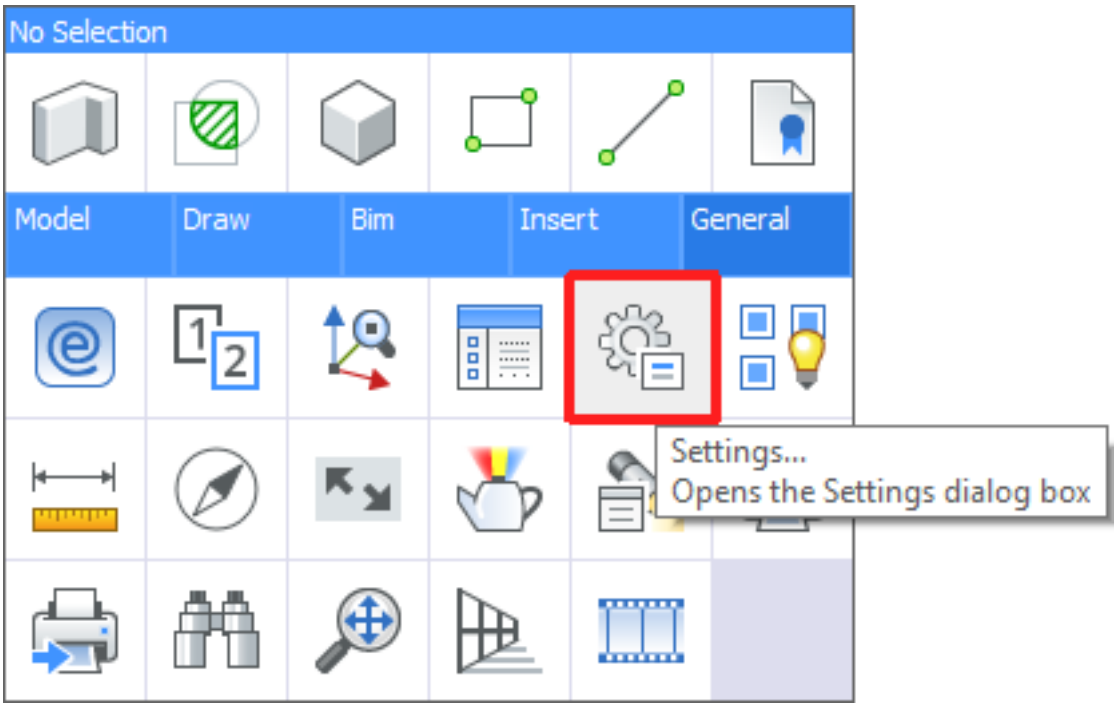

No Selection
Model
Draw
Bim
Insert
General
Settings...
Opens the Settings dialog box

Figure 1.375

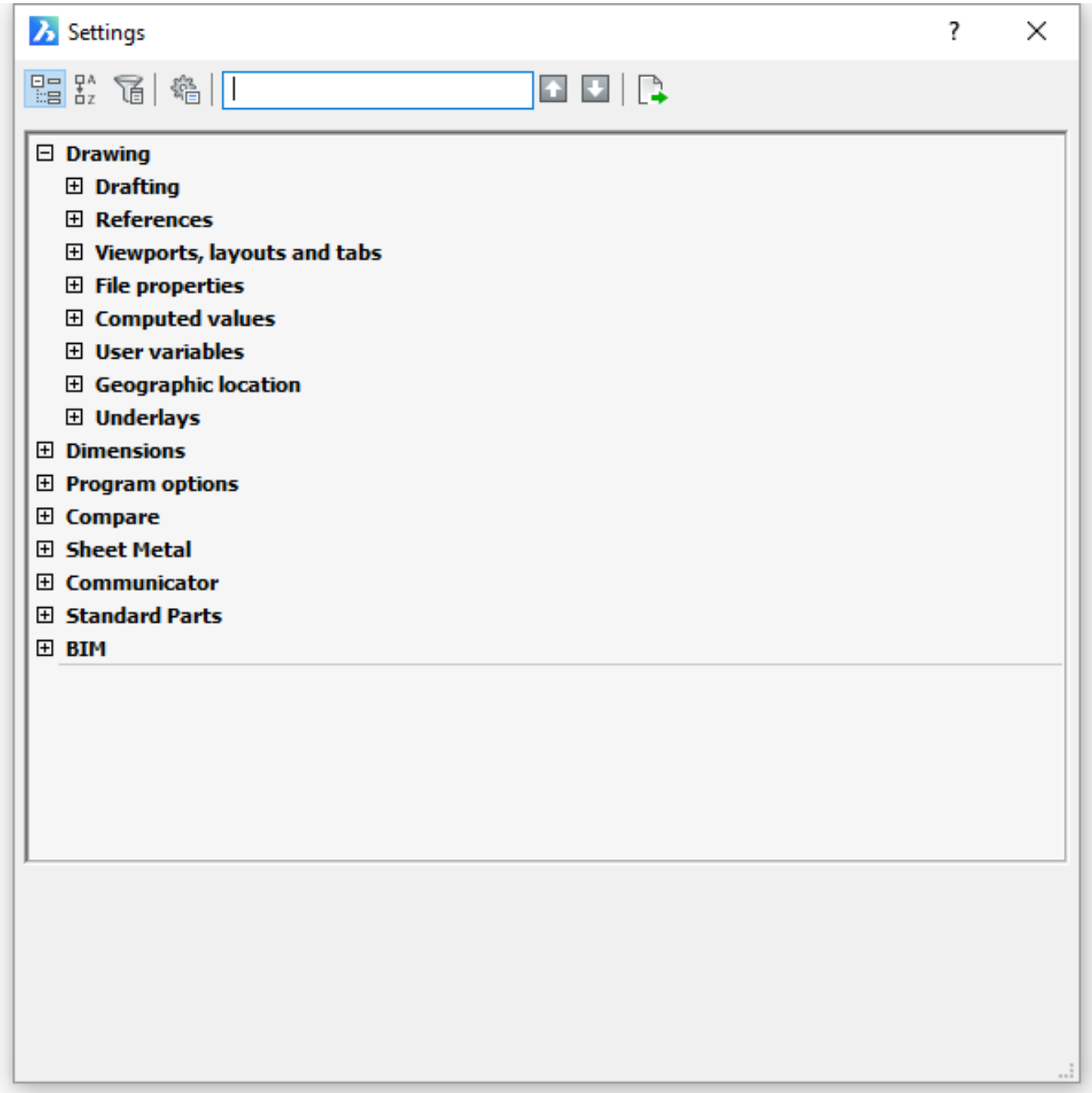
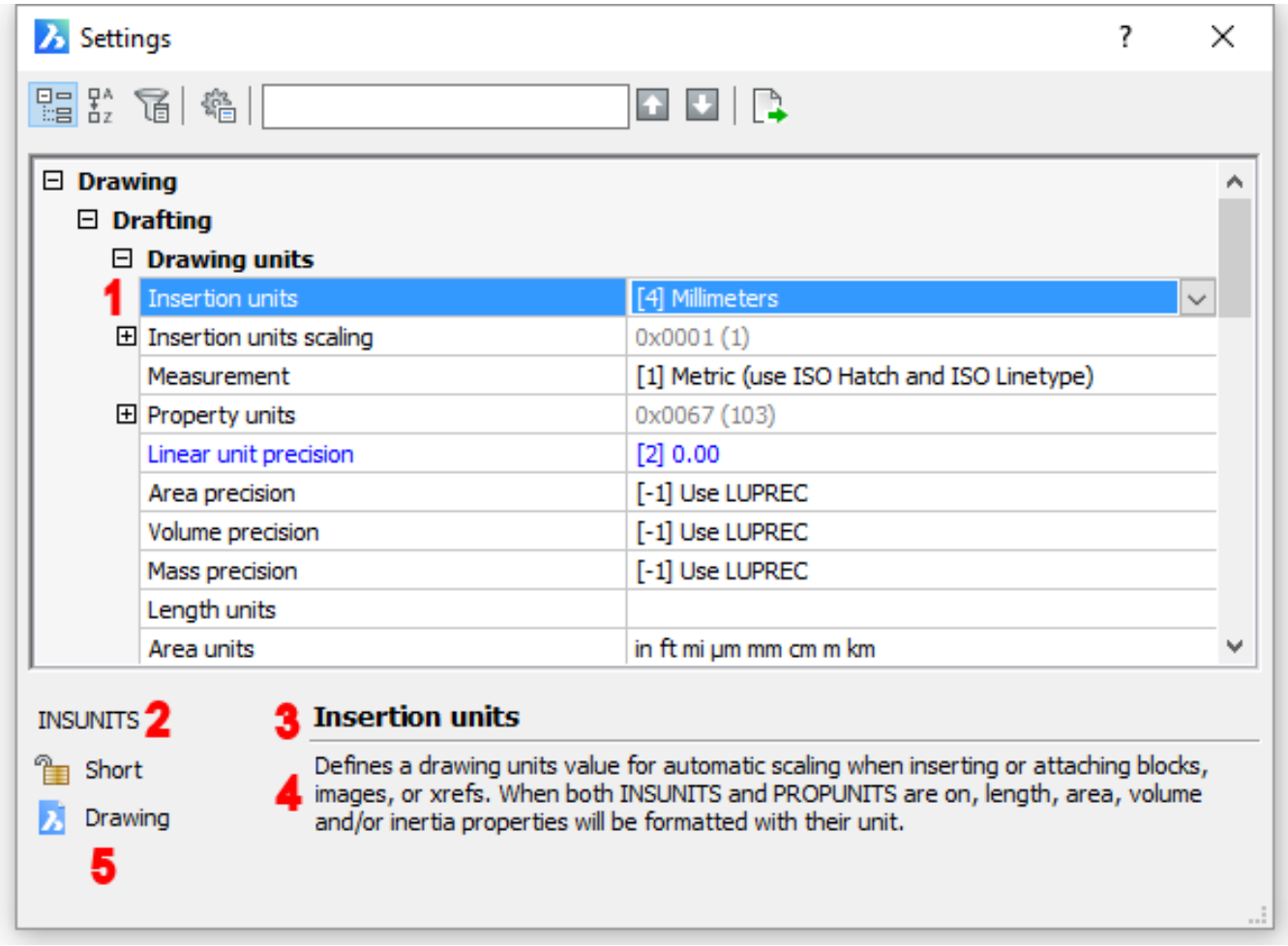

Settings
Drawing
Drafting
References
Viewports, layouts and tabs
File properties
Computed values
User variables
Geographic location
Underlays
Dimensions
Program options
Compare
Sheet Metal
Communicator
Standard Parts
BIM

Figure 1.376


Settings
Drawing
Drafting
Drawing units
1
Insertion units
[4] Millimeters
Insertion units scaling
0x0001 (1)
Measurement
[1] Metric (use ISO Hatch and ISO Linetype)
Property units
0x0067 (103)
Linear unit precision
[2] 0.00
Area precision
[-1] Use LUPREC
Volume precision
[-1] Use LUPREC
Mass precision
[-1] Use LUPREC
Length units
Area units
in ft mi µm mm cm m km
INSUNITS 2
Short
Drawing
5
3 Insertion units
4 Defines a drawing units value for automatic scaling when inserting or attaching blocks, images, or xrefs. When both INSUNITS and PROPUNITS are on, length, area, volume and/or inertia properties will be formatted with their unit.

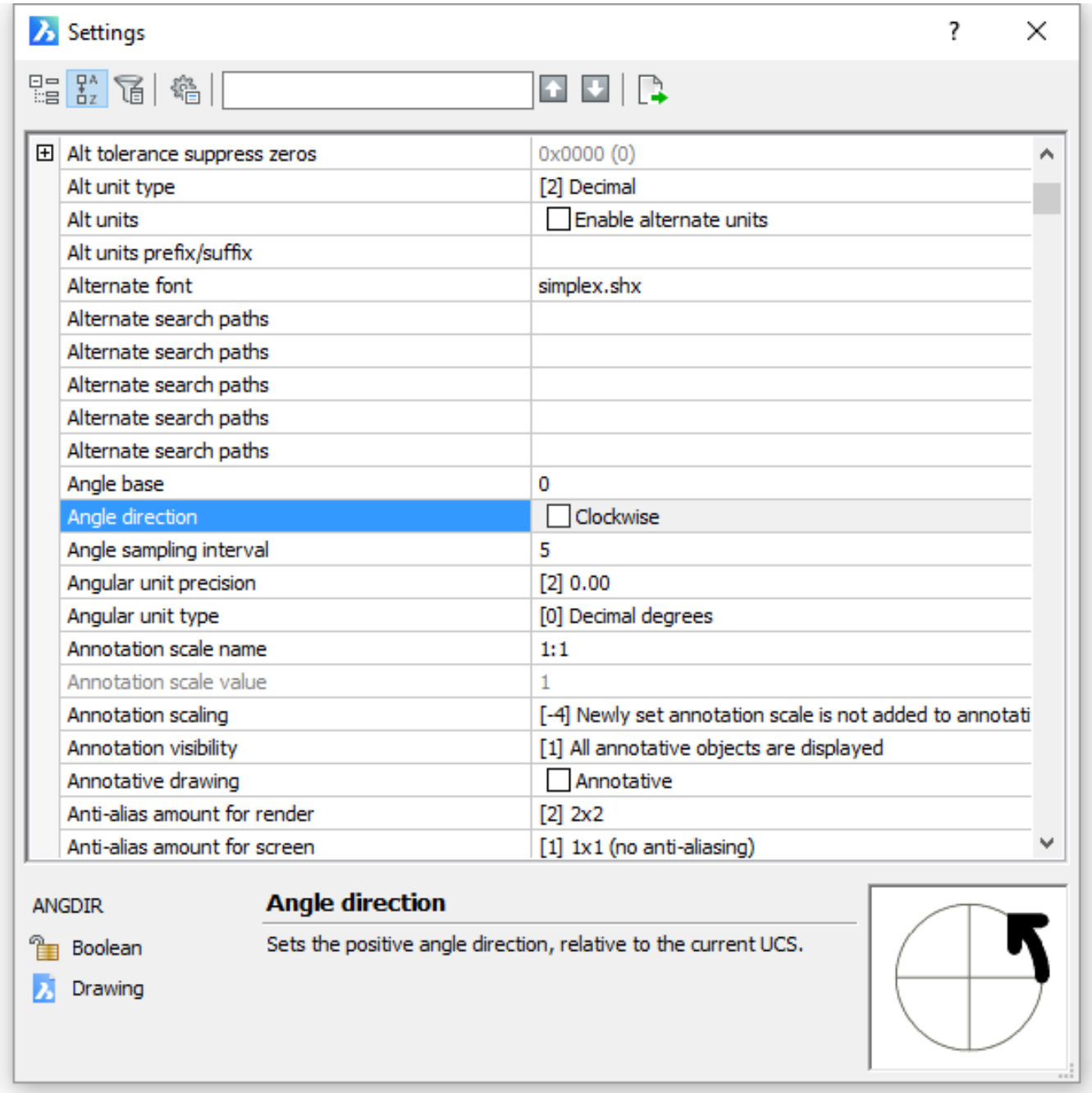
Categorized View

The Categorized tool displays a tree view that groups the variables into the various categories. You can expand each category to access relevant variables. And, in many cases, you can expand additional subcategories.

Alphabetical View

If you know the name of the setting you want to modify, you may prefer to display the settings instances rather than categorically. Doing so lists all the settings alphabetically based on their **Title**. The title is different from the actual variable name. For example, **Angle direction** controls the ANGDIR system variable. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/SettingsAlphabeticalDialog.png})

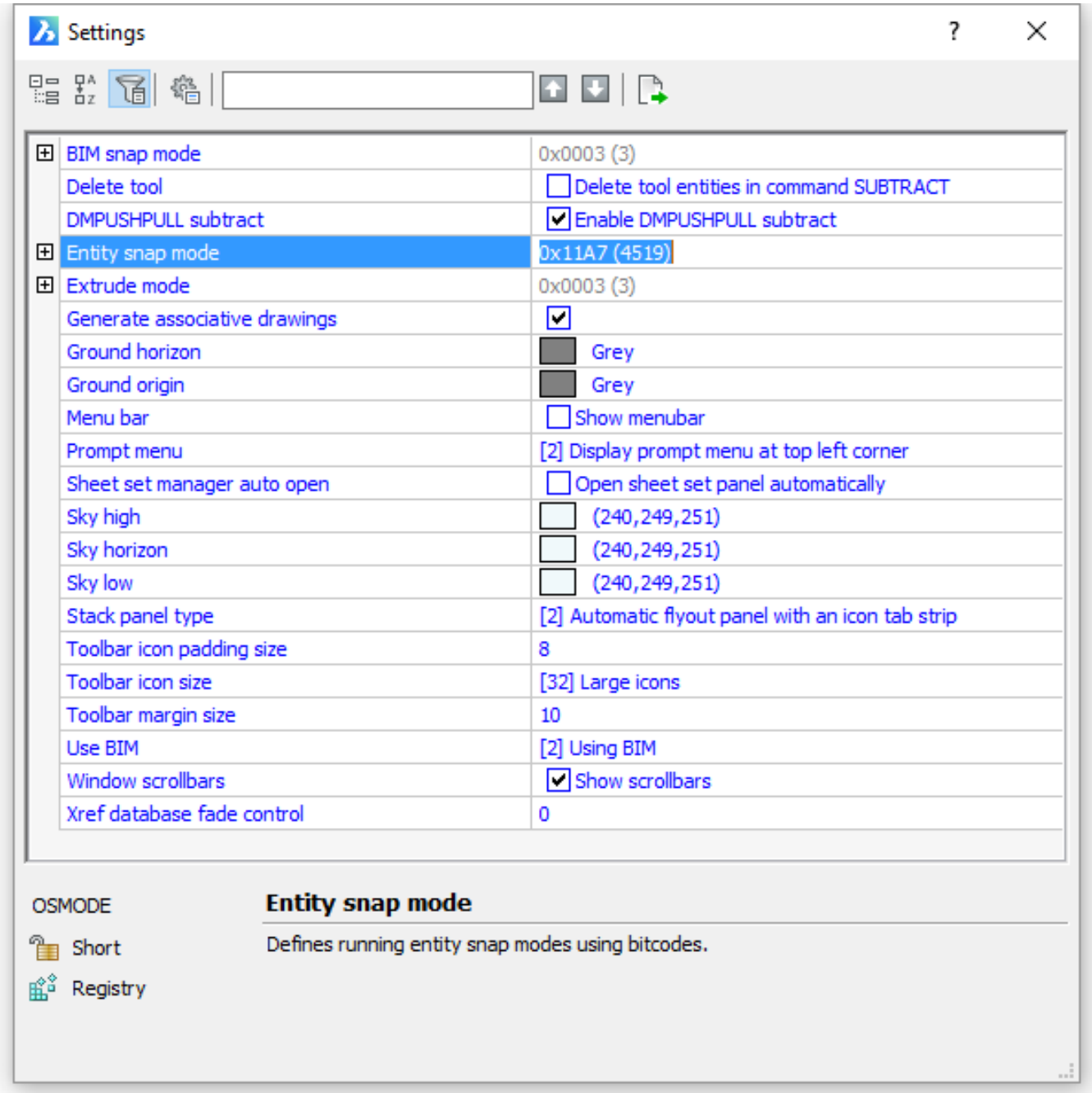
Figure 1.377


Settings
Alt tolerance suppress zeros
0x0000 (0)
Alt unit type
[2] Decimal
Alt units
Enable alternate units
Alt units prefix/suffix
Alternate font
simplex.shx
Alternate search paths
Alternate search paths
Alternate search paths
Alternate search paths
Alternate search paths
Angle base
0
Angle direction
Clockwise
Angle sampling interval
5
Angular unit precision
[2] 0.00
Angular unit type
[0] Decimal degrees
Annotation scale name
1:1
Annotation scale value
1
Annotation scaling
[-4] Newly set annotation scale is not added to annotati
Annotation visibility
[1] All annotative objects are displayed
Annotative drawing
Annotative
Anti-alias amount for render
[2] 2x2
Anti-alias amount for screen
[1] 1x1 (no anti-aliasing)
ANGDIR
Boolean
Drawing
Angle direction
Sets the positive angle direction, relative to the current UCS.

Differences with default view

Only settings which are different from the defaults are listed in this view mode. (\vref {fig:C:/Dev/Task-2448/images/staticbrics

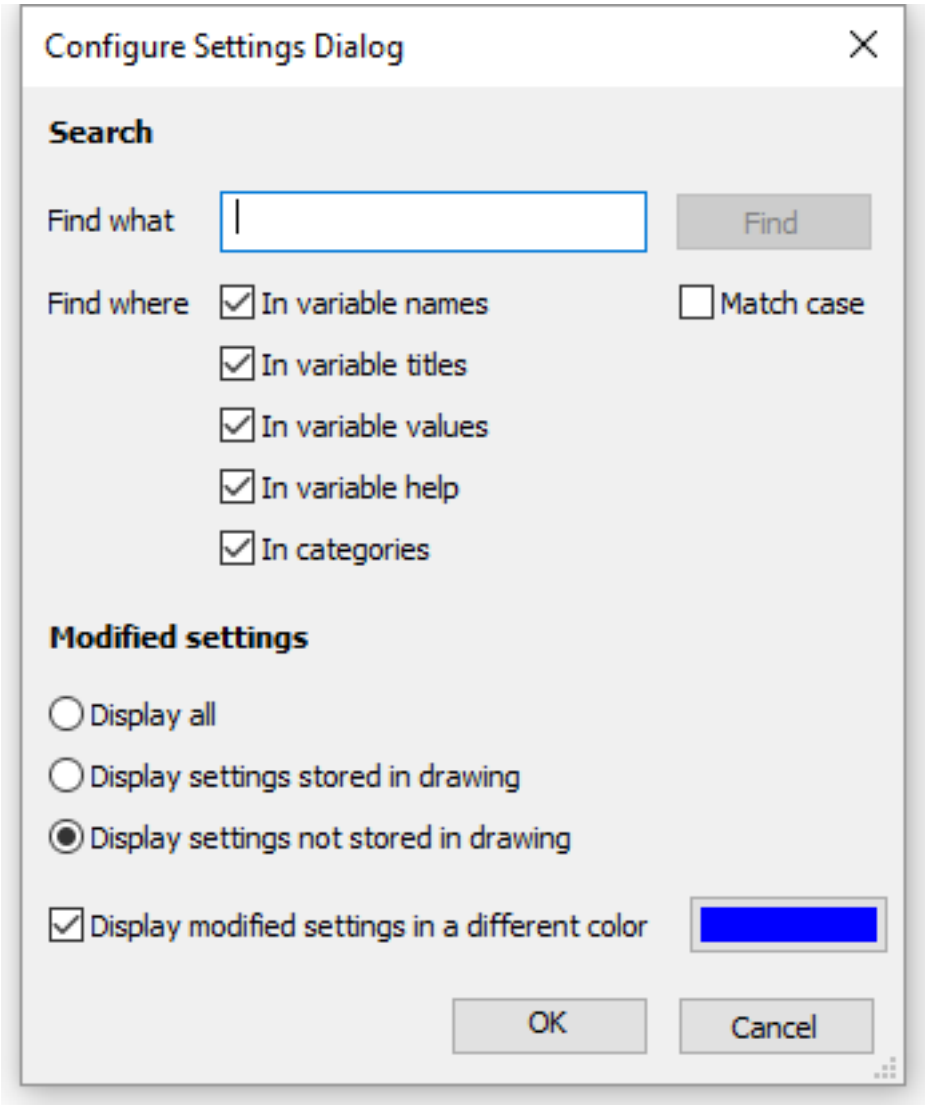
Figure 1.378


Settings
BIM snap mode
0x0003 (3)
Delete tool
Delete tool entities in command SUBTRACT
DMPUSHPULL subtract
Enable DMPUSHPULL subtract
Entity snap mode
0x11A7 (4519)
Extrude mode
0x0003 (3)
Generate associative drawings
Ground horizon
Grey
Ground origin
Grey
Menu bar
Show menubar
Prompt menu
[2] Display prompt menu at top left corner
Sheet set manager auto open
Open sheet set panel automatically
Sky high
(240,249,251)
Sky horizon
(240,249,251)
Sky low
(240,249,251)
Stack panel type
[2] Automatic flyout panel with an icon tab strip
Toolbar icon padding size
8
Toolbar icon size
[32] Large icons
Toolbar margin size
10
Use BIM
[2] Using BIM
Window scrollbars
Show scrollbars
Xref database fade control
0
OSMODE
Short
Registry
Entity snap mode
Defines running entity snap modes using bitcodes.

Configure Settings dialog box

Click the **Configure** icon () on the **Settings** toolbar. The **Configure Settings Dialog** allows you to control how BricsCAD searches for settings and how modified settings are listed when the **Differences** view mode is selected.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/SettingsConfigureDialog.png})

Figure 1.379


Configure Settings Dialog
Search
Find what
Find
Find where
In variable names
Match case
In variable titles
In variable values
In variable help
In categories
Modified settings
Display all
Display settings stored in drawing
Display settings not stored in drawing
Display modified settings in a different color
OK
Cancel

Find settings

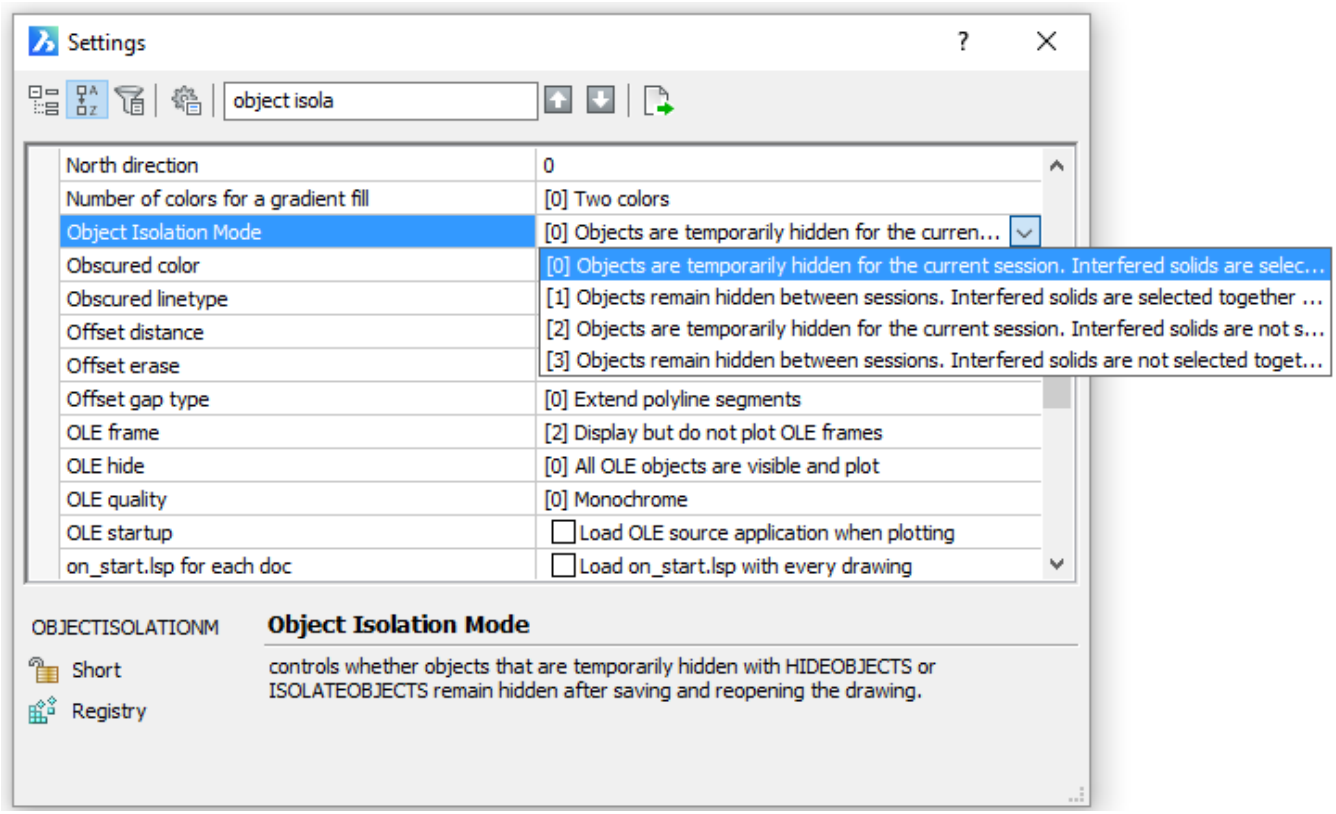
To find a setting do the following:

1. **Optional** Open the **Configure Settings Dialog**to set the Find where options.
2. Type a string in the Search field. As you type the first setting matching the content of the search field highlights.
3. Do one of the following:
   - Keep typing typing: the next matching setting highlights as you type.
   - Click the **Find Next** () or **Find Previous** () buttons to jump to the previous or next setting matching the content of the search field. The background color of the search field turns green when the final matching setting has been reached.

Edit settings

After you find the setting you want, select it to view its current value and properties. If the setting is editable, you can click in the value box to change it. The value options vary depending on the setting. For example, some may require you to enter a number, while others may include a check box or drop-down list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Customization/SettingsEditDialog.png}) Some settings are grey indicating they're read-only. In addition, a lock icon () in the lower left corner of the Settings dialog box clearly identifies them as read-only .

Figure 1.380



Edit system variables at the Command line

The Settings dialog box offers a graphical and intuitive method for you to view and edit system variables and user preferences. It can be particularly useful if you don't know the exact name of the variable or if you want to change several variables. However, if you prefer to view and edit variables via the Command line, you can do that too! Simply enter the variable name or launch the SETVAR (6.27.26) command. System variable names are recognized by the **Autocomplete** feature of the command bar if the **Exclude the display of system variables** option of the **AUTOCOMPLETEMODE** system variable is not checked. User preference names arerecognized by the **Autocomplete** feature of the command bar if the **Display preference variables** option of the **AUTOCOMPLETEMODE** system variable is checked.

Export settings

Regardless of how you view or edit settings in BricsCAD, you can easily export them to a comma delimited (CSV) file. Click the **Export** icon ( ) then enter the file name and location on the **Export Settings** dialog box. With the CSV file created, you can open it in a spreadsheet app to easily view all the variable properties. This can be especially helpful for troubleshooting behavior differences between different systems or drawings.

1.14 Managing Drawings

1.14.1 Creating a drawing

Commands: NEW (6.22.4), NEWWIZ (6.22.6) and QNEW (6.25.2) In BricsCAD you can create new drawings:

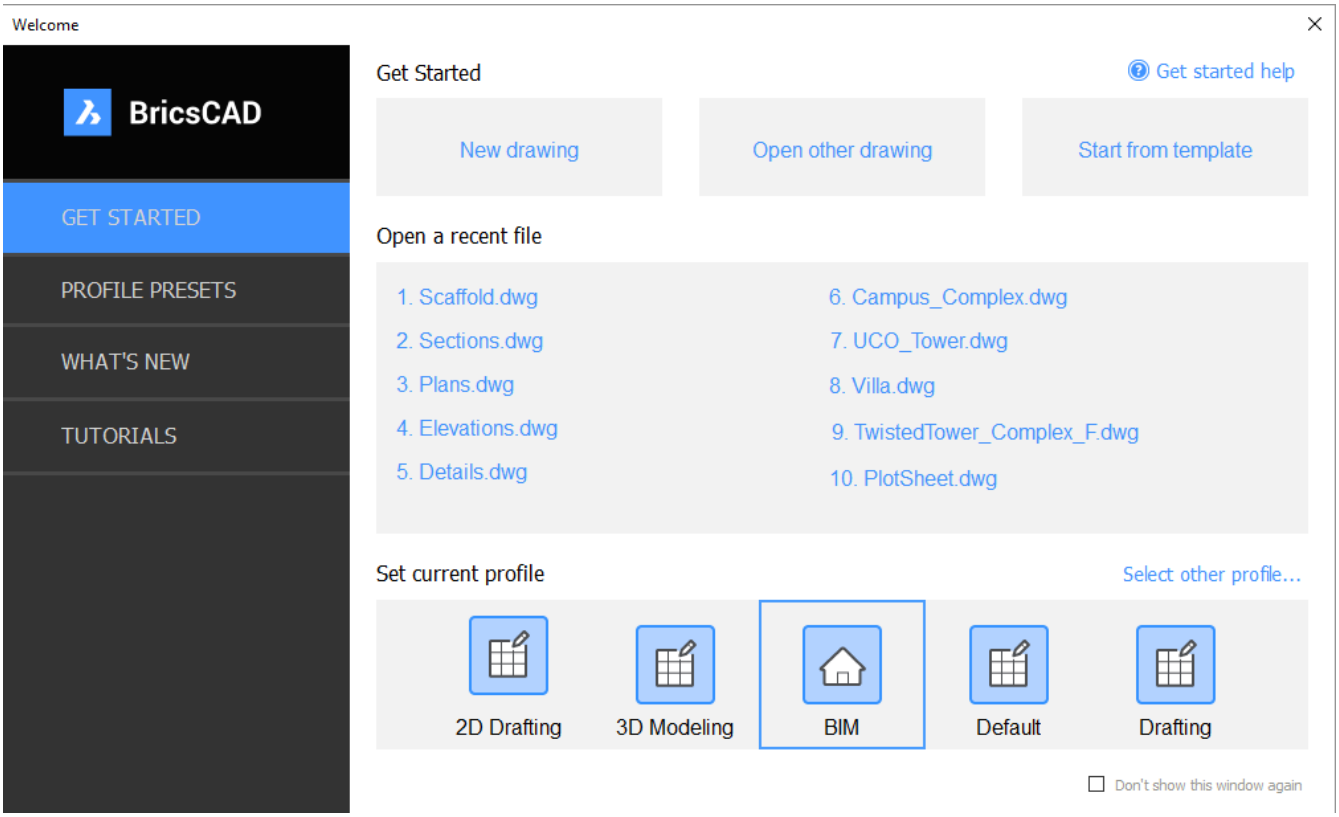
- Using a default template drawing.
- Starting from scratch.
- Using a template drawing of your choice.
- Using a wizard.
- Double clicking a *.dwt file in a Windows Explorer window.

Using the Welcome dialog

When you launch BricsCAD and the value of the GETSTARTED (1.10.6) system variable is ON, the GET STARTED page of the Welcome dialog box (1.10.6) displays. Check the **Don't show this window again** option to suppress the display of the Welcome dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDraw
On the GET STARTED page, do one of the following:

- Select a profile in the **Set current profile** list, then click **New Drawing** to start BricsCAD with a new drawing using the default template of the selected profile. Default template drawings in the **Profile Presets** are: **Profile Preset**
- Select a profile in the **Set current profile** list, then click **Start from template** to start BricsCAD with a new drawing using a template, different from the current default template of the selected profile. The selected template becomes the default template of the selected profile.

Figure 1.381

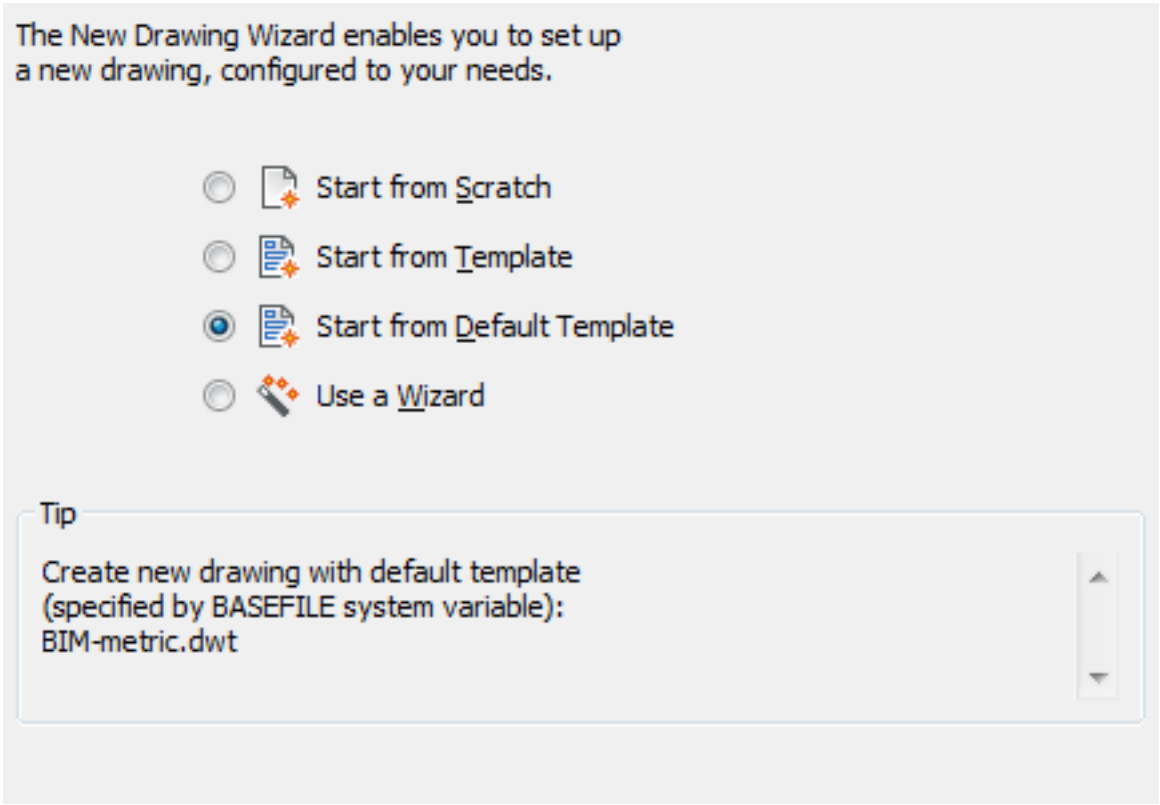


To create a drawing using a default template drawing

First method:

1. Do one of the following:
   - Choose **New wizard...** in the File menu.
   - Type **newwiz** in the command window, then press Enter.
2. On the Create New Drawing dialog choose **Start from Default template**, then click the **Next** button. The drawing is created as a copy of the default template. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/I

Figure 1.382



Second method: Do one of the following:

- Click the **QNew** tool button ( ) on the **Standard** toolbar.
- type **qnew** in the command bar, then press Enter.

The drawing is created as a copy of the default template defined by the BASEFILE system variable. If the default template drawing is not specified, the Create New Drawing wizard is launched if the STARTUP system variable is ON, otherwise the Select Template dialog displays.

To set the default template drawing

1. Open the Settings (1.13.11) dialog.
2. Under **Program Options / Files / Templates** select **Template (BASEFILE)**
3. Click the **Browse** button.
4. On the Choose a File dialog, select a drawing file (*.dwg) or drawing template (*.dwt), then click the **Open** button.

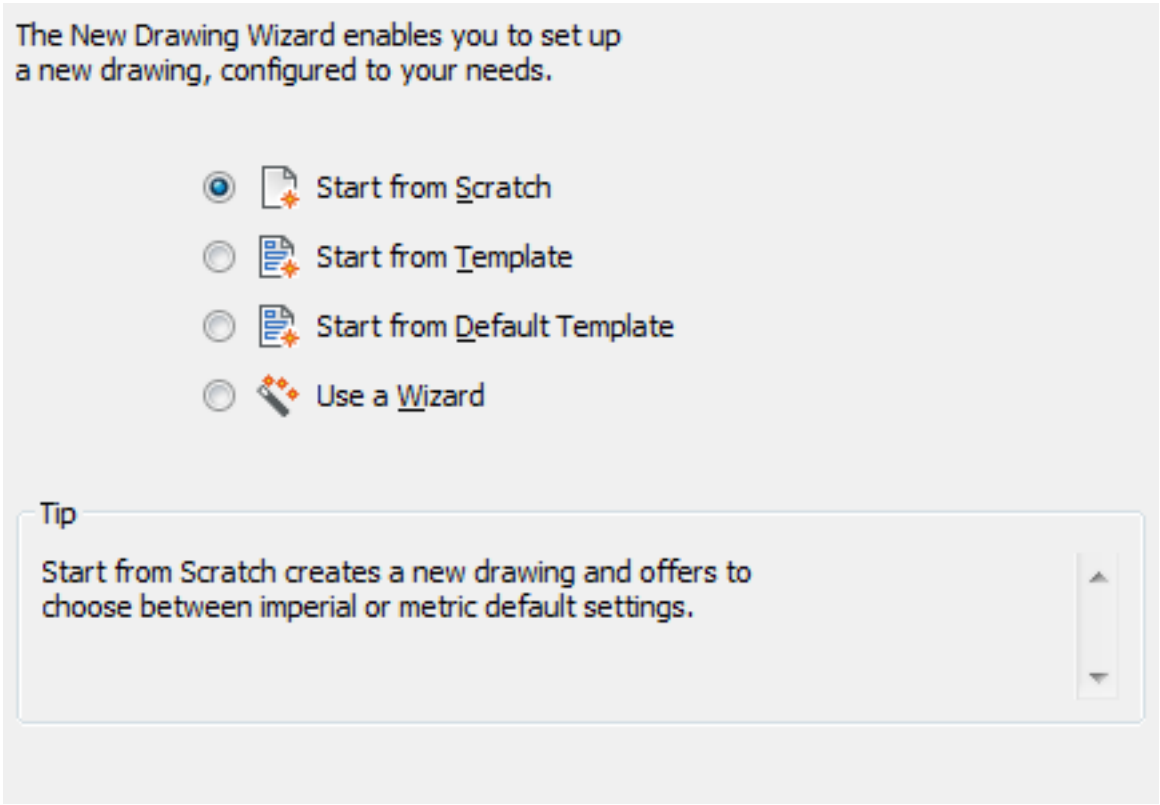
Tutorial: Creating Custom Template Drawings Click here (1.3.15) to watch

To create a drawing from scratch

1. Do one of the following:
   - Choose **New wizard...** in the File menu.
   - Type **newwiz** in the command window, then press Enter.
2. On the Create New Drawing dialog choose **Start from Scratch**, then click the **Next** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/NewFromScratch.png})
3. Choose between either **Imperial** or **Metric** default settings. Click the **Finish** button to create the drawing.

Note: Using the above procedure it depends on the status of the BASEFILE and PSTYLEPOLICY variables whether **CTB** (Color table) or **STB** (Named plot style table) will be used to set up plotter configurations in the newly created drawing.

Figure 1.383



BASEFILE	PSTYLEPOLICY	CTB or STB?
not specified	OFF	STB
not specified	ON	CTB
specified	ON or OFF	as defined in the template drawing

To set the Plot style policy

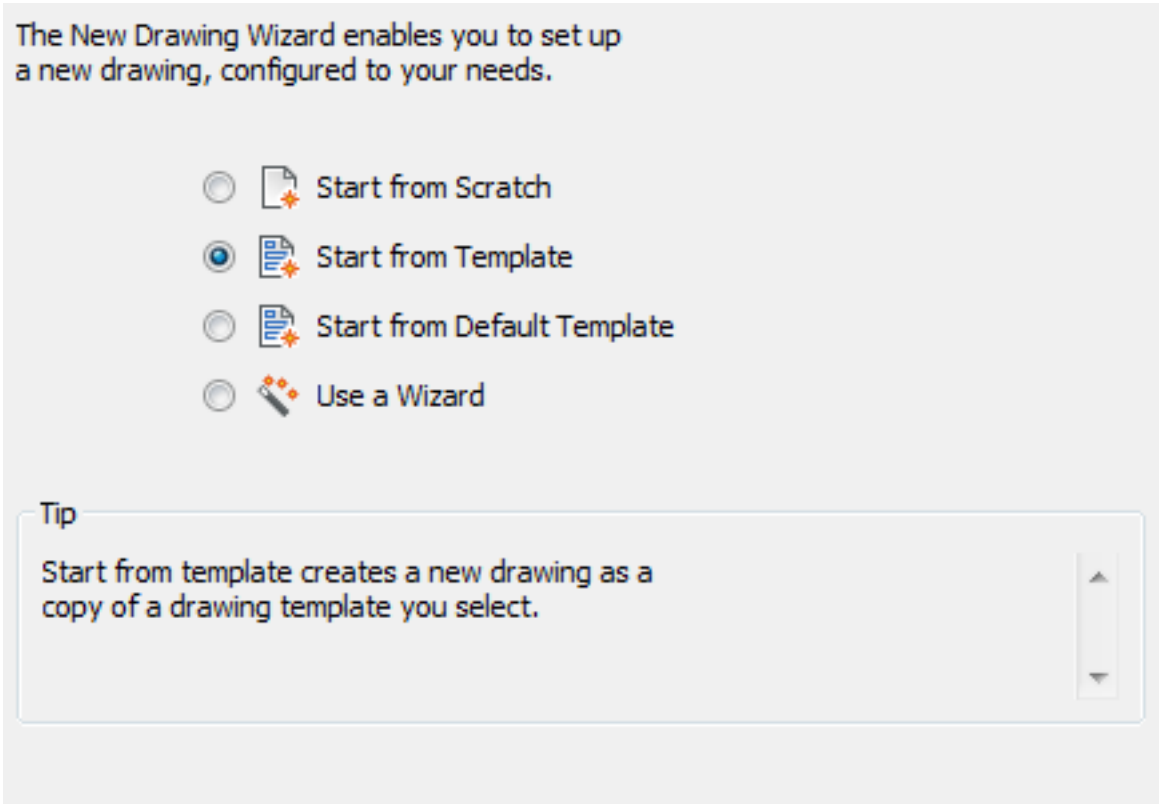
1. Open the Settings (1.13.11) dialog.
2. Under **Program Options / Plot and publish** select **Plot style policy (PSTYLEPOLICY)** .
3. Do one of the following:
   - Choose **Associate an entity's plot style with its color** for Color dependent plot styles.
   - Choose **No association between color and plot style** for Named plot styles.

To create a drawing using a template drawing

1. Do one of the following:
   - Choose **New wizard...** in the File menu.
   - Type **newwiz** in the command window, then press Enter.
2. On the Create New Drawing dialog choose **Start from template**, then click the **Next** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/NewTemplate.png})
3. Do one of the following:
   - Choose a template drawing in the **Select a template list**. By default this is the content of the **Template** folder in the **Local root** folder. Edit the **Template Path** variable to set your template folder differently.
   - Click the **Browse...** button to select a template drawing. You can choose between ***.dwg** and ***.dwt** file types.

The drawing is created as a copy of the selected template.

Figure 1.384



To create a drawing using the New command

1. Do one of the following:
   - Choose **New...** in the File menu.
   - Type **new** in the command bar, then press Enter.
2. The Select template dialog opens showing the content of the **Template** folder as defined in the **Template Path** variable.
3. Select the template drawing, then click the **Open** button. The drawing is created as a copy of the selected template.

To set the Template Path variable

1. Open the **Settings** dialog.
2. Under **Program Options / Files / Templates** select **Template Path (templatePath)**
3. Click the **Browse** button.
4. On the Browse for folder dialog, select a folder , then click the **OK** button.

To create a drawing using a wizard

1. Do one of the following:
   - Choose **New wizard...** in the File menu.
   - Type **newwiz** in the command window, then press Enter.
2. On the Create New Drawing dialog choose **Use a Wizard**, then click the **Next** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/NewWizard.png})
3. Choose between either **Imperial** or **Metric** default settings, then click the **Next** button. Select the **Unit of measurement** and the display **Precision**, then click the **Next** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h
4. Select the **Angle of measurement** and the display **Precision**, then click the **Next** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/SetAngleMeasurement.png})

Figure 1.385

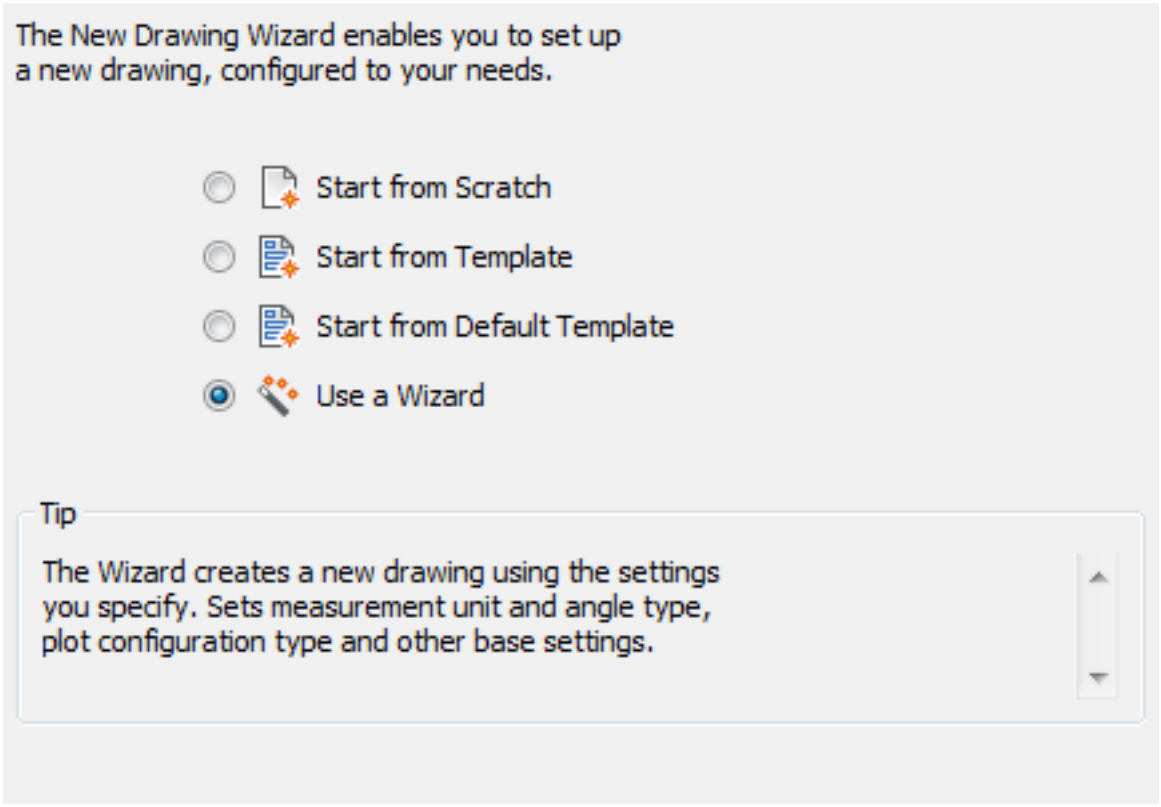


Figure 1.386

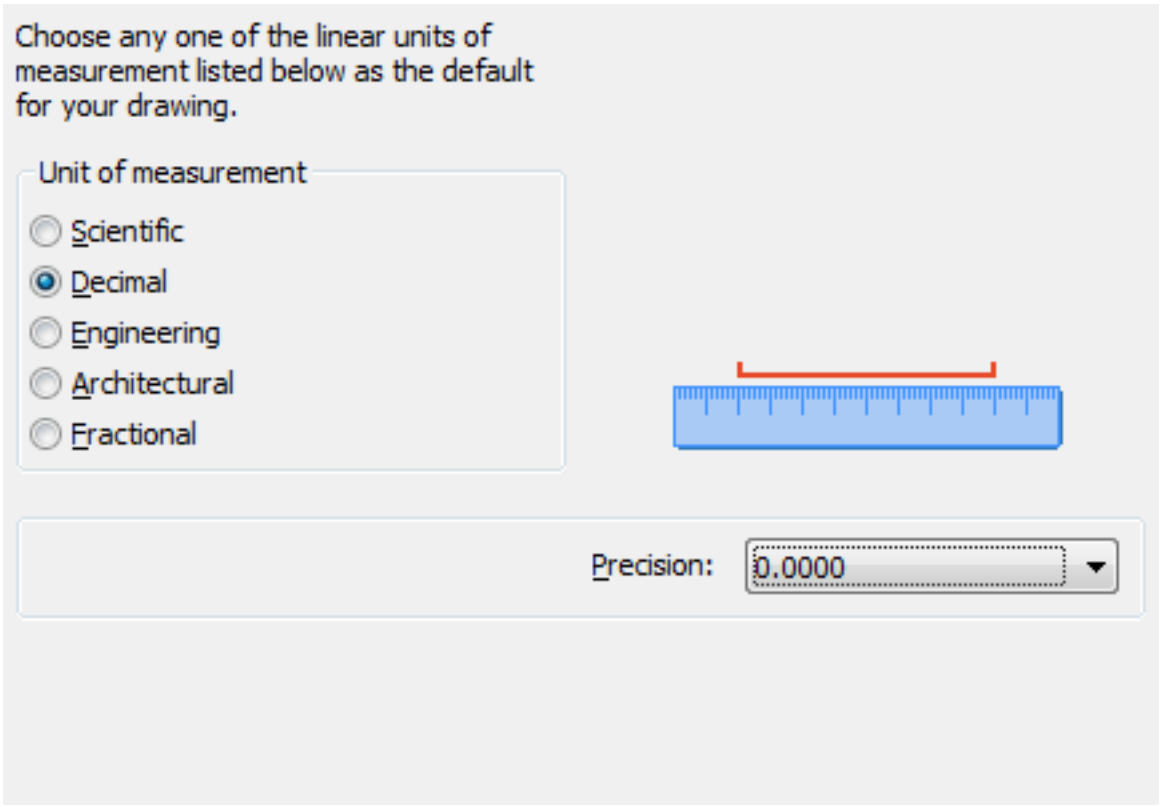
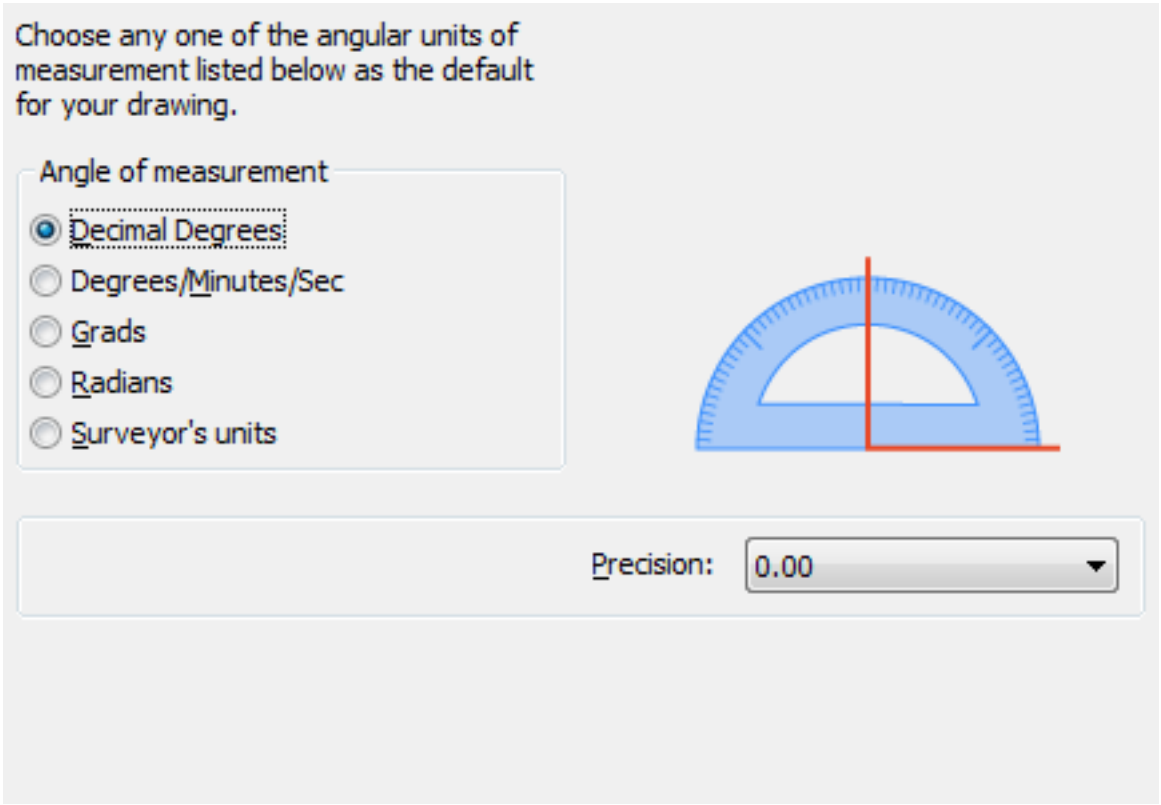


Figure 1.387



5. Choose between **Color Dependent** plot styles (CTB) or **Named** plot styles (STB), then click the **Next** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/SetPlotStyle.png})

6. Select the default **Color** and **Linetype**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Ma

7. Define the **Grid**, **Snap**, **UCS icon** and **Marker Blips** settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e

Figure 1.388

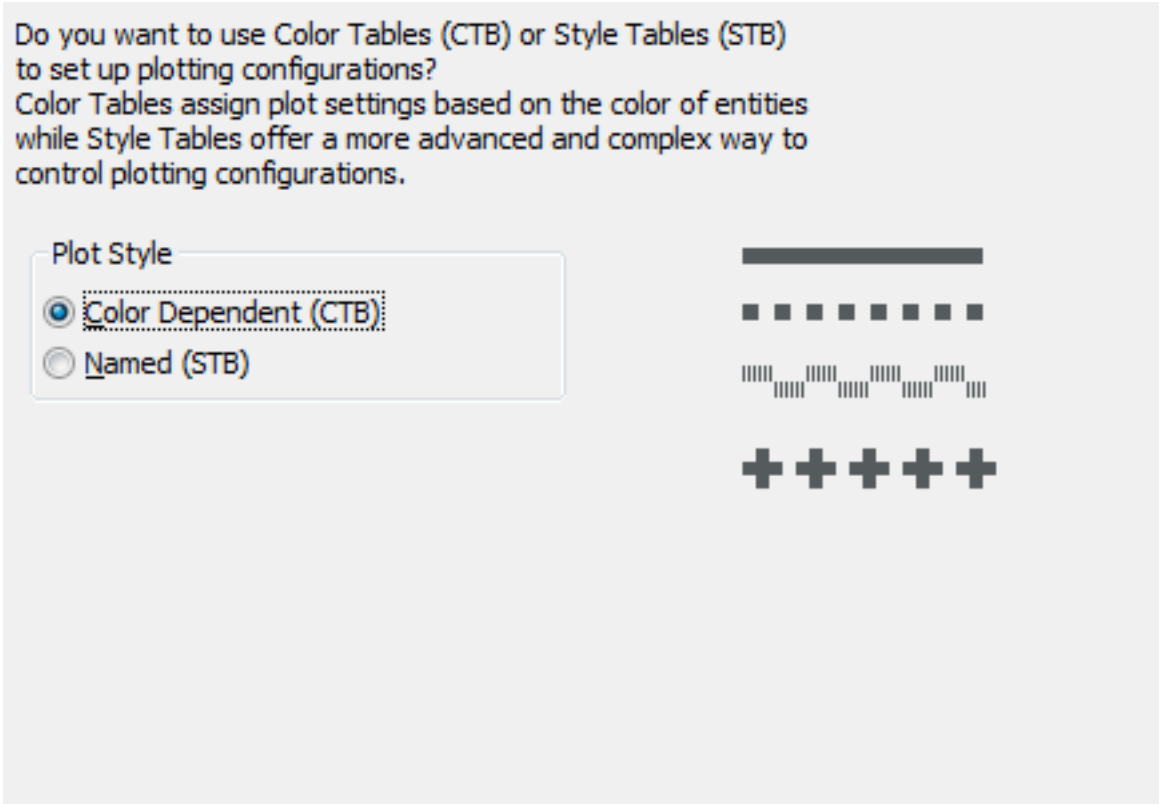
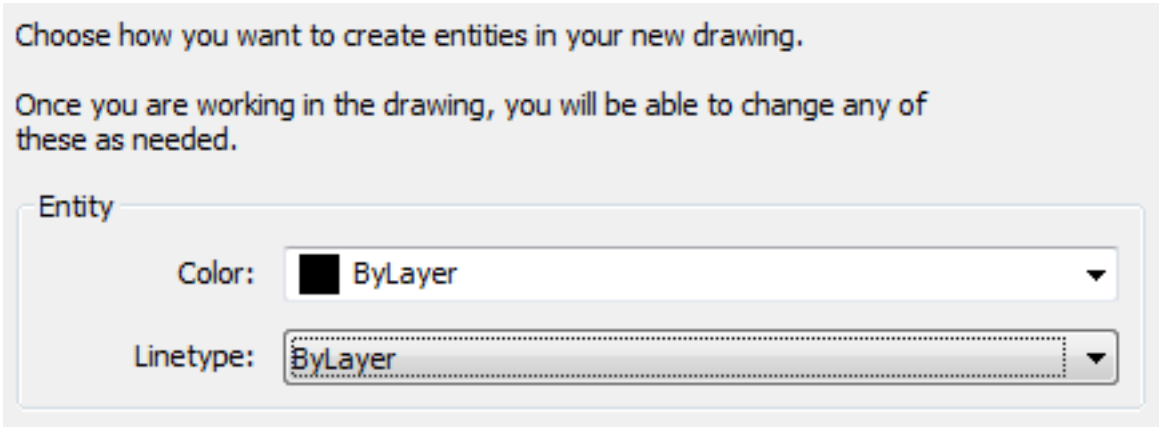


Figure 1.389



8. Click the **Finish** button to create the drawing. (option) Choose **Save As..** . in the File menu to save the drawing as a template in the **Templates** subfolder of the Local root (1.13.2) folder.

Figure 1.390

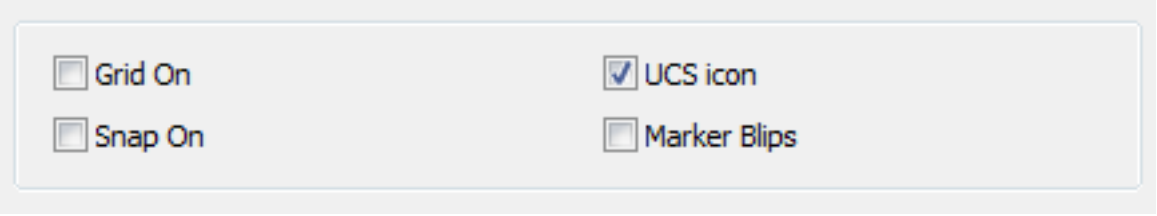
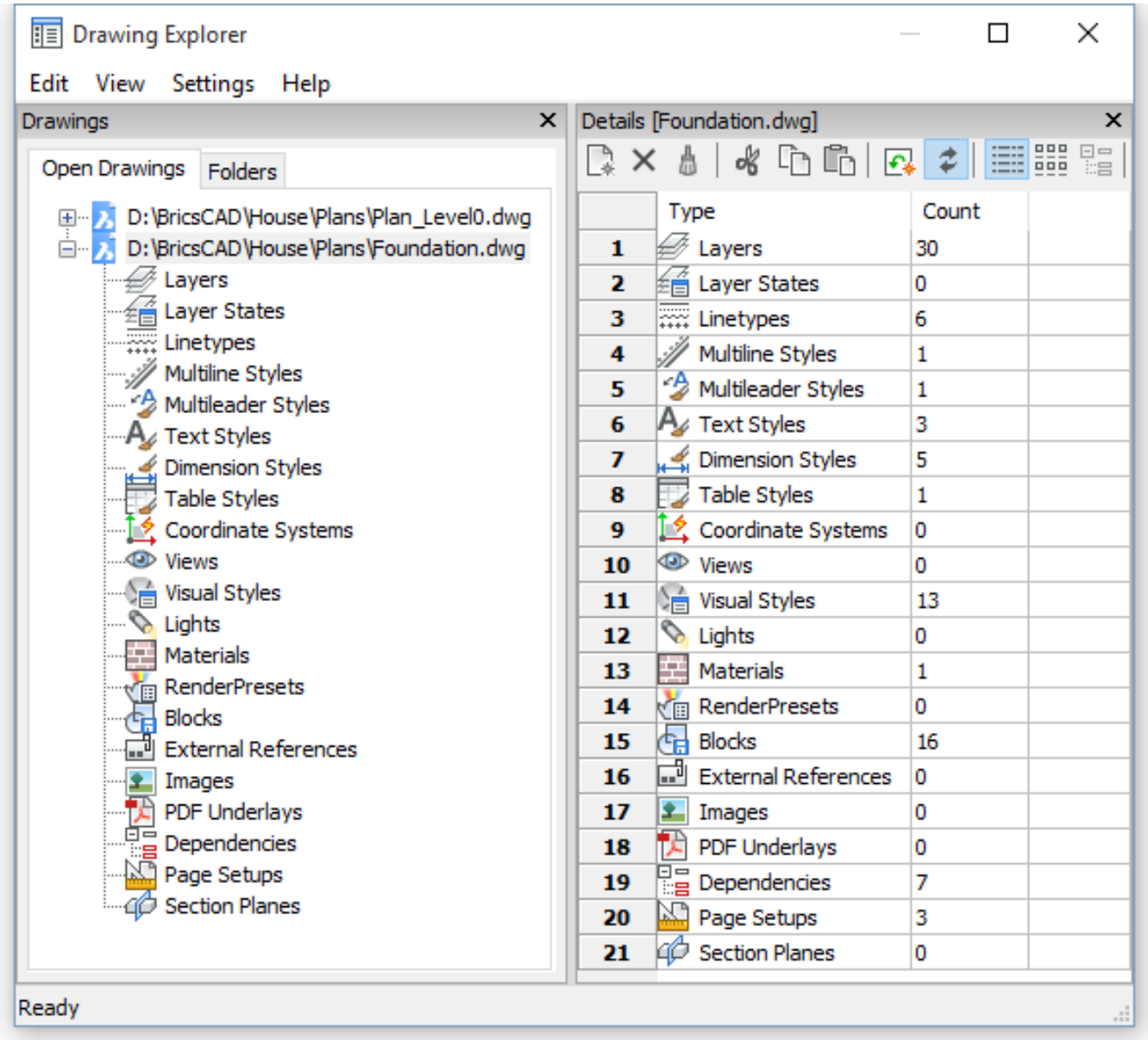


Figure 1.391



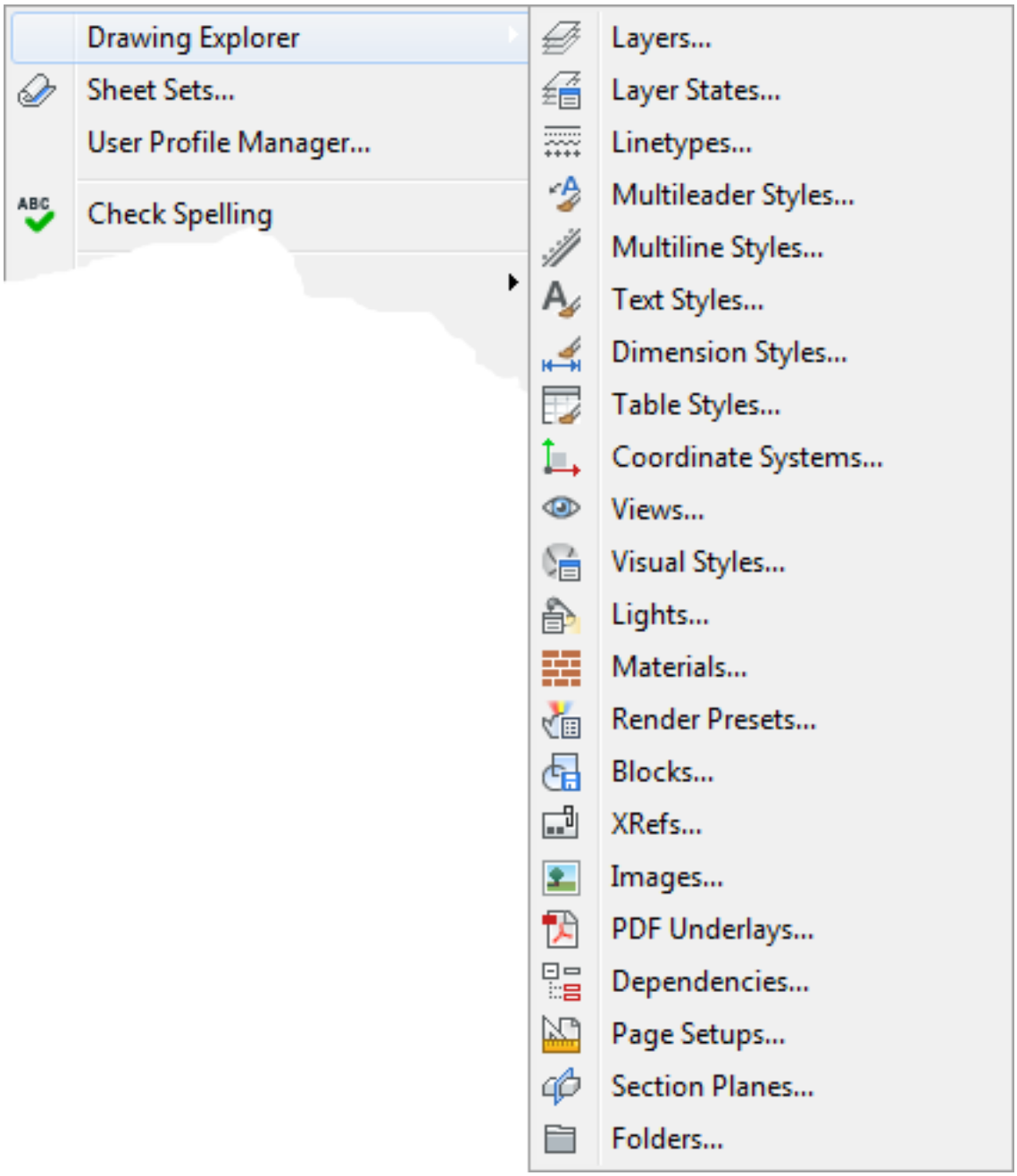
1.14.2 Drawing Explorer

Commands: Explorer (6.13.14), Layer (6.20.2), Layerstate (6.20.7), Linetype (6.20.33), MlStyle (6.21.27), MLeaderStyle (6.21.25), Style (6.27.53), DimStyle (6.11.20), TableStyle (6.29.6), ExpUcs (6.13.18), View (6.31.1), VisualStyles (6.31.13), LightList (6.20.29), Materials (6.21.11), RenderPresets (6.26.24), ExpBlocks (6.13.11), XRef (6.34.8), Image (6.17.3), Pdf (6.24.12), eTransmit (6.13.9),PageSetup (6.24.1), SectionPlaneSettings (6.27.13) The **Drawing Explorer** window consists of a number of sub-windows or panes:

- **Drawings**:
  - **Open Drawings**: a list of all drawings that are currently open.
  - **Folders**: your favorite drawing folders
- **Details**: the details of the selected drawing or the details of the selected category in a drawing, e.g. layers, blocks, images, ...
- **Preview**: a preview of the selected drawing, layer, block, view, dimension style, table style, XRef, image or material. Use the arrow keys to browse through the items.
- **Editor**: to define or edit the properties of **Multiline Styles**, **Multileader Styles**, **Table Styles**, **Views**, **Visual Styles**, **Sunlight settings**, **Materials**, **Render Presets** and **Section Planes**.

You can select which panes you want to have open or closed in the View (6.31.1) menu. BricsCAD remembers the display settings for each category. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/20

Figure 1.392

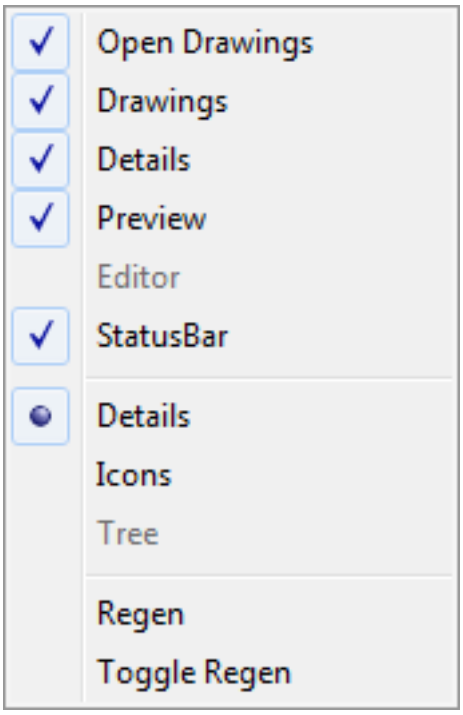


To open the Drawing Explorer

To open the Drawing Explorer window do one of the following:

- Click the **Drawing Explorer** tool button ( ) on the Standard toolbar. The Drawing Explorer window opens showing the details of the most recently used category.
- Click the **Layer**, **Linetype**, **Text Style** or **Dimension Style** field in the Status Bar. The Drawing Explorer window opens showing the details of the clicked definitions category in the current drawing. (See Customizing the Status Bar (1.12.2) to enable/disable fields in the Status Bar.)
- Choose **Drawing Explorer** in the Tools menu, then select a category in the flyout. The Drawing Explorer window opens showing the details of the selected category. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
- Type **explorer** in the command bar, then press Enter. The Drawing Explorer window opens showing the details of the most recently used category.

Figure 1.393



To control the Drawing Explorer layout

- In the View menu, check/uncheck the sub-window you want to open or close. (\vref {fig:C:/Dev/Task-2448/images/statich
- If you click a settings category in the Drawings sub-window when the >Details sub-window is closed, the Details window is re-opened, showing the content of the selected settings category.
- To close a sub-window, click the close button ( ) in its title bar.
- To restore the default layout choose **Restore Default Explorer Layout** in the Settings menu.

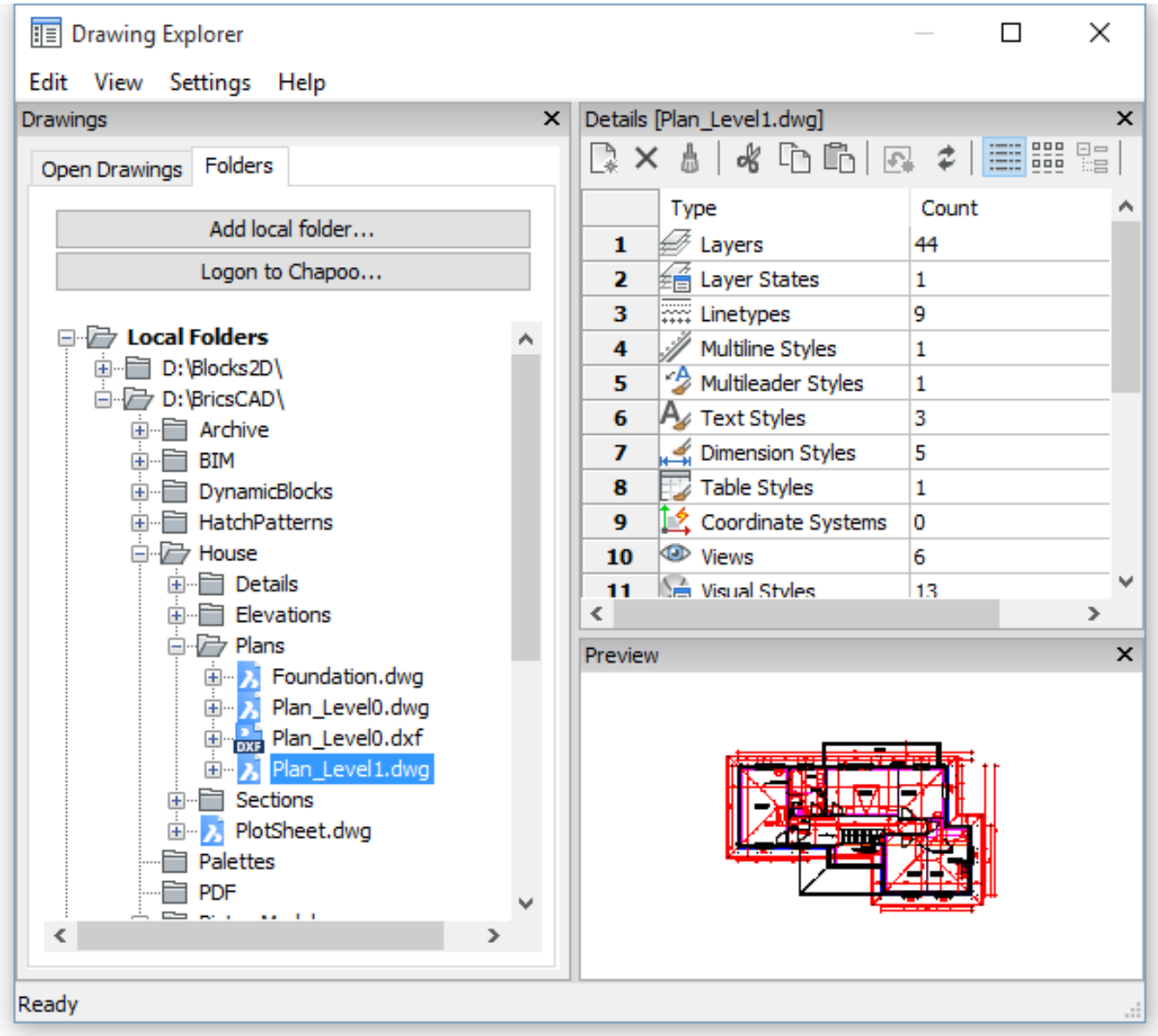
Opening a drawing

To open a drawing using the Drawing Explorer do the following:

1. Launch the Drawing Explorer.
2. In the **Drawings > Folders** sub-window browse to the folder of the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbric
3. Double click the drawing. The drawing opens, while the Drawing Explorer window stays open.
4. (optional) Repeat steps 2 and 3 to open more drawings.
5. Close the Drawing Explorer.

Note: When you select a drawing, a preview displays in the Preview sub-window.

Figure 1.394


Drawing Explorer
Edit View Settings Help
Drawings
Open Drawings
Folders
Add local folder...
Logon to Chapoo...
Local Folders
D:\Blocks2D\
D:\BricsCAD\
Archive
BIM
DynamicBlocks
HatchPatterns
House
Details
Elevations
Plans
Foundation.dwg
Plan_Level0.dwg
Plan_Level0.dxf
Plan_Level1.dwg
Sections
PlotSheet.dwg
Palettes
PDF
Details [Plan_Level1.dwg]
Type
Count
1 Layers 44
2 Layer States 1
3 Linetypes 9
4 Multiline Styles 1
5 Multileader Styles 1
6 Text Styles 3
7 Dimension Styles 5
8 Table Styles 1
9 Coordinate Systems 0
10 Views 6
11 Visual Styles 13
Preview
Ready

Adding a drawing folder

1. In the **Drawings > Folders** sub-window of the Drawing Explorer main window click the **Add local folder...** button. The Browse For Folder window opens.
2. In the Browse For Folder window do one of the following:
   - To add an existing folder: select the folder you want to add.
   - To create a new folder: click the **Make New folder** button.
3. Click the **OK** button. The folder is added.

Notes

- Drawings in the subfolders of the selected folder can be opened too.
- The parent folder(s) of the selected folder is (are) greyed out in the drawing folder tree. Drawings in such folders cannot be opened.

To insert a block from an unopened drawing

The Block Manager (6.13.14) feature of the Drawing Explorer allows to insert blocks in the current drawing from unopened drawings.

To copy definitions between drawings

1. Open both the source and the target drawing.
2. Launch the Drawing Explorer (1.14.2).
3. In the source drawing select the category you want to copy items from, e.g. **Blocks**.
4. (option) In the Details toolbar, click the **Icon View** button ( )
5. Select the item(s) you want to copy, e.g. a series of blocks. Press and hold the Ctrl key to select multiple items.
6. Press and hold the left mouse button to drag the selection set to the **Open Drawings** section of the Drawing Explorer.
7. Release the left mouse button when on the target drawing name. The selected items are copied to the target drawing. You will be prompted to overwrite items of the same name.

To merge the content of two layers

1. Open the Drawing Explorer > (1.14.2) Layers dialog window.
2. Select the layer you want to merge into another layer.
3. Choose **Merge to...** in the Edit menu. The Merge Layers dialog box displays.
4. Select the layer you want to merge to. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/so
5. Click the **Merge** button. The content of the first layer is merged to the second layer. The first layer is purged from the drawing.

Tutorial: Tips and Tricks for Drawing Explorer Click here (1.2.6) to watch

Figure 1.395

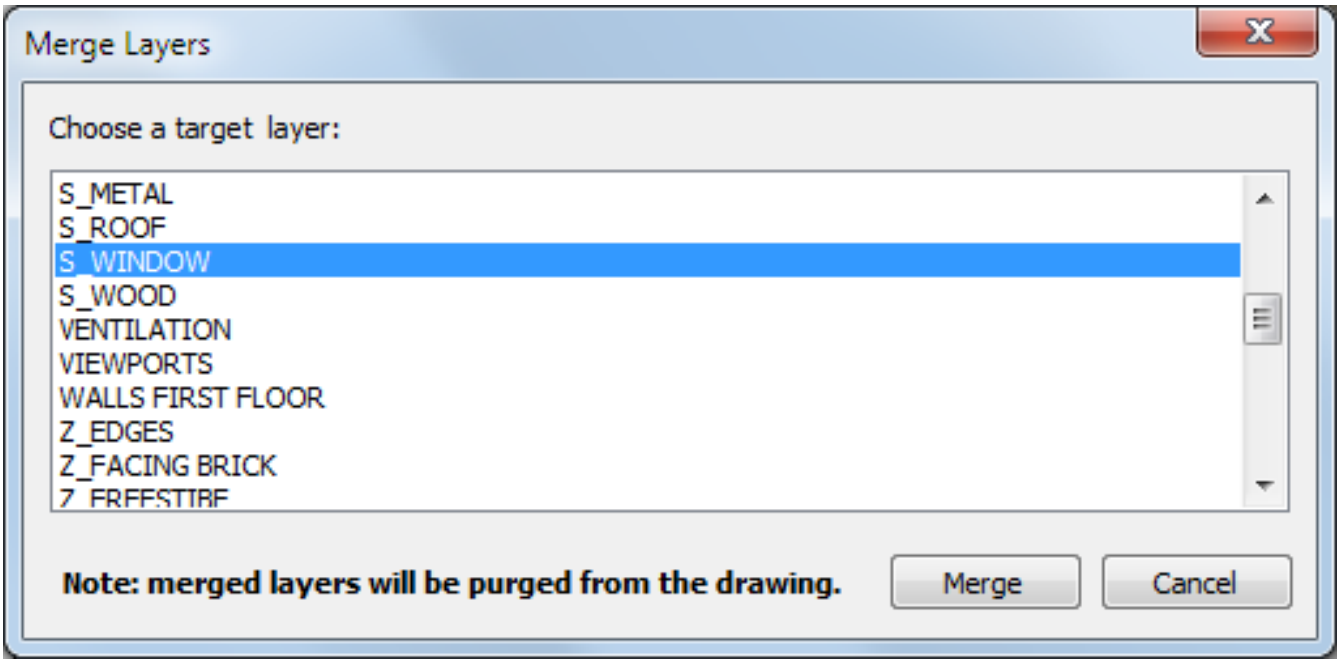
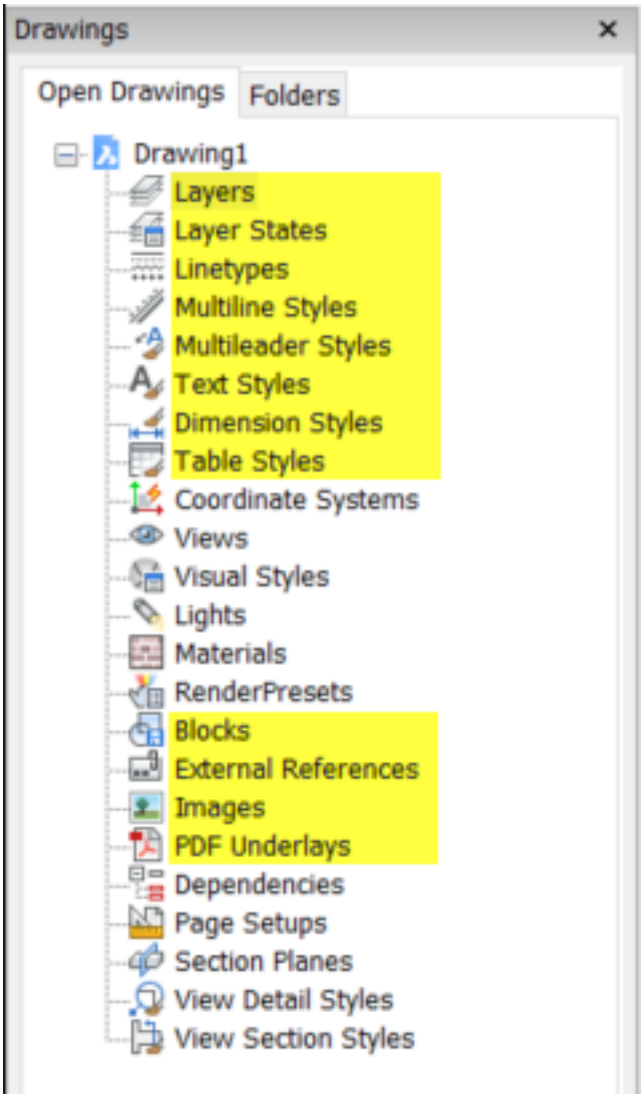

Merge Layers
Choose a target layer:
S_METAL
S_ROOF
S_WINDOW
S_WOOD
VENTILATION
VIEWPORTS
WALLS FIRST FLOOR
Z_EDGES
Z_FACING BRICK
Note: merged layers will be purged from the drawing.
Merge
Cancel

Figure 1.396



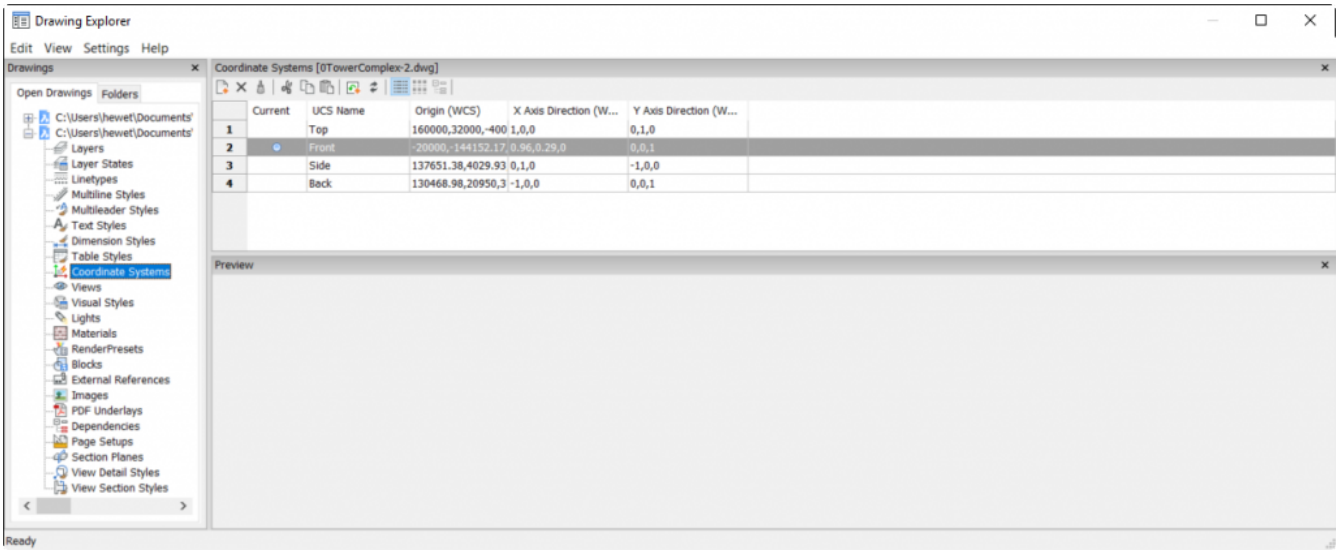
1.14.3 Exploring the Drawing Explorer

Tutorial: Tips and Tricks for Drawing Explorer Click here (1.2.6) to watch (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc
Content00.png})

Coordinate systems

The Coordinate Systems panel displays all the named User Coordinate Systems (UCS) in the current drawing. If you choose the option to add a new coordinate system, the UCS command is launched enabling you to choose from options such as picking points or selecting an entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Usr
Content01.png})

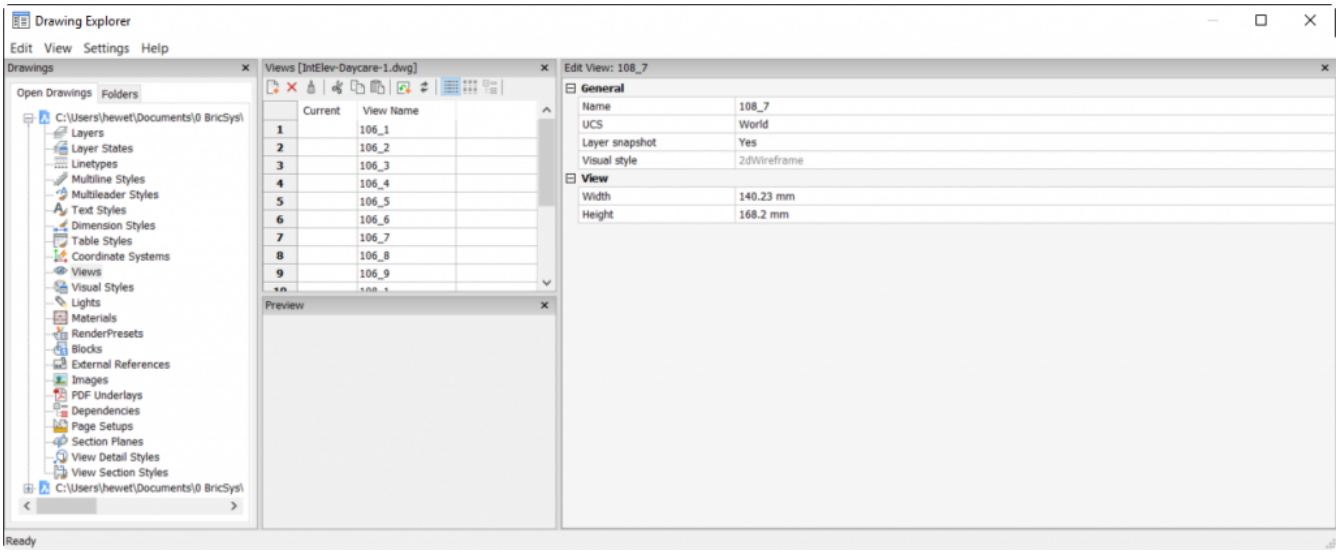
Figure 1.397



Views

The Views panel displays all the named views in the drawing. If you choose the option to add a new view, the VIEW command is launched. It allows you to save the current view or pick a window to define the view. You can view and edit properties for a selected view in the Edit View panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 Content02.png})

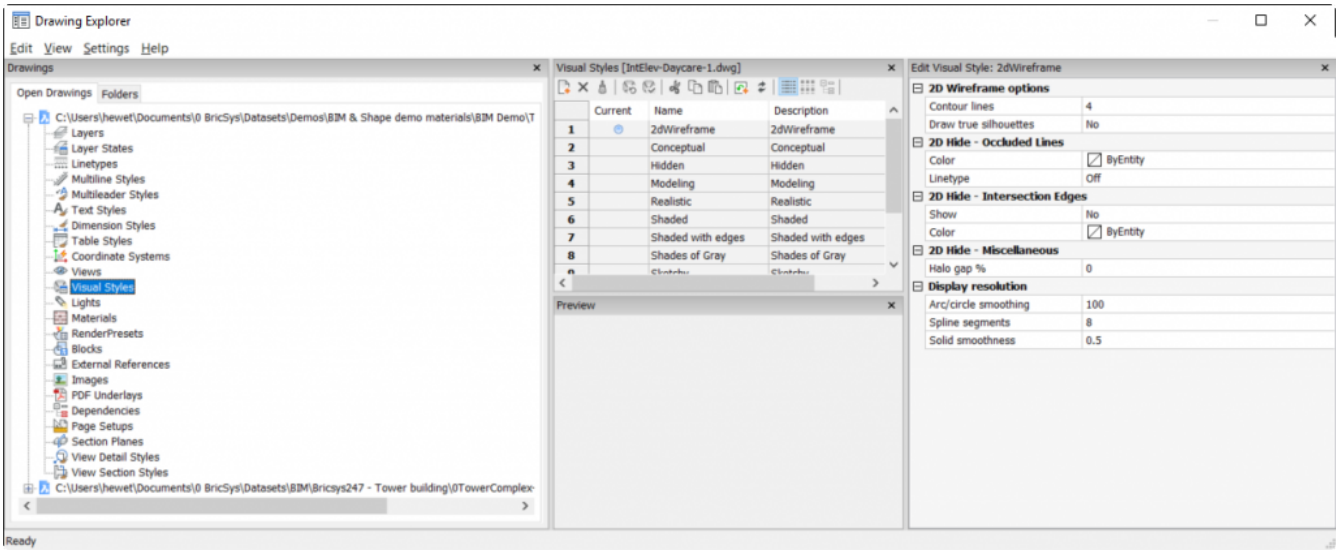
Figure 1.398



Visual Styles

The Visual Styles panel displays all the visual styles in the drawing. It includes 11 predefined visual styles and you can easily add your own. You can view and edit visual style properties in the Edit Visual Style panel. While most visual styles are intended for 3D visualization, some of them, such as Sketchy, can be useful for 2D design. Therefore, visual styles are fully supported in all editions of BricsCAD, including Classic. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/DrawingExplorer-Content03.png})

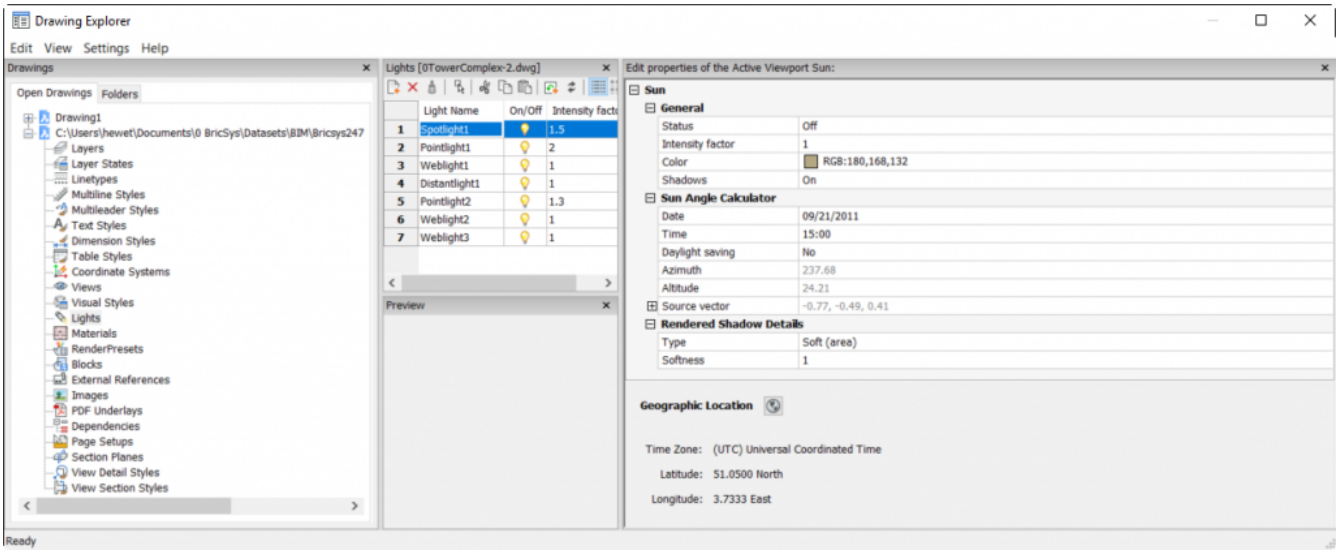
Figure 1.399



Lights

The Lights panel displays all the lights in the drawing. If you choose the option to add a new light, it launches the LIGHT command where you can select from Point, Spot, Web, or Distant lights. You can view and edit sun properties in the Edit Properties of the Active Viewport Sun panel. That panel also provides access to the geographic location dialog box. While you can create and modify lights in BricsCAD Classic, it doesn't support rendering. Therefore, the usefulness of lights in BricsCAD Classic is somewhat limited. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/DrawingExplorer-Content04.png})

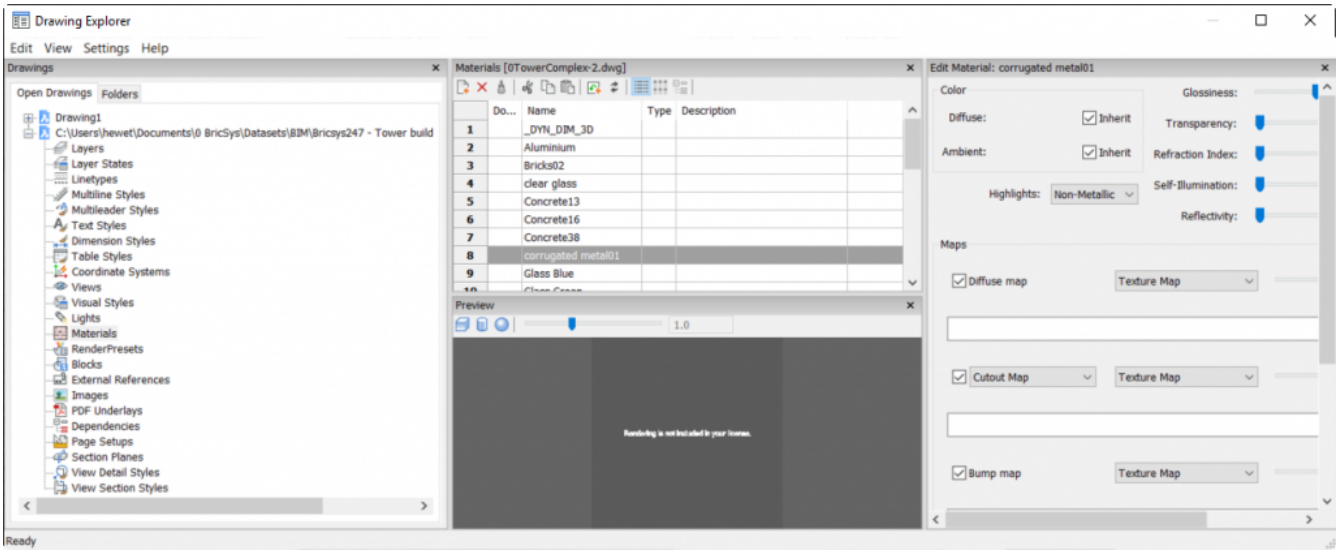
Figure 1.400



Materials

The Materials panel displays all the materials in the drawing. You can view and edit Material properties in the Edit Material panel. Like lights, however, material functionality is limited in the BricsCAD Classic edition because it doesn't support rendering. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawingsContent05.png})

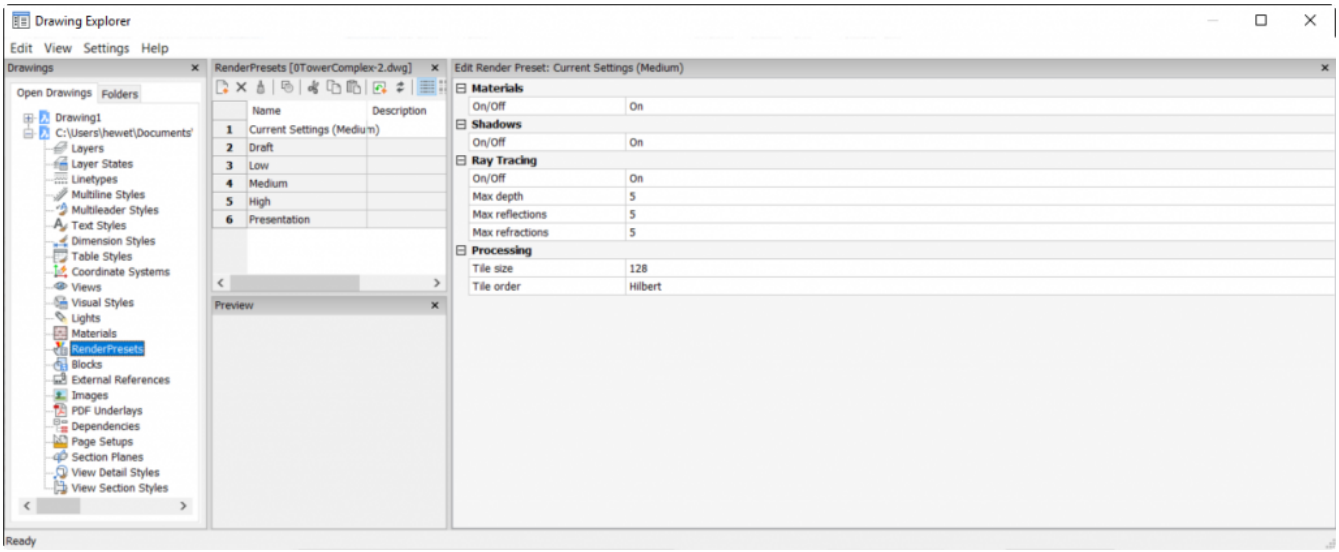
Figure 1.401



Render Presets

The Render Presets panel displays all the Render Presets in the drawing. It includes five pre-defined settings and you can easily add your own. While you can create and modify render presets in BricsCAD Classic, it doesn't support rendering. As a result, its usefulness in the Classic edition is limited. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/Content06.png})

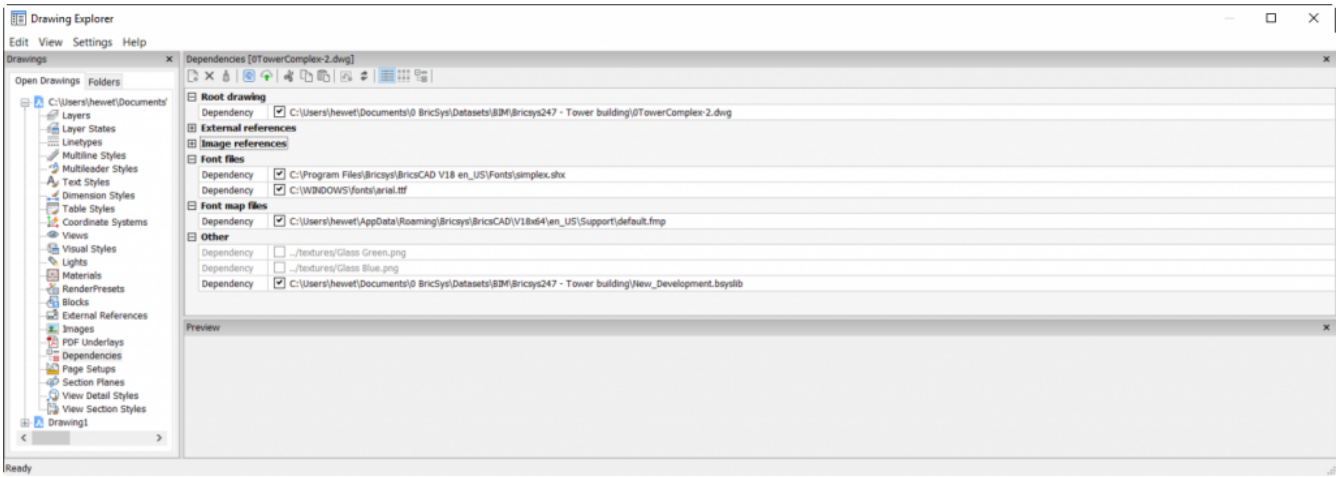
Figure 1.402



Dependencies

Dependencies in BricsCAD can best be compared to transmittal sets in AutoCAD®. In fact, if you launch the ETRANSMIT command, BricsCAD opens Drawing Explorer with the Dependencies panel displayed. There you can view all the dependent files including xrefs, images, fonts and more. Select which files to include in the transmittal set and then save them to a zip file or folder. You can also upload them to Bricsys 24/7 cloud storage. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/DrawingExplorer-Content07.png

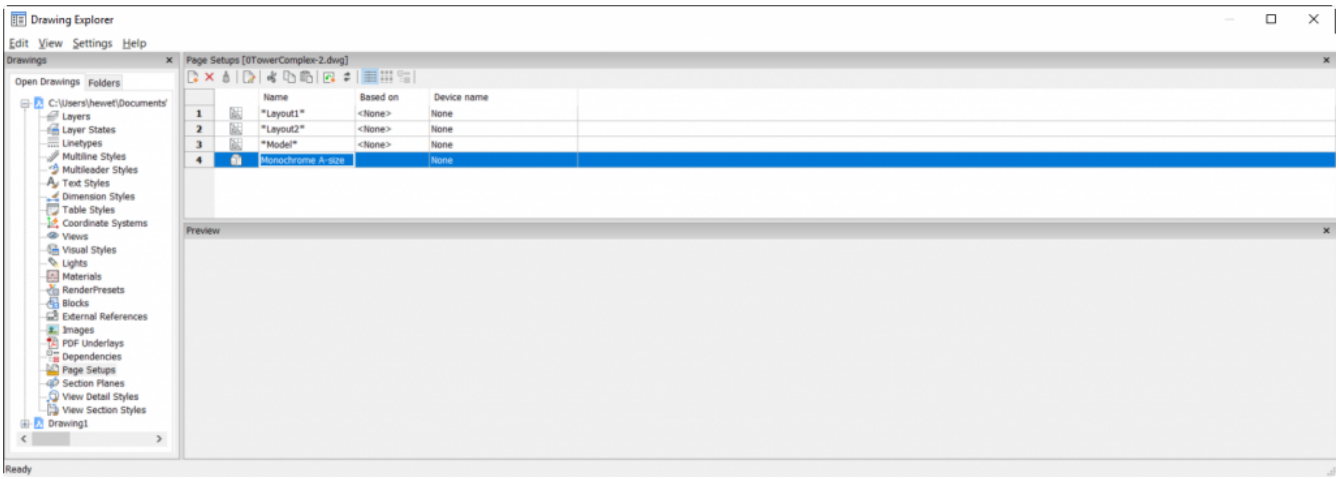
Figure 1.403



Page Setups

The Page Setups panel displays all the page setups in the drawing. If you choose the option to add a new page setup, it displays the Page Setup dialog box with its familiar controls. You can also select and edit any existing page setups. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/DrawingExplorer-Content08.png})

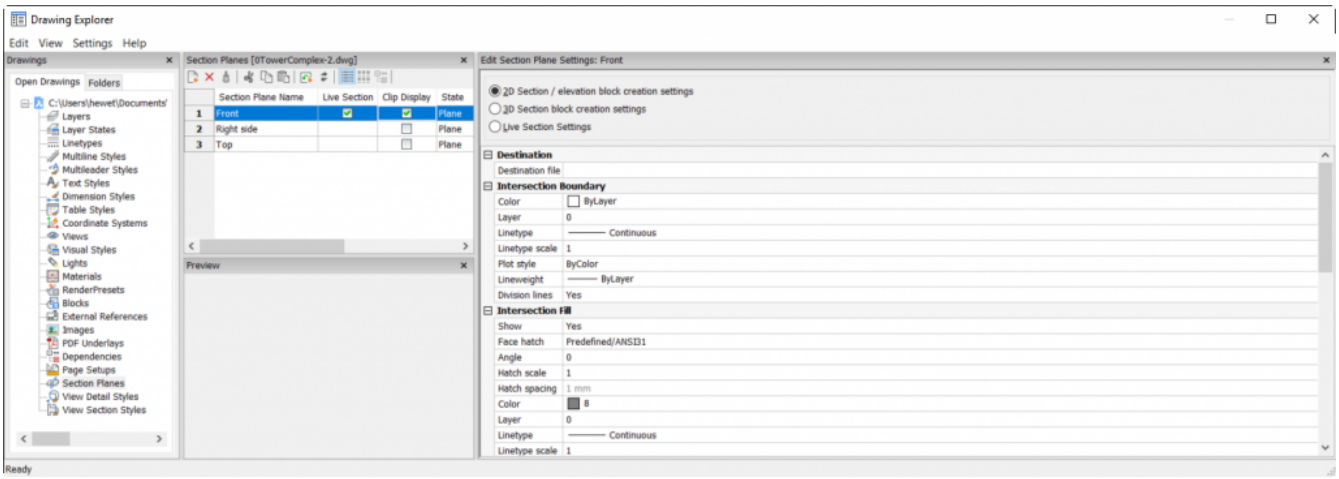
Figure 1.404



Section Planes

The Section Planes panel displays all the section planes in the drawing. If you choose the option to add a new section plane, it launches the SECTIONPLANE command where you can select a method, such as Orthographic, to define the new section plane. Easily view and modify properties of a section plane in the Edit Section Plane Settings panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/DrawingExplorer-Content09.png})

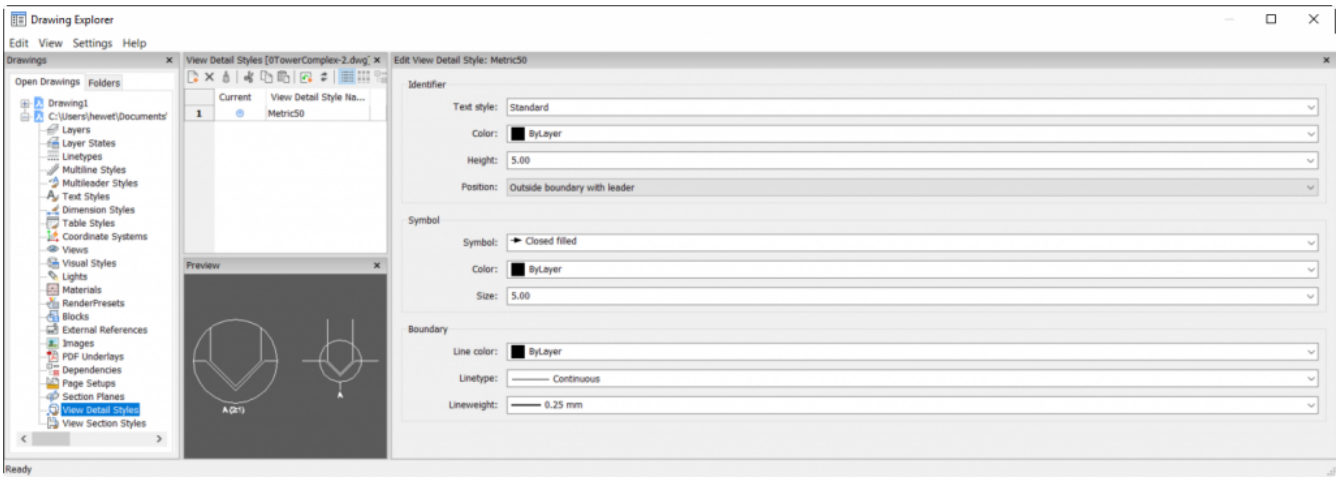
Figure 1.405



View Detail Styles

The View Detail Styles panel displays all the view detail styles in the drawing. Easily view and modify properties of a selected view detail style in the Edit View Detail Style panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/Content10.png})

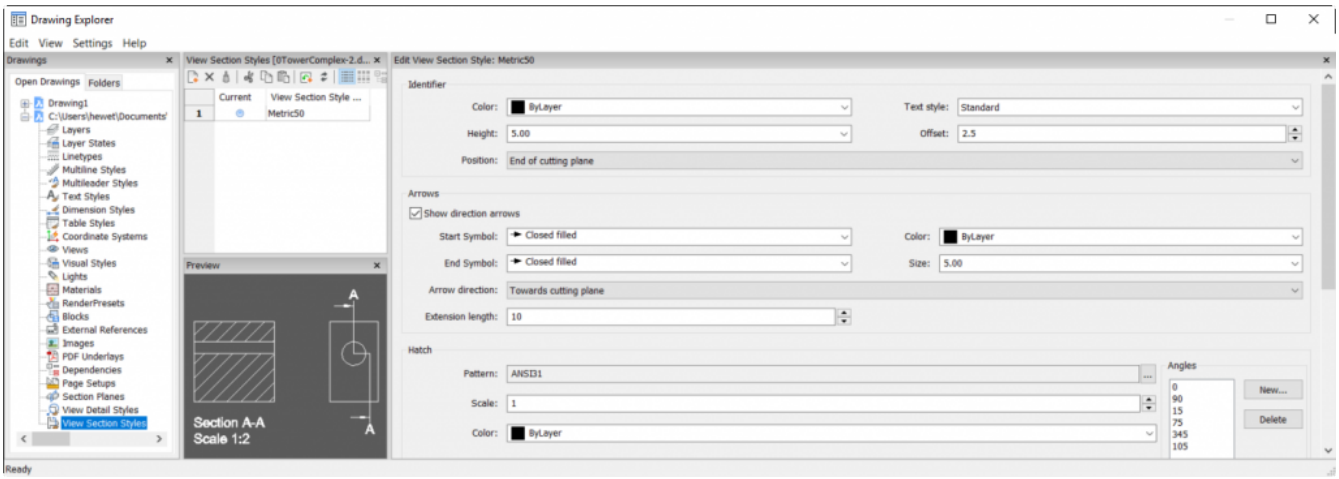
Figure 1.406



View Section Styles

The View Section Styles panel displays all the view section styles in the drawing. Easily view and modify properties of a selected view section style in the Edit View Section Style panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/helContent11.png})

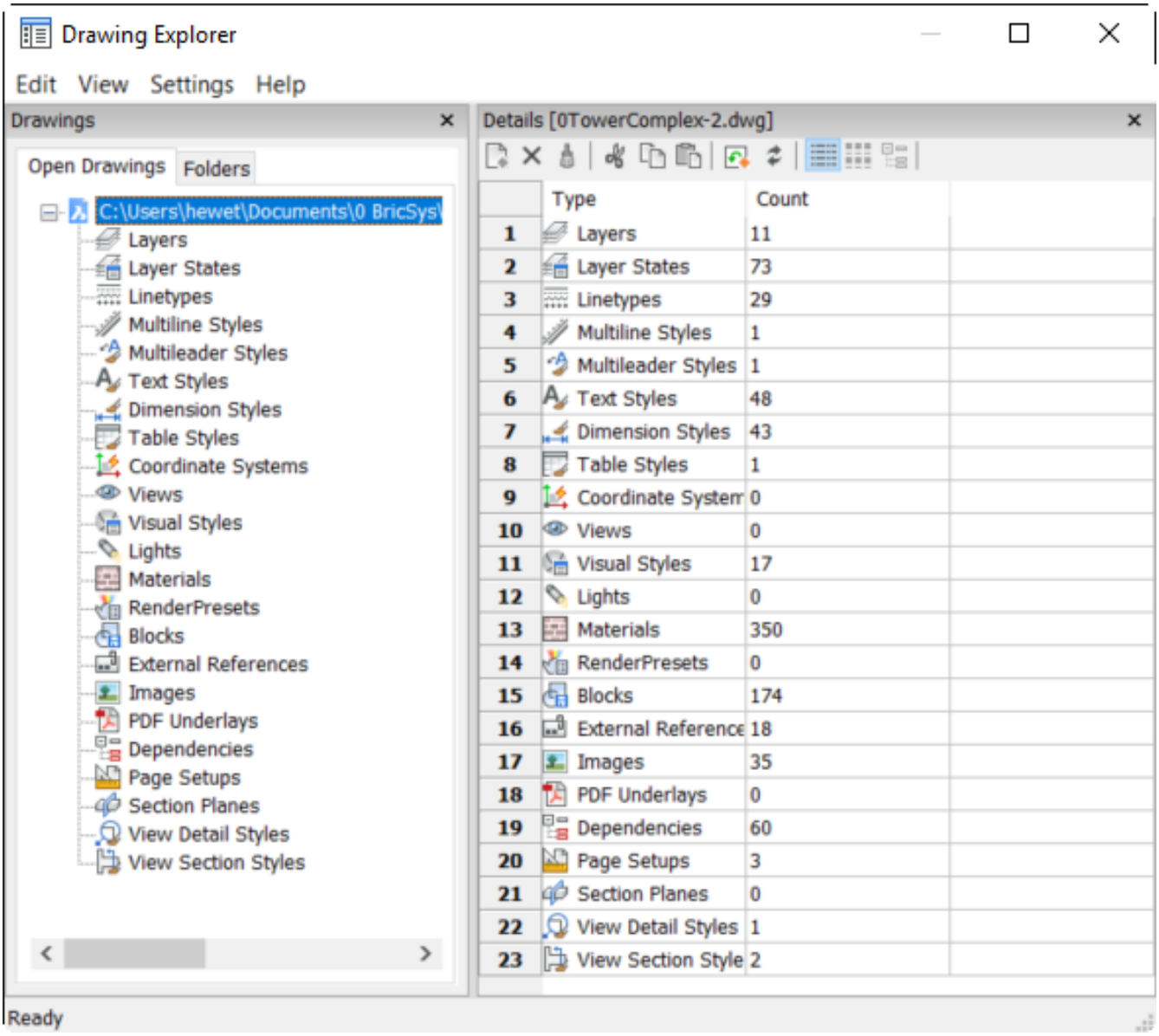
Figure 1.407



Drawing Content

In addition to being able to select and modify each of the content types in Drawing Explorer, you can select the drawing itself. Doing so displays the number of occurrences of each type of content in the drawing. If you double-click on one of the items in the Details panel, that drawing content is displayed. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/Content12.png})

Figure 1.408

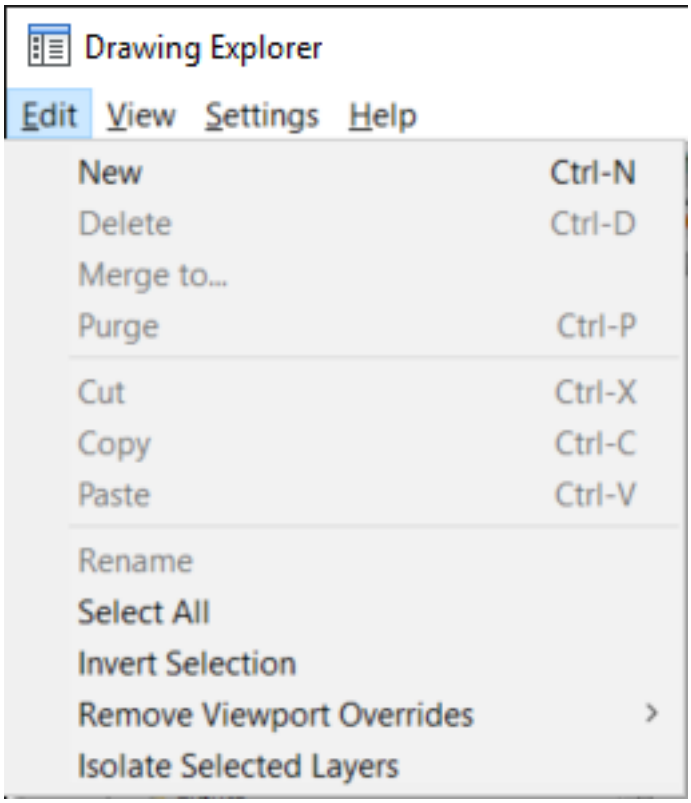

Drawing Explorer
Edit View Settings Help
Drawings
Open Drawings Folders
C:\Users\hewet\Documents\0 BricSys\
Layers
Layer States
Linetypes
Multiline Styles
Multileader Styles
Text Styles
Dimension Styles
Table Styles
Coordinate Systems
Views
Visual Styles
Lights
Materials
RenderPresets
Blocks
External References
Images
PDF Underlays
Dependencies
Page Setups
Section Planes
View Detail Styles
View Section Styles
Details [0TowerComplex-2.dwg]
Type Count
1 Layers 11
2 Layer States 73
3 Linetypes 29
4 Multiline Styles 1
5 Multileader Styles 1
6 Text Styles 48
7 Dimension Styles 43
8 Table Styles 1
9 Coordinate System 0
10 Views 0
11 Visual Styles 17
12 Lights 0
13 Materials 350
14 RenderPresets 0
15 Blocks 174
16 External Reference 18
17 Images 35
18 PDF Underlays 0
19 Dependencies 60
20 Page Setups 3
21 Section Planes 0
22 View Detail Styles 1
23 View Section Style 2
Ready

Menus

Drawing Explorer includes four menus: Edit, View, Settings and Help. Many of the options in these menus are also available from the tools and right-click menus in the panels. Others are only available from these menus.

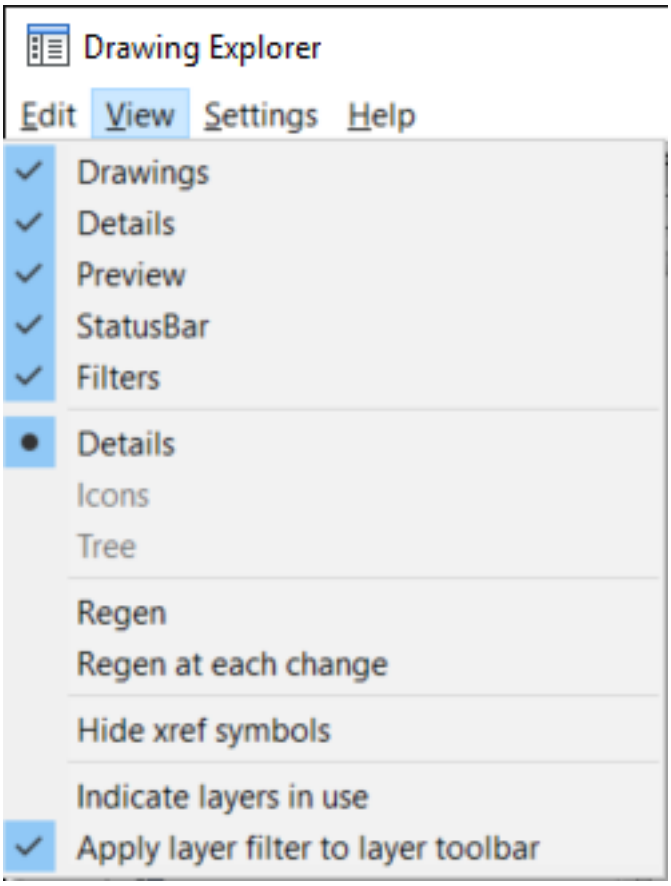
Edit The tools in the Edit menu can vary depending which type of content you select. These tools are also available in the various panels. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/DrawMenus01.png})

Figure 1.409



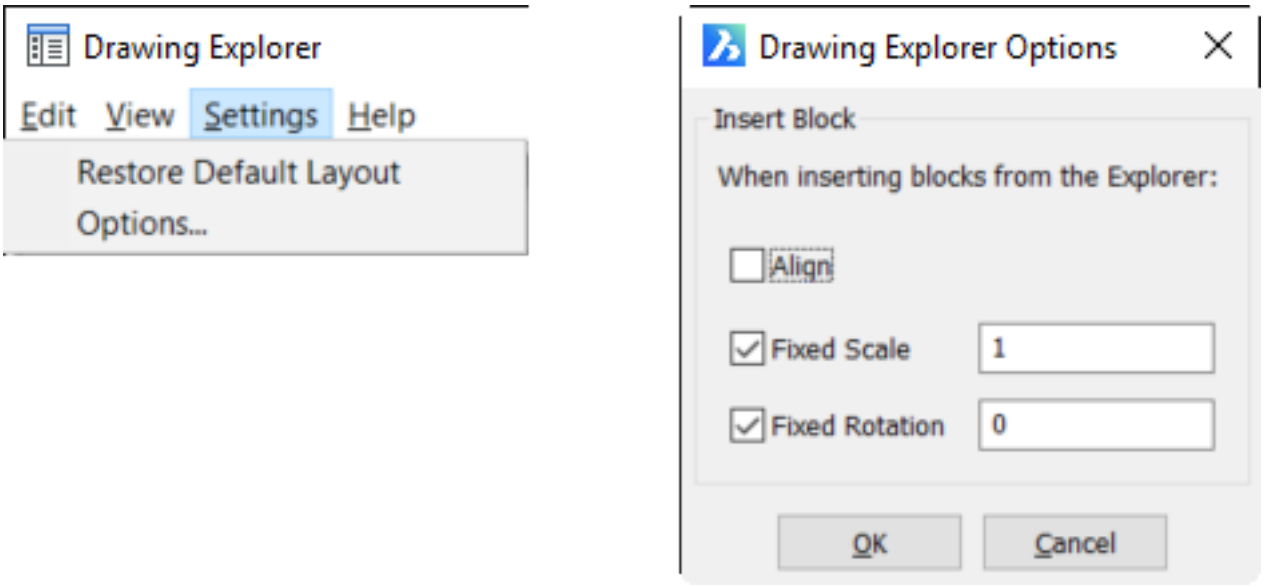
View The View menu enables you to control which panels display for each type of content. You can also specify if you want to display the content details, icons, or tree view. The Regen tools allow you to specify, for each type of content, whether you want the drawing area to regen each time you make a change to that content or if you want to manually regenerate. Hide xref symbols is relevant for content, such as layers or text styles, that may be listed as part of an xref. If you choose to hide xref symbols, the reference content does not display in the list. Most of the tools in the View menu are available for all the content types. However, a few tools are added for specific content. For example, the last two options in the following View menu are only available for Layers. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/DrawingExplorer-Menus02.png})

Figure 1.410



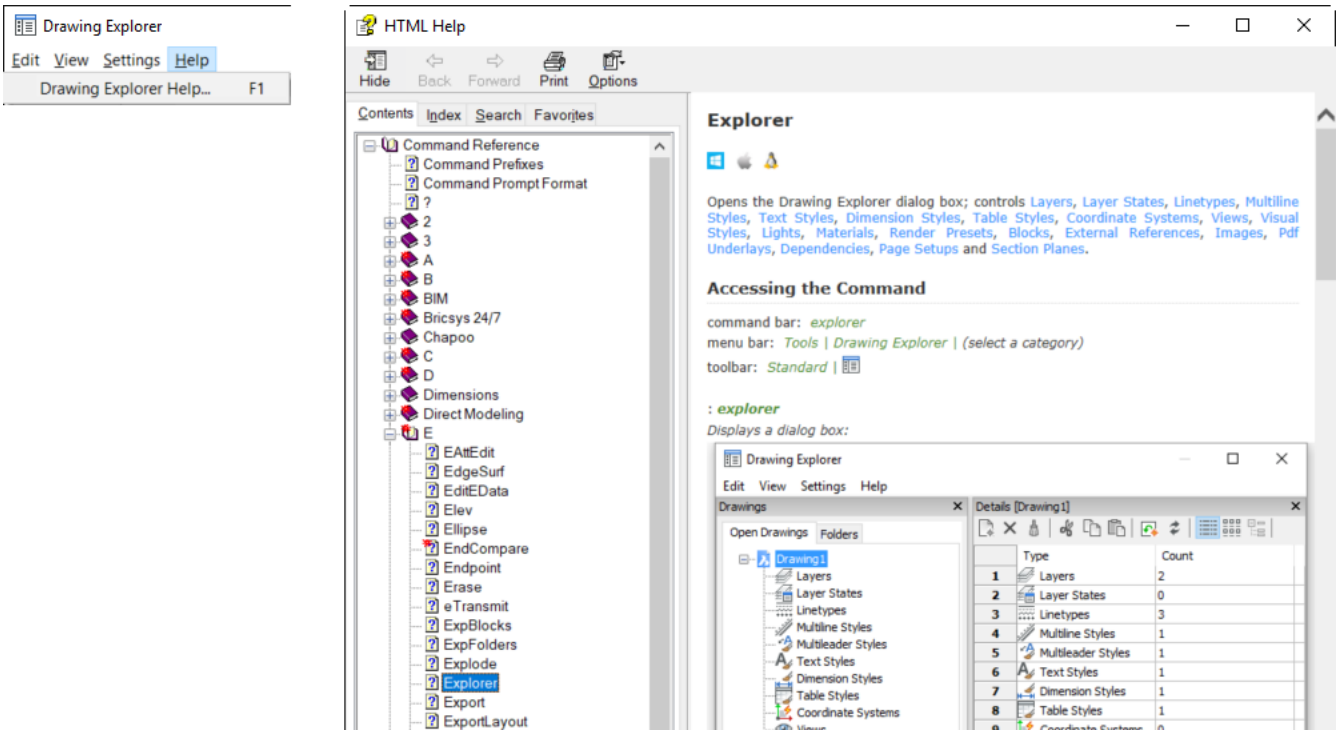
Settings The Settings menu has only two tools. The first one enables you to restore the layout of the selected content type back to the default settings. The reset will not affect other content types. The Options tool displays a dialog box that controls how blocks are inserted using Drawing Explorer. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hMenus03.png})

Figure 1.411



Help The Help tool opens the BricsCAD Help window with the Explorer help content displayed. BricsCAD Help is also available online! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/DravMenus04.png})

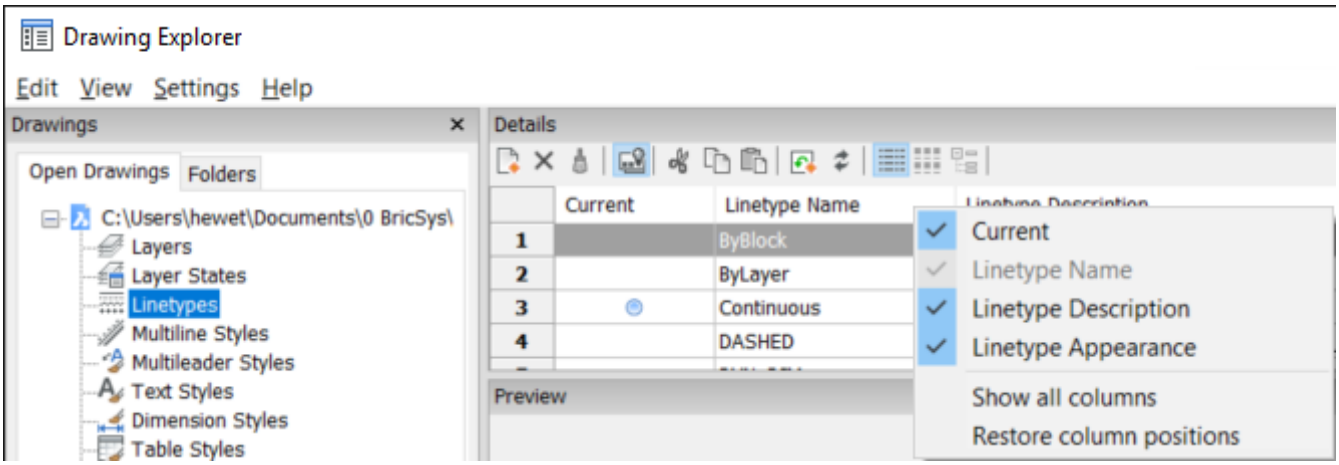
Figure 1.412



Column Controls

You can control the display of the columns for any named content by right-clicking on a column header or row number. Turn columns on or off individually or choose the option to Show all columns. You can drag and drop columns to rearrange them and easily restore them to their default positions. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/Columns01.png})

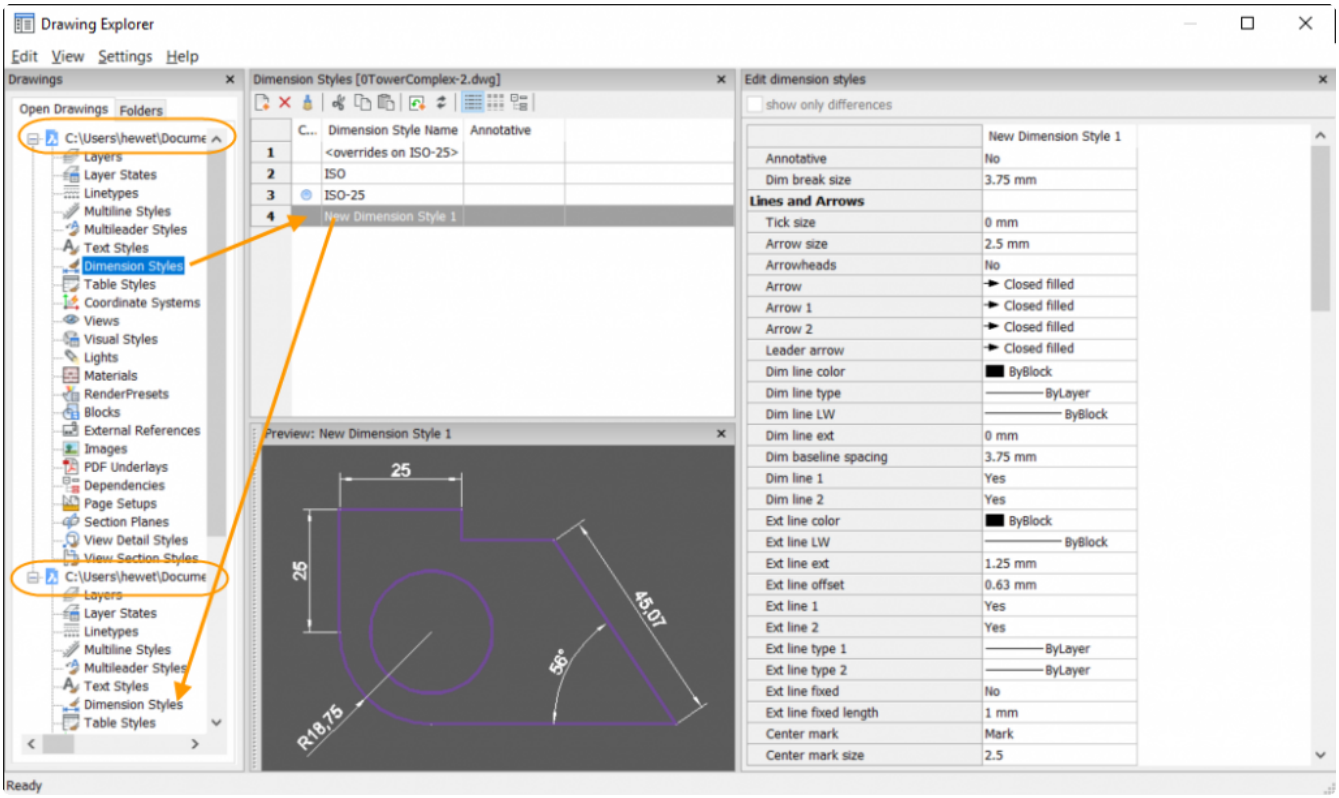
Figure 1.413



Open Drawings

Until now, we've focused on the named content of a single drawing. However, the Open Drawings tab in Drawing Explorer, lists all the open drawings enabling you to access named content from any of them. You can easily drag and drop or copy and paste named content from one open drawing to another! While drag and drop works for most of the content, it doesn't work for all. If drag and drop doesn't work for you, try copy and paste from the Edit or right-click menus. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/DrawingExplorer-OpenDrawings01.png})

Figure 1.414

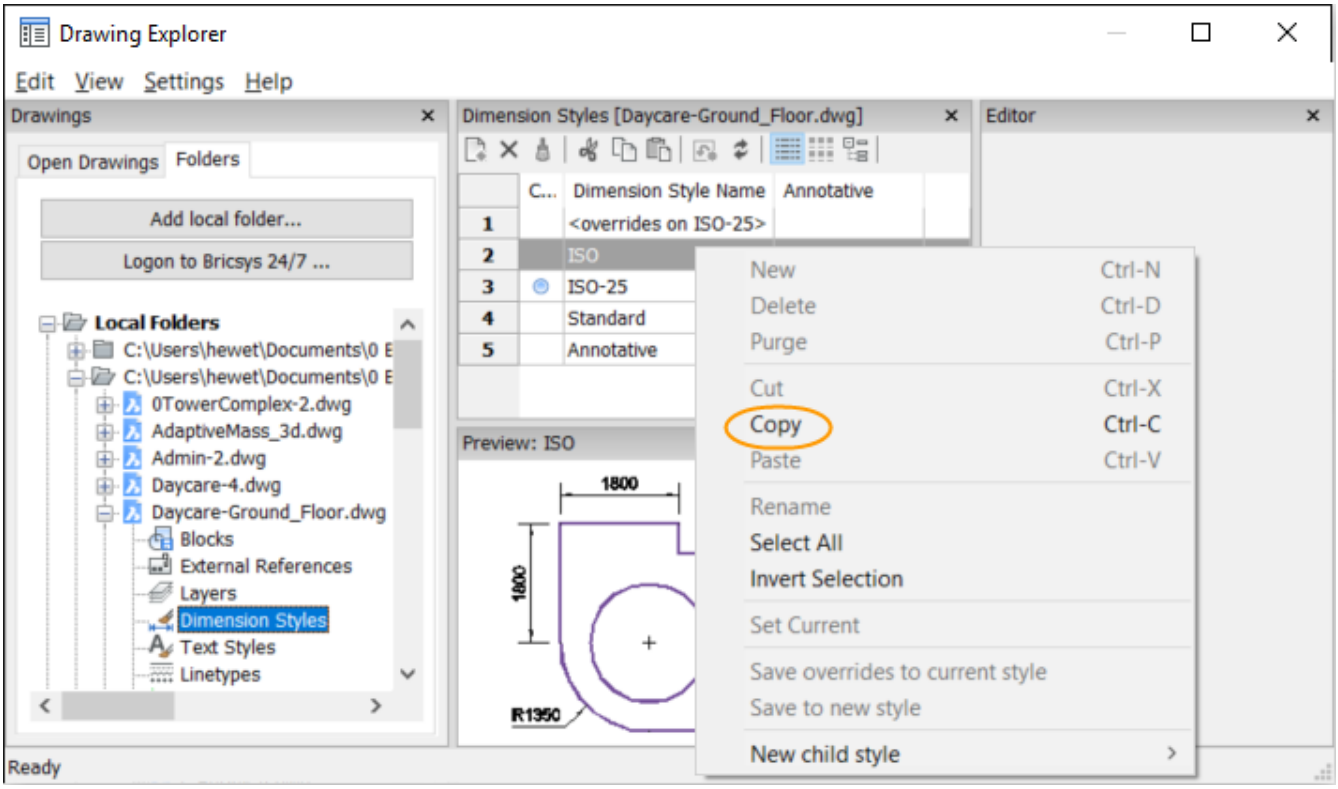


Folders

The Folders tab, to the right of the Open Drawings tab, enables you to access drawings from local folders and even from Bricsys 24/7.

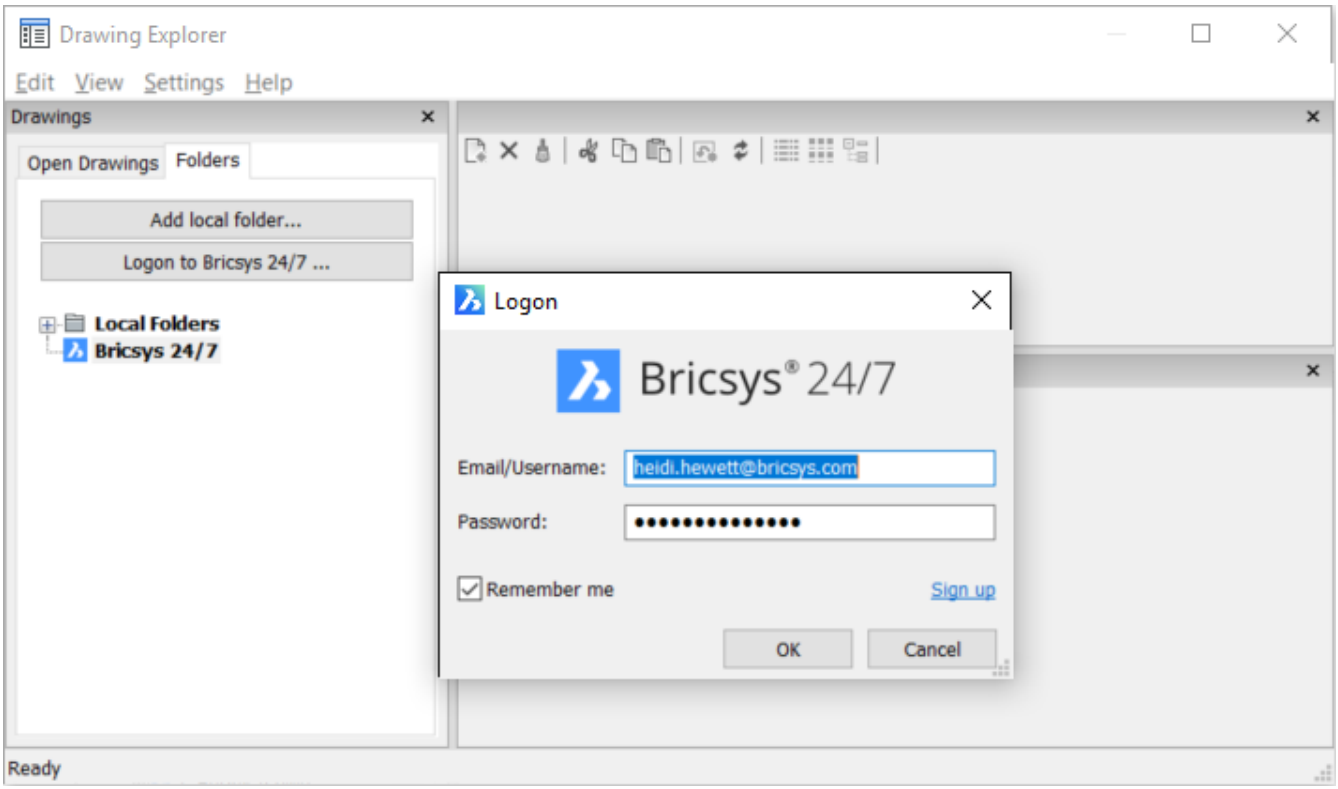
Local Folders Add local folders to Drawing Explorer for easy access to named drawing content even if the drawings aren't open! Simply expand the drawing and select the content. Then copy it from that drawing and paste it into an open drawing. Voila! It's that easy! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Managin Folders01.png})

Figure 1.415



Bricsys 24/7 Bricsys 24/7 offers secure and simple document management. You can Access your documents from anywhere and on any device. Share them with your global teams for secure project collaboration. And, the best part is that you access the drawings you store on Bricsys 24/7 directly from BricsCAD! From the Folders tab in Drawing Explorer, choose the option to Logon to Bricsys 24/7. If you don't yet have an account, choose Sign up to start your free trial. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ManagingDrawings/DrawingExplorerFolders02.png})

Figure 1.416


Drawing Explorer
Edit View Settings Help
Drawings
Open Drawings
Folders
Add local folder...
Logon to Bricsys 24/7 ...
Local Folders
Bricsys 24/7
Logon
Bricsys® 24/7
Email/Username:
heidi.hewett@bricsys.com
Password:
Remember me
Sign up
OK
Cancel
Ready

1.14.4 Drawing Properties

Command: DWGPROPS (6.10.42) The **Dwgprops** command opens the **Drawing Properties** dialog box, showing the general information and user defined properties stored with a drawing. Drawing properties include: general information, summary, statistics and user defined custom properties. Drawing properties can be referenced in a field (1.21.23) in multiline text entities or tables.

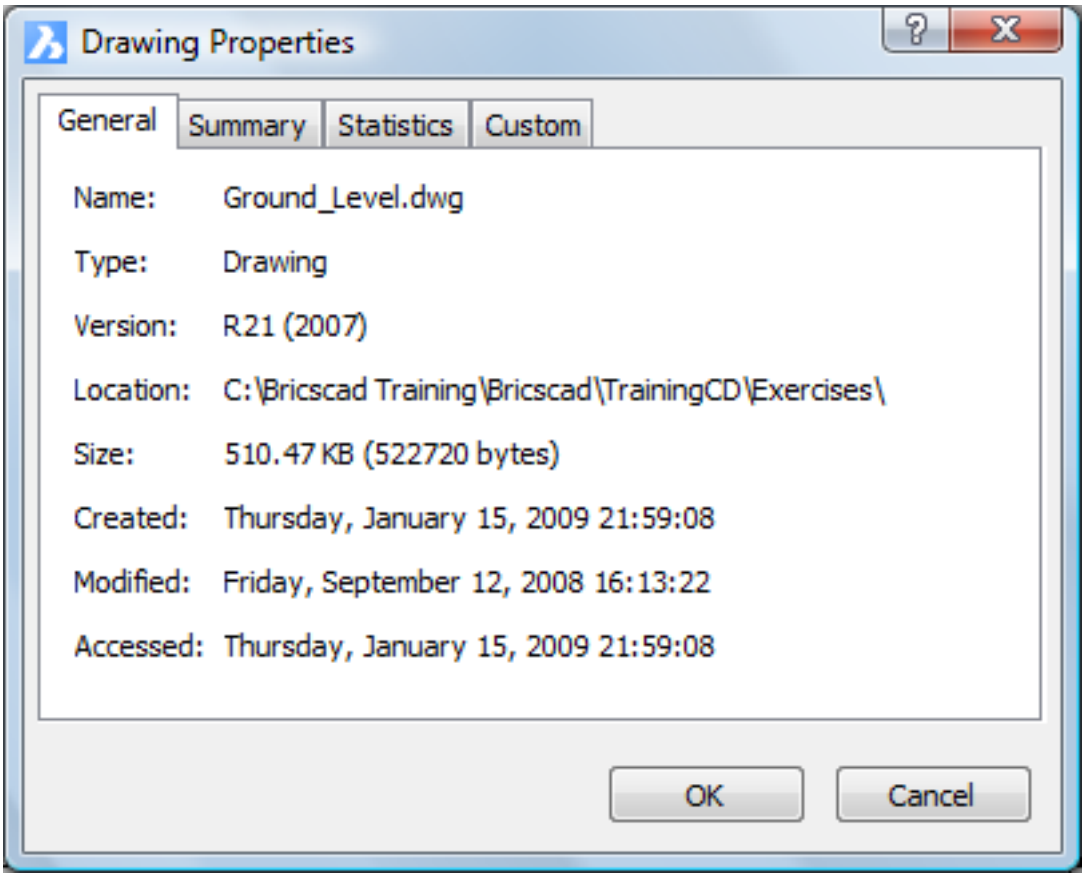
To open the Drawing Properties dialog

Do one of the following:

- Choose **Drawing Properties...** in the **File** menu.
- Type **dwgprops** in the command bar and press Enter.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/20ManagingDrawings/ART/02Drawin

Figure 1.417

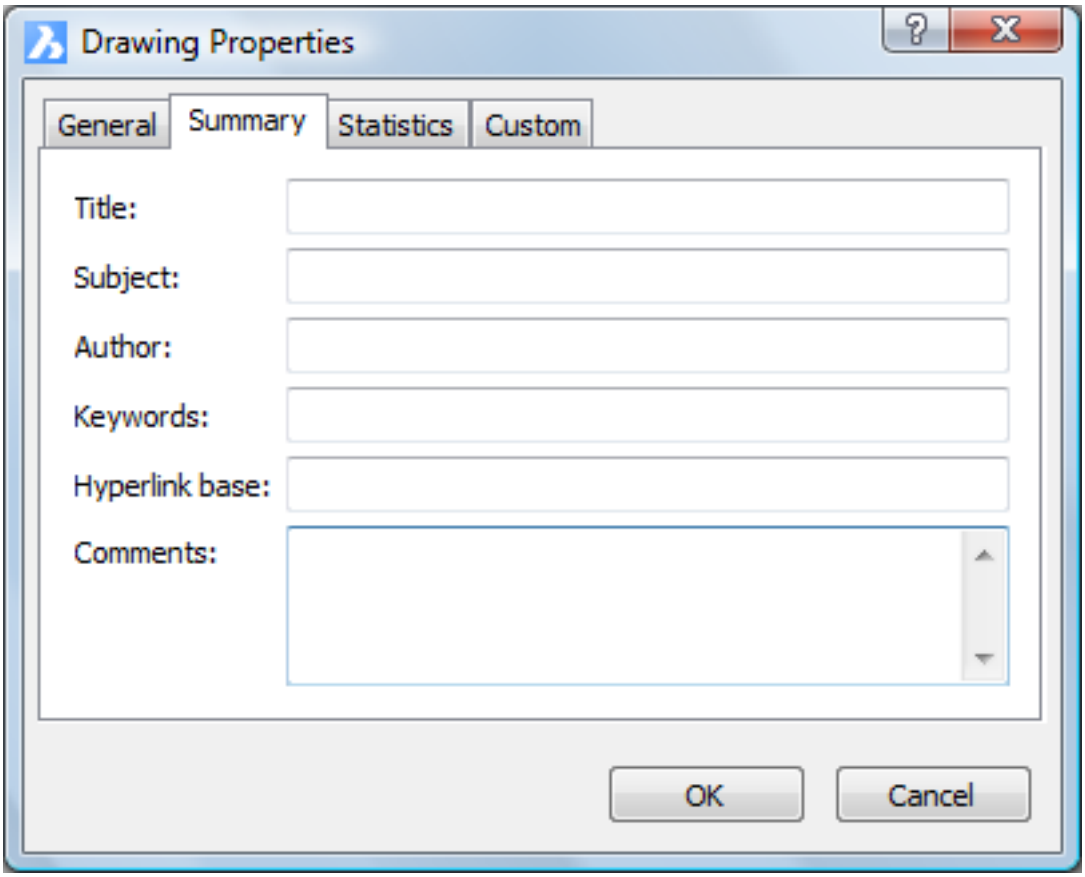


Defining default properties

1. Open the Drawing Properties (1.14.4) dialog.
2. Open the **Summary** tab page. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/20

3. Fill out the property fields
4. Click the **OK** button.

Figure 1.418



Adding custom properties

1. Open the Drawing Properties (1.14.4) dialog.
2. Open the **Custom** tab page. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/20M
3. Click the **Add** button. The **Custom Property** dialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
4. Type a name in the **Name** field. The custom property name is listed under **Document** in the Field dialog.
5. Type a value in the **Value** field. The value displays in multiline texts (1.21.20) where the custom property is inserted as a field.

Figure 1.419

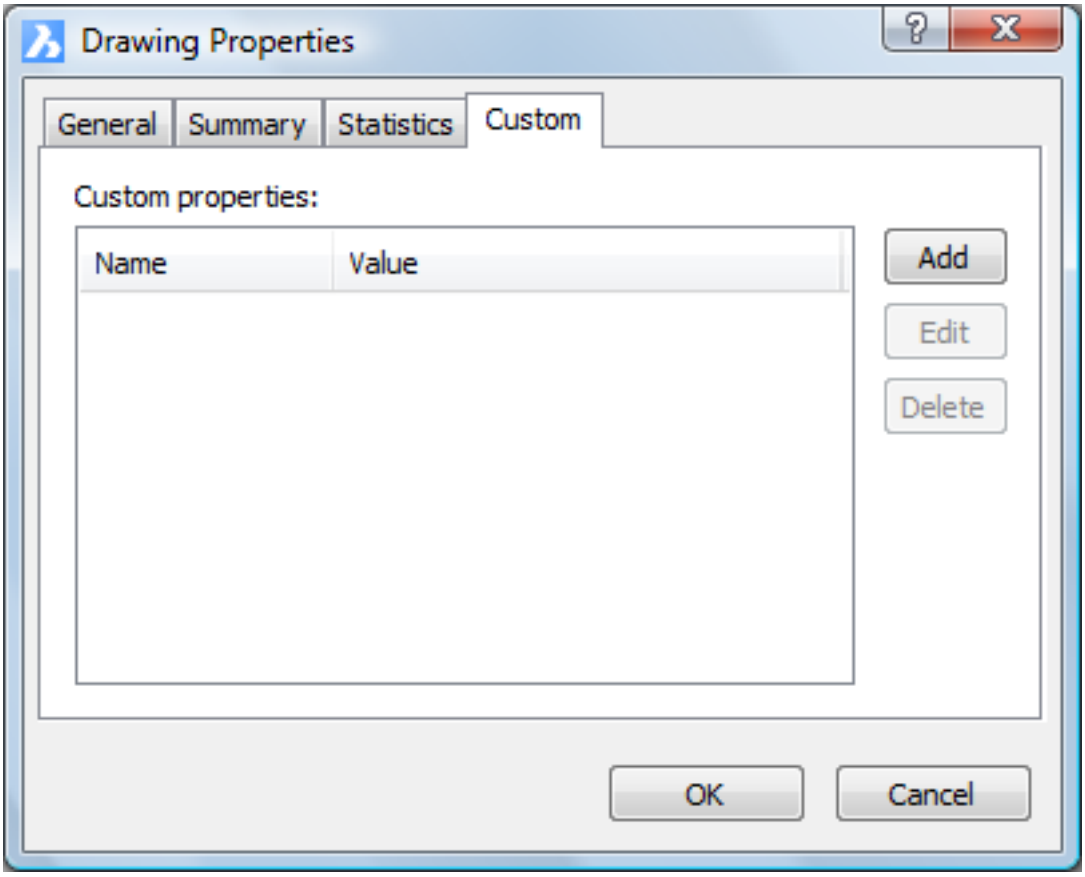
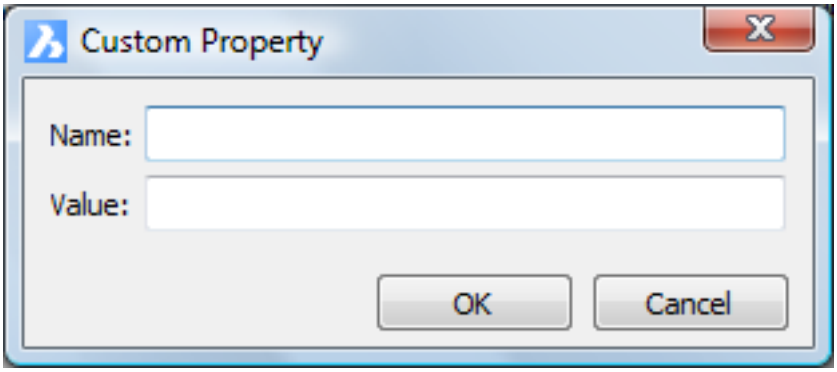


Figure 1.420



1.14.5 Drawing Security Options

Command : SECURITYOPTIONS (6.27.15) The **Securityoptions** command allows to protect your drawings with a password. Password protected drawings cannot be opened without entering the password first.

To set the password

Do one of the following:

- Choose **Security Options...** in the **File | Drawing Utilities** menu.
- Type **securityoptions** in the command bar, then press Enter.

1. The **Security options** dialog opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sour
2. Type the password in the **Enter Password** field.
3. Choose a crypt provider from the **Crypt Provider** list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
4. Set the **Key Length** to **40**, **48** or **56**.
5. Click the **OK** button or press Enter. The **Security options** dialog closes. The Password dialog opens.
6. Type the password in the **Confirm Password** field.
7. Click the **OK** button or press Enter. The password is defined.

Figure 1.421

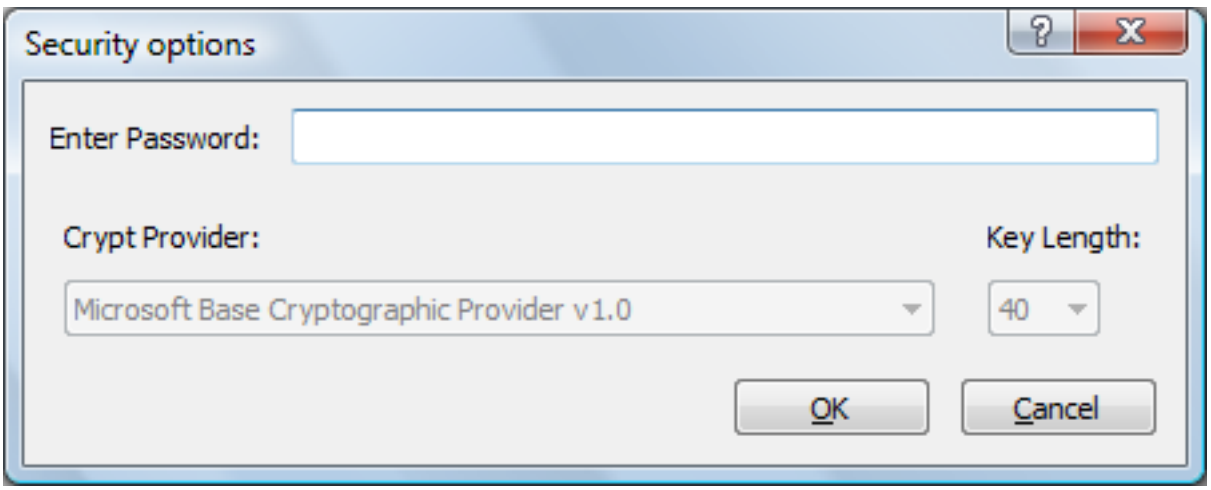
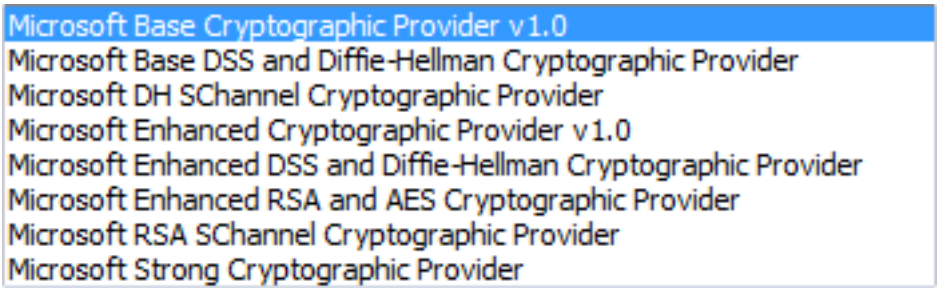


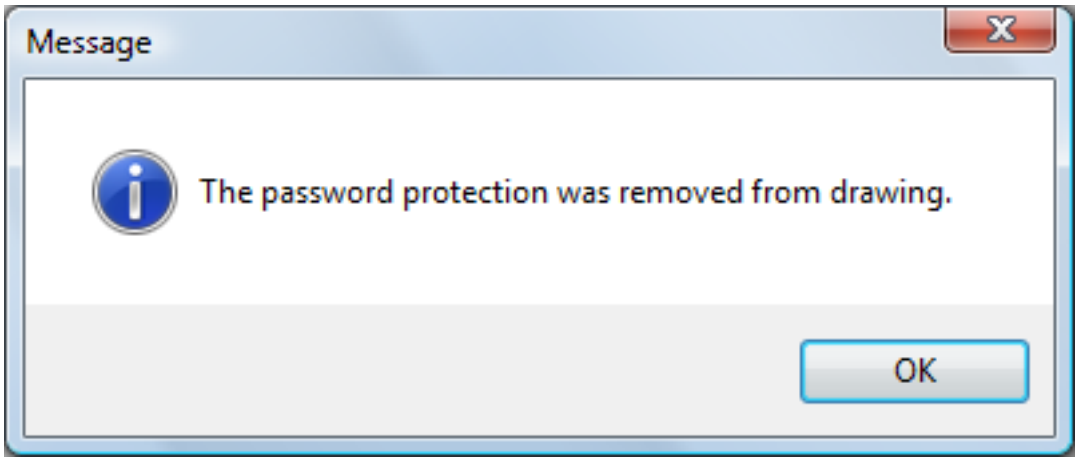
Figure 1.422



To remove the password

1. Type **securityoptions** in the command bar, then press Enter. The **Security options** dialog opens.
2. Empty the **Enter Password** field.
3. Click the **OK** button or press Enter. A message box displays to confirm that the password is removed. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/20ManagingDrawings/ART/05Passw
4. Click the **OK** button or press Enter to close the message box.

Figure 1.423



1.14.6 Opening and saving drawings

Commands: OPEN (6.23.8), SAVE (6.27.1), SAVEAS (6.27.3), SAVEALL (6.27.2), SAVEFILEFOLDER (6.27.5) The way BricsCAD opens and saves drawings is controlled through a series of system variables and user preferences. In the table below the SYSTEM VARIABLES are fully capitalized.

Name	Title	Description
ShowFullPathInTitle	Display full path in title	Controls wether the full path of a drawing displays in the title bar, or the file name only.
DrawingPath	Drawings path	Specifies an additional folder in the file dialog for the Open (6.23.8) and SaveAs (6.27.3) commands. On the Windows platform: the fifth folder in the **Open File** dialog's **Places** bar.
PlacesBarFolder1 - 4	First, Second, Third and Fourth folder	Sets the folders 1 through 4 in the **Places** bar of the non-standard **Open File** dialog (Windows platform only). Values between 0 and 5 are accepted: 0=Desktop, 1=My Computer, 2= My Documents, 3= Favorites, 4=Network, 5=My Recent Documents. Tip: Place shortcuts to your favorite drawing folders on your desktop or in your Favorites folder. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
THUMBSIZE	Thumbnail preview image size	Range is 0 - 8; Default = 1 0 (64 x 64) 1 (128 x 128) 2 (256 x 256) 3 (512 x 512) 4 (1024 x 1024) 5 (1440 x 1440) 6 (1600 x 1600) 7 (1920 x 1920) 8 (2560 x 2560)
INDEXCTL	Index Control	Determines whether layer and/or spatial indexes are created and saved. The layer index lists all entities in the drawing and what layer they are on. The spatial index stores the location of all the entities in 3D space. The indexes are used to load Xrefs if demand loading is enabled. If a layer index exists, entities on layers which are Off or Frozen are not loaded. If a spatial index exists, entities in clipped Xrefs which are outside the clipping boundary are not loaded. The demand loading of Xrefs is controlled through the XLOADCTL system variable). Creating indexes slows down the saving of a drawing, but increases the speed of Xrefs. Therefore create indexes only for drawings that will be used as an external reference. 0 = No indexes 1 = Layer index only 2 = Spatial index only 3 = Layer index and spatial index.

XLOADCTL	Xref load control	Controls demand loading of entities in external references. If demand loading is turned on you will experience a performance increase: • When clipping external references which are saved with a spatial index (INDEXCTL = 2 or 3), entities in the external reference within the clipped area are loaded only. If the clipping area is modified, more or less entities are loaded from the reference drawing. • When freezing layers in external references which are saved with a layer index (INDEXCTL = 1 or 3), entities in the external reference on thawed layers are loaded only. If xref-dependent layers are thawed more entities are loaded. 0: Demand loading is OFF - The entire external reference is loaded. 1: Demand loading is ON - Referenced drawings are kept open and locked. Other users can open such drawings only as read-only. 2: Demand loading is ON - Temporary copies of referenced drawings are kept open and locked. Other users can open and modify the referenced drawings.
ISAVEBAK	Incremental save backup	Controls the creation of a backup file (BAK). Improves the speed of incremental saves if switched off, especially for large drawings. Please notice that in some cases (i.e. a power failure during a save), it might be possible drawing data are lost.
ISAVEPERCENT	Save percent	This system variable controls how often BricsCAD performs a full save. A full save reorganizes all the data in the drawing database and creates a compact file. A partial save stores all changes a the end of the drawing data base. A full save takes longer to be executed than a partial save. If ISAVEPERCENT = 0, each save is a full save. The higher the value of ISAVEPERCENT, the longer the time between two full saves.
NFILELIST	Recent File List	Specifies the number of files shown in the **Recent Files** flyout of the **File** menu. Values between 0 and 20 are accepted. Default =10.

PROXYGRAPHICS	Proxy graphics	Determines whether images of proxy entities(1) are saved in the drawing. If switched off, a bounding box displays instead.
PROXYNOTICE	Proxy notice	Displays a notice when you open a drawing containing proxy entities(1) created by an application that is not present.
PROXYSHOW	Proxy show	Controls how proxy entities(1) display in a drawing. 0 = Not displayed 1 = Graphic images 2 = Bounding box.
PROXYWEBSEARCH	Proxy web search	Determines whether BricsCAD checks for object enablers: 0 = Never 1 = If a live Internet connection is present.
SaveFormat	Save format	Sets the default dwg version to save a drawing.
SAVEFILEPATH	Save file path	Specifies the path and folder where automatic saves and temporary files are stored. The SaveFileFolder (6.27.5) command opens the SAVEFILEPATH location in the system file browser.
SAVETIME	Save time interval	Sets the interval in minutes for automatic saves. If set to zero, automatic saves are turned off. Automatic saves are created with a **.SV$** extension in the folder defined by the SAVEFILEPATH system variable.
WorspaceSecurity	Enable macros	Defines whether macros are enabled when loading a VBA-project.
AutosaveChecksOnlyFirstbitDBMOD	Ignore all but first bit of DBMOD(2) for autosave	If this user preference is switched on, no autosave file will be created for files that are viewed only.
CloseChecksOnlyFirstBitDBMOD	Ignore all but first bit of DBMOD(2) for close	If this user preference is switched on, you can view a file, then close it without getting a request to save the file.

(1) Proxy entities are custom entities which are created by an add-on application. If the add-on is currently not loaded or not installed, such entities cannot be displayed properly. (2) DBMOD: Stores the status of the drawing modification as a bit-code as the sum of the following values: 1 = Object database modified, 4 = Database variable modified, 8 = Window modified, 16 = View modified, 32 = Field modified.

Using the Drag-and-Drop option menu

Select a file in a **Windows Explorer** dialog, then hold down the right mouse button and drag the file onto the drawing area of the BricsCAD application window. A context menu displays When a drawing file is selected: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/20ManagingDrawings/ART/08DragDropOpt
When another file type is selected: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/20M
Do one of the following: Choose **Default by File Extension** to

- open a drawing file (*.DWG, *.DXF or *.DWT)
- attach a PDF
- attach a raster image file
- insert a file as an OLE object (e.g. an Excel spreadsheet)

Choose **Insert Here** to:

- insert a drawing file (*.DWG, *.DXF or *.DWT) as a block in the current drawing.
- attach a PDF
- attach a raster image file

Figure 1.424

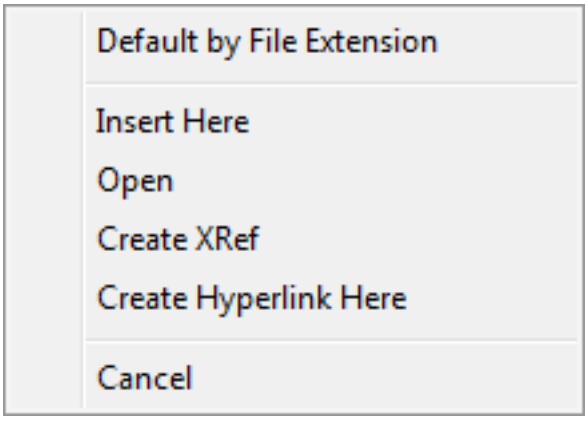
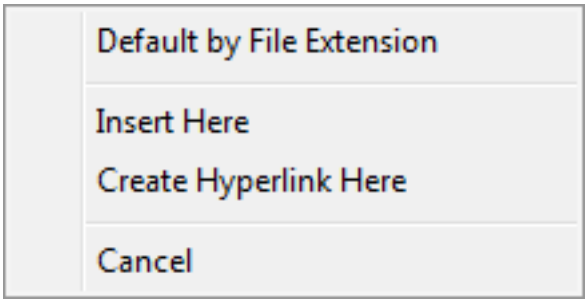


Figure 1.425

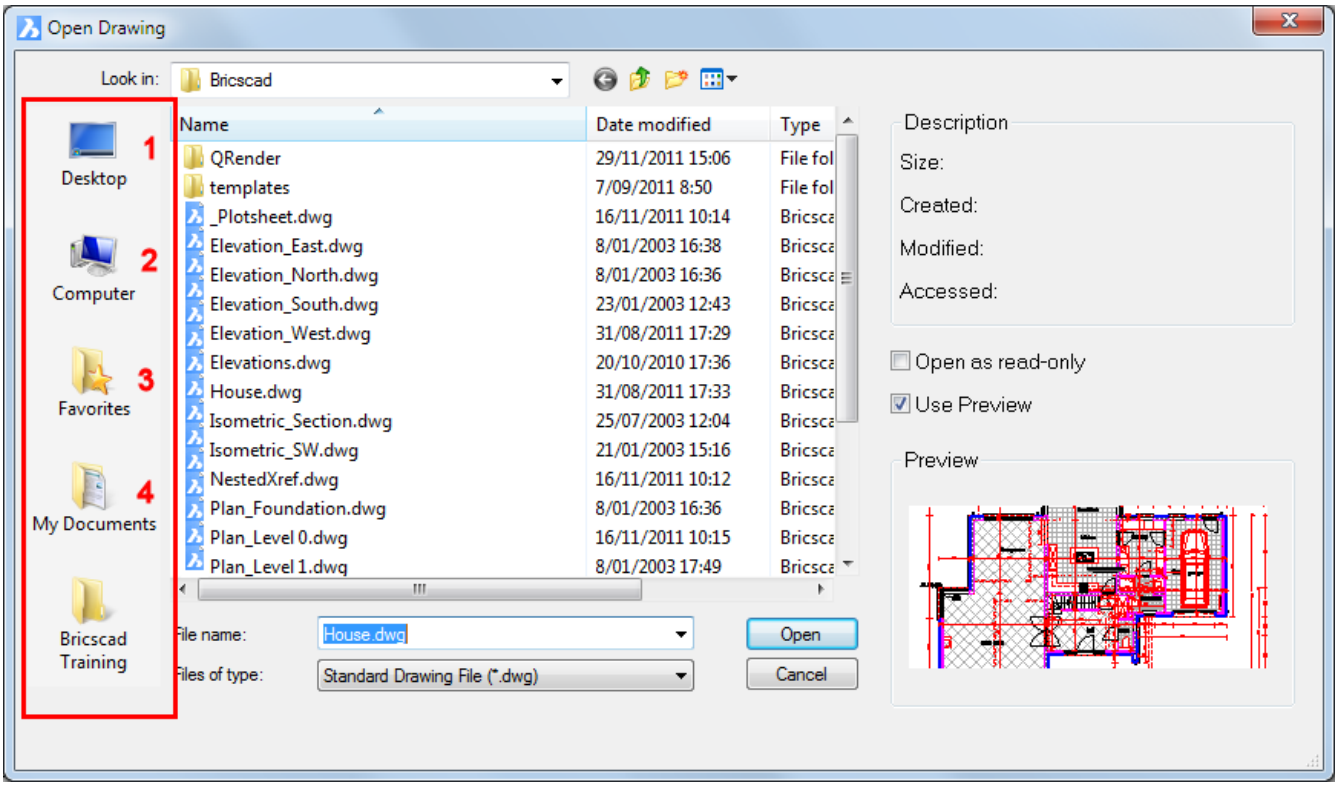


- insert the file as an OLE object

Choose **Open** to open a drawing file (*.DWG, *.DXF or *.DWT). Choose **Create Xref** to attach a drawing file (*.DWG, *.DXF or *.DWT) as an external reference to the current drawing. Choose **Create Hyperlink Here** to create an hyperlink to the selected file.

NOTE	When multiple files are selected, the files are processed one after the other.

Figure 1.426


Open Drawing
Look in:
Bricscad
Desktop
Computer
Favorites
My Documents
Bricscad Training
Name
Date modified
Type
QRender
templates
_Plotsheet.dwg
Elevation_East.dwg
Elevation_North.dwg
Elevation_South.dwg
Elevation_West.dwg
Elevations.dwg
House.dwg
Isometric_Section.dwg
Isometric_SW.dwg
NestedXref.dwg
Plan_Foundation.dwg
Plan_Level 0.dwg
Plan_Level 1.dwg
File name:
House.dwg
Files of type:
Standard Drawing File (*.dwg)
Open
Cancel
Description
Size:
Created:
Modified:
Accessed:
Open as read-only
Use Preview
Preview

1.14.7 Getting drawing information

You can retrieve two types of information from a drawing:

- General information about the drawing.
- Information about the entities in the drawing

General drawing information

Commands : DWGPROPS (6.10.42), SETVAR (6.27.26), STATUS (6.27.46), TIME (6.29.15) The **Dwgprops** command opens the Drawing Properties (1.14.4) dialog box, showing the general information and user defined properties stored with a drawing. The **Setvar** command displays or changes the values of system variables (short for "set variables"). The **Status** command reports the status of the drawing: drawing name, the status of the most important system variables and the total number of entities in the drawing. The **Time** command reports on time in the drawing: current time, creation time, last update, total editing time and user elapsed time.

Information about entities

Commands : STATUS (6.27.46), AREA (6.4.13), MASSPROP (6.21.4), ID (6.17.1), LIST (6.20.35) The **Status** command reports the total number of entities in the drawing. Using Quick Select (1.23.7) you can see the number of entities for each category. The **Area** command finds the area and perimeter (or length) of 2D entities. The **Massprop** command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties"). The **ID** command reports the x,y,z coordinates of picked points. The **List** command lists the properties of selected entities.

1.14.8 Comparing Drawings

Commands: DWGCOMPARE (6.10.41), ENDCOMPARE (6.13.6), KEEPME (6.19.1) The **DwgCompare** command compares a selected drawing with the current drawing during a comparison session. The **EndCompare** command concludes a comparison session. The **KeepMe** command allows to merge the compared drawings visually.

Comparison system variables

Name	Title	Description
CMPCLRMISS	Compare missing entities display color	Sets the color for displaying entities which are missing in the other drawing. The default color is Red .
CMPCLRMOD1	Compare modified entites display color 1	Sets the color for displaying modified entities in the current drawing. The default color is index color 253.
CMPCLRMOD2	Compare modified entities display color 2	Sets the color for displaying modified entities in the other drawing. The default color is Yellow.
CMPCLRNEW	Compare new entities display color	Sets the color for displaying entities which are new in the other drawing. The default color is Green.
CMPDIFFLIMIT	Maximal number of output entities	Sets the limit for entities comparison. The comparison will be aborted after reaching the difference limit. Values between 1 and 10 000 000 are accepted.

Executing a comparison session

1. Launch the DWGCOMPARE (6.10.41) command. You are prompted: CMPDIFFLIMIT = 10000000. Select the file to compare with or [Limit]:
2. (option) Choose the Limit option to set a new value for entities comparison.
3. Press Enter to display **File** dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/so
4. Select the drawing to be compared with the current drawing, then click the **Open** button or double click the drawing name. The differences between the two drawings display using the Compare Display Colors (**??**). The Compare toolbar displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01Us
5. To check the **Diff Type** of an entity, do one of the following:
   - If Rollover Tips (1.12.2) are active, hover over the entity: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
   - Select the entity and check the **Diff type** field in the Properties (6.24.49) panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/20ManagingDrawings/ART/10DiffTypeProperties
6. Analyze the comparison in the Structure panel.
7. Merge the compared drawings (1.14.8).
8. Click the **Endcompare** tool button ( ) on the **Compare** toolbar to conclude the comparison session.

Figure 1.427

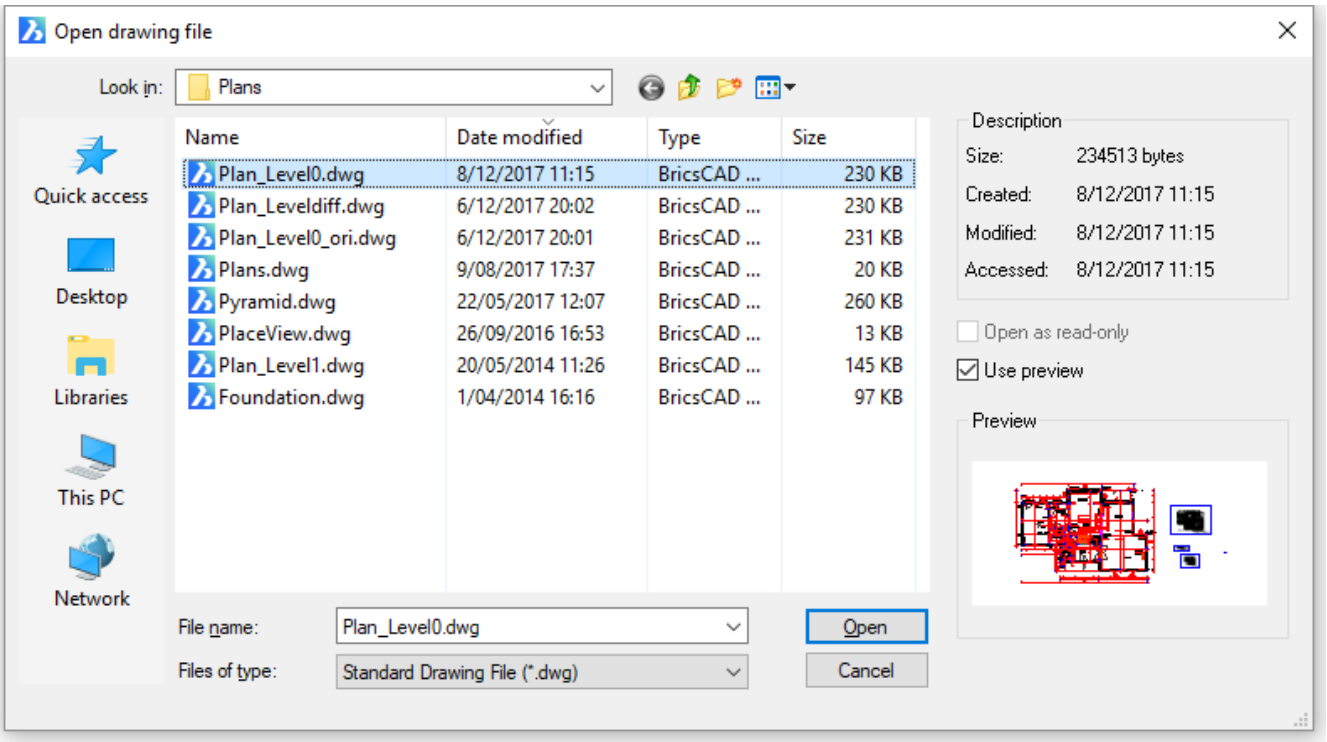


Figure 1.428

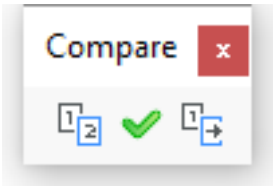


Figure 1.429

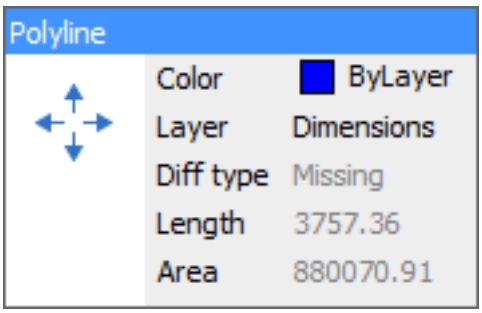
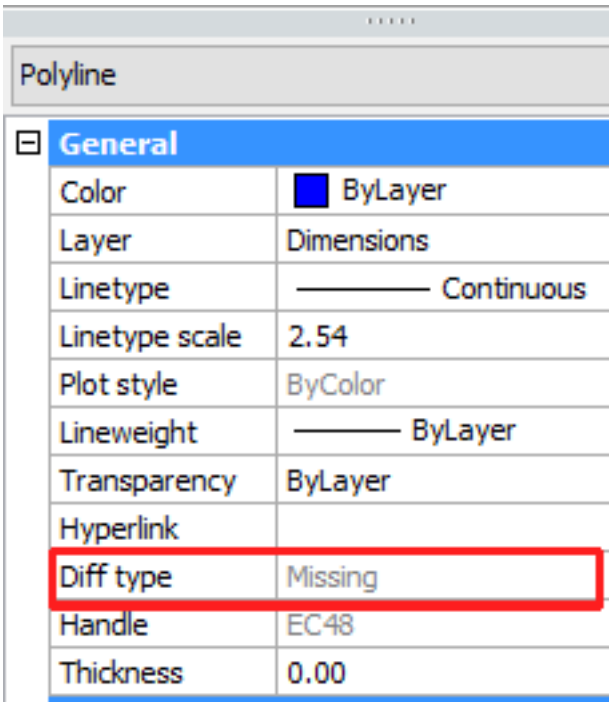


Figure 1.430



Analyzing the comparison in the Structure panel

1. Open the Structure panel (1.23.8).
2. Load the **Default.cst** configuration. See Load a Structure configuration file (1.23.8).
3. Expand the **Comparison** node. Four subnodes display: **Missing** , **Modified (Original**), **New** and **Modified (Compared)** . The number of entities in each subnode is indicated between brackets. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/20ManagingDrawings/ART/10DwgCompareStructureP
4. Click a subnode to select the corresponding entities in the drawing (see also the note below).
5. Expand a subnode, then click an entity to highlight the corresponding entity in the drawing. Press and hold the Ctrl-key to select multiple entities.
6. Right click, then choose **Zoom** in the context menu to zoom onto the selected entity or entities.

NOTE	When selecting entities in the Structure panel, entities are either selected or highlighted in the drawing, or tree selection is ignored. See setting the options in the Structure panel (1.23.8).

Figure 1.431



Merging the compared drawings

1. Missing entities only exist in the current drawing; do one of the following:
   - Select the entities you want to keep, then click the **Keep Me** tool button (✔) on the **Compare** toolbar.
   - Select the entities you don't want to keep, then click the **Delete** tool button (✖).
2. Modified entities exist in both drawings; do the following:
   - In the **Modified (Original)** node: select the entities you want to keep, then click the **Keep Me** tool button (✔) on the **Compare** toolbar. The corresponding entities in the **Modified (Compared)** node are removed.
   - In the **Modified (Compared)** node: select the entities you want to keep, then click the **Keep Me** tool button (✔) on the **Compare** toolbar. The corresponding entities in the **Modified (Original)** node are removed.
3. New entities exist in the compared drawing only. Do the following:
   - Select the entities you want to keep, then click the **Keep Me** tool button (✔) on the **Compare** toolbar.
   - Select the entities you don't want to keep, then click the **Delete** tool button (✖).

The merge is complete when the **Comparison** node in the **Structure** panel disappears. A message displays in the Command bar: No more differences to show. Session is closed

1.14.9 Repairing a drawing

Commands: AUDIT (6.4.32) and RECOVER (6.26.3) The **Audit** command analyzes the integrity of the current drawing and optionally tries to fix errors. The **Recover** command repairs a damaged drawing, then opens the drawing.

To analyze the integrity of the current drawing

1. Do one of the following:

   - Choose **Audit** in the **File | Drawing Utilities** menu.
   - Type **audit** in the command bar, then press Enter.

   You are prompted: Fix any errors detected? Yes/<No>

2. Do one of the following:

   - Choose **Yes** in the context menu or type **Y** , then press Enter to analyze the integrity of the current drawing and fix the errors detected.
   - Choose **No** in the context menu or type **N** , then press Enter to analyze the integrity of the current drawing.

   The drawing is analyzed.

3. (option) Press F2 to open the **Prompt History** window in order to read the audit report: # objects audited Total errors found during audit #, fixed #.

NOTE

- Set the **AUDITCTL** settings variable to 1, to let **Audit** create an ASCII file describing problems and the action taken. This report, with the file extension .adt, is placed in the same directory as the current drawing .
- Use the **Recover** command to repair the errors that **Audit** cannot fix.

To repair a drawing

1. Do one of the following:

   - Choose **Recover** in the **File | Drawing Utilities** menu.
   - Type **recover** in the command bar, then press Enter.

   The **Open Drawing** dialog window displays.

2. Select the drawing, then press the **Open** button on the **Open Drawing** dialog window. The drawing is repaired and - if the repair succeeded - opened.

3. (option) Press F2 to open the **Prompt History** window in order to read the recover report.

1.14.10 Etransmit

Command : ETRANSMIT (6.13.9) The **Etransmit** command creates a package of a drawing file and all its dependencies, such as: external references, images, font files, plot configuration files, plot style tables and font map files. The root drawing and all its dependencies can be copied to a folder are compressed in a ZIP file. Optionally you can send the ZIP file attached to an email.

Tutorial: Package Your Drawings with eTransmit Click here (1.4.12) to watch

1.14.11 Projects

Projects are especially useful to manage xrefs and images when exchanging drawings. If external references (xrefs) and images are not found in the saved path, the project search paths are used to find the external references and images. Projects are saved in the registry and contain one or more search paths.

Opening the Project settings

1. Click the **Settings** tool button ( ) on the **Standard** toolbar. **Or** Choose **Settings...** in the **Settings** menu. The **Settings** dialog opens.
2. Go to **Program Options > Files > Projects > Project search paths** .

Creating projects

To create a project:

1. Open the Project settings.
2. Click the **Browse** button at the right hand side of the **Project search paths** settings field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/02Definingyourpreferences/ART/11SettingsProjects.p
The **Project Settings** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sour
3. To create the first project:
   - Click the **New** button ( ) on the **Project Settings** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
   - Type a name to replace the **NewProject1** default name.
4. To create additional projects.
5. Click **Projects** (on top of the projects tree) then click the **New** button ( ) .
6. Type a name to replace the **NewProject1** default name.

Figure 1.432



Figure 1.433

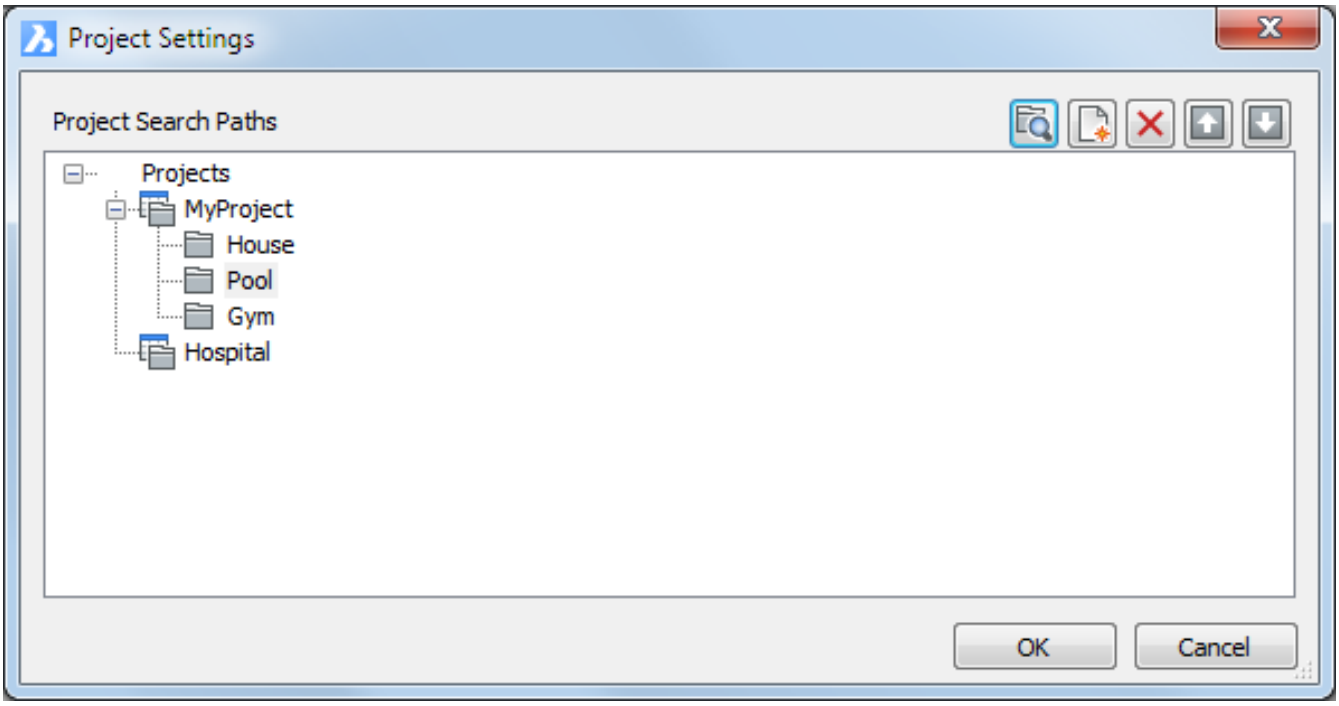
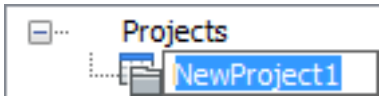


Figure 1.434

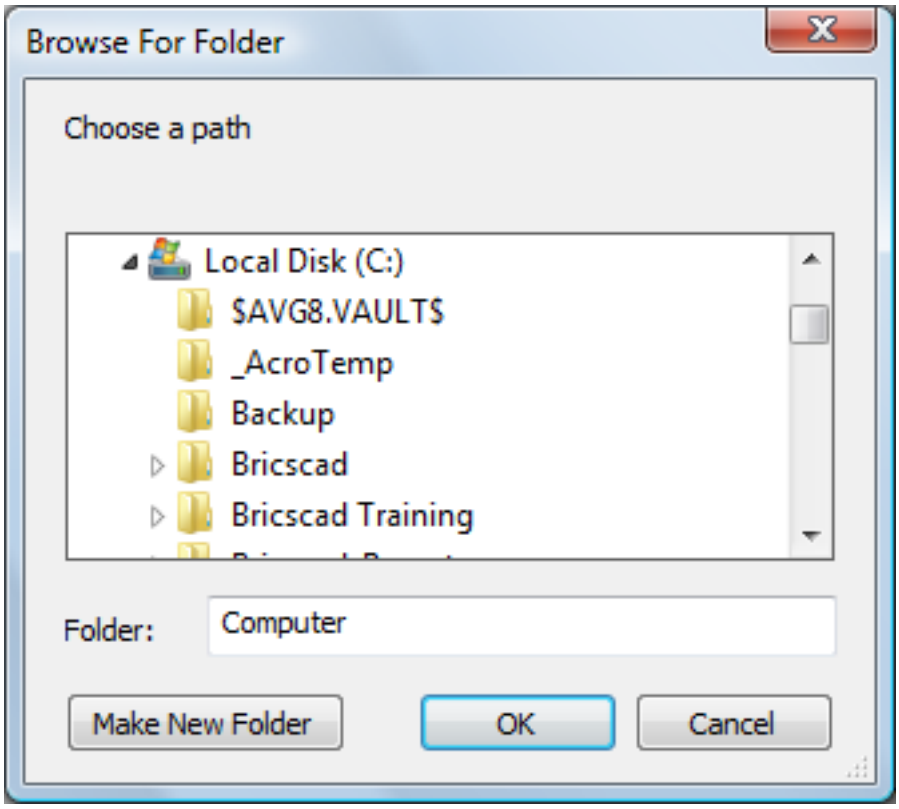


Adding search paths to a project

To add a search path to a project:

1. Open the Project settings.
2. Select the project in the **Projects** tree.
3. Click the **Browse** button ( ). The **Browse for folder** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscon
4. Do one of the following:
   - Select a folder.
   - Select a parent folder, then click the **Make New Folder** button to create a new folder.
5. Click the **OK** button to add the selected folder to the project search paths.

Figure 1.435



Changing the order of the search paths

1. Open the Project settings.
2. Click the **expand** button (+) of the project you want to edit.
3. Select a search path, then click the up ( ) or down ( ) buttons to change the order of the search paths.

Assigning a project to the current drawing

1. Open the Project settings.
2. Type the name of an existing project in the **Project name** settings field (**PROJECTNAME** system variable).

1.14.12 Export to PDF

Command: EXPORT (6.13.15) Drawing files can be saved as a PDF file, optionally including layers.

Export to PDF settings

Choose **PDF Export Options...** in the **File** menu to display the **PDF Export** settings in the **Settings** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/20ManagingDrawings/ART/04ExportPdfSe

- **Pdf embedded fonts** : Controls whether True Type Fonts are embedded in the PDF export.
- **Pdf TTF text as geometry** : If selected, TTF text is exploded to geometry.
- **Pdf SHX text as geometry** : If selected, SHX text is exploded to geometry.
- **Pdf text is searchable** : If selected, it is possible to search text strings in the PDF document.
- **Pdf simple geometry optimization** : Enable simple geometry optimization: e.g. separate segments to one polyline, use of Bezier curve control points.
- **Pdf zoom to extents mode** : If exporting model space, zoom to extents first, if the current model space view is zoomed in.
- **Pdf merge control** : Controls the appearance of crossing lines. The options are: **Lines Overwrite** : The topmost line obscures the line(s) under it. **Lines Merge** : The colors of crossing lines are merged.
- **Pdf layer support** : Controls whether layers are included in the PDF export. The options are: do not export layers, export layers which are on only, export all layers.
- **Pdf layouts to export** : Controls which layouts are exported. The options are: **Active** : only the active layout or model space is exported as a single-page pdf. **All** : All layouts, including model space are exported as a multi-page pdf.
- **Pdf papersize override** : If on, the papersize as defined in the BricsCAD Print settings is overridden. The papersize width and height defined by **PdfPaperWidth** and **PdfPaperHeight** is used instead.
- **Pdf overridden paper width** : Width (in mm) of the PDF paper, if **PdfPaperSizeOverride** is on.
- **Pdf overridden paper height** : Height (in mm) of the PDF paper, if **PdfPaperSizeOverride** is on.
- **Export solid hatches** : Controls the export of solid hatches. The options are: **as bitmap** , **use vectorizer** , **use PDF paths** .
- **Pdf use plotstyles** : If on, the plotstyle of the layout controls the color and lineweight in the PDF export.
- **Image anti-aliasing** : Enables anti-aliasing for images that require upscaling when exporting.
- **Image compression** : Defines the compression for exported images: **None** or **JPEG** .
- **Vector Resolution DPI** : Controls the resolution of vector graphics when exporting to PDF. Default is 600 DPI.
- **Image DP** I: When a raster image is exported, this value controls the minimal resolution for the image.
- **Render DPI** : Resolution of a rendering when exporting.
- **Hatch to bitmap DPI** : Defines the resolution of hatch patterns and gradient fills in DPI (= dots per inch).

Figure 1.436

PDF export

Pdf embedded fonts	☑ Enable embedded TTF fonts
Pdf TTF text as geometry	☐ Convert TTF text to geometry
Pdf SHX text as geometry	☐ Convert SHX text to geometry
Pdf text is searchable	☐ Enable pdf text to be searchable
Pdf simple geometry optimization	☑ Enable simple geometry optimization
Pdf zoom to extents mode	☑ Zoom to extents
Pdf Merge Control	[0] Lines Overwrite
Pdf layer support	[1] Use all layers with visible entities
Pdf layouts to export	[0] Active
Pdf papersize override	☐ Enable papersize override
Pdf overridden paper width	210
Pdf overridden paper height	297
Pdf use plotstyles	☑ Enables usage of plotstyles
Export solid hatches	[2] use PDF paths
Image anti-aliasing	☑ Enables anti-aliasing for images
Image compression	[1] JPEG
Vector Resolution DPI	600
Image DPI	300
Render DPI	300
Hatch to bitmap DPI	300

Exporting a drawing to PDF

1. (option) Adjust the Export to PDF (1.14.12) **settings**.
2. Choose **Export...** in the **File** menu. The **Export Drawing As** dialog opens.
3. In the **Save as type** : list select **Adobe PDF (*.pdf)** .
4. Select a folder to save the PDF file.
5. (option) Type a name in the File name field to replace the default <DRAWING_NAME>.pdf name.
6. Click the **Save** button to create the PDF.

Tutorial: Output As Pdf Click here (1.8.1) to watch

1.15 Drawing Accurately

1.15.1 Drawing Accurately Overview

In BricsCAD the following drawing aids are available:

- Entity Snaps (1.15.15)3D Entity Snaps (1.15.14)
- Dynamic dimensions (1.15.9)
- Polar Tracking (1.15.16)
- Entity Snap Tracking (1.15.17)
- Coordinate Input (1.15.20)
- Snap and Grid (1.15.11)
- Drawing Limits (1.15.11)
- Ortho Mode (1.15.2)
- User Coordinate Systems (1.15.18)
- Direct Distance Entry (1.15.3)
- Nearest Distance (1.15.10)

Tutorial: Drawing Accurately: Snap and Grid Click here (1.4.1) to watch

Tutorial: Drawing Accurately: Polar and Snap Tracking Click here (1.4.2) to watch

Tutorial: Drawing Accurately: Dynamic dimensions Click here (1.4.3) to watch

1.15.2 Orthogonal Mode

Command: Ortho (6.23.11) **Overview:** Orthogonal Mode restricts the movement of the cursor to be parallel to the X-axis or the Y-axis of the current coordinate system (1.15.18). For example, with the default 0-degree orientation (angle 0 at the "three o'clock" or "east" position), when the Orthogonal Mode setting is enabled, lines are restricted to 0 degrees, 90 degrees, 180 degrees, or 270 degrees. As you draw lines, the rubber-banding line follows either the horizontal or vertical axis, depending on which axis is nearest to the cursor: type the length of the line in the command bar, then press Enter.

NOTES

- When you enable the Isometric snap style (1.15.11), cursor movement is restricted to orthogonal equivalents within the current isometric plane.
- You can rotate the **Orthogonal Mode** axes using the Snap Angle (1.15.11) setting.
- Press and hold the Shift key to reverse the **Orthogonal Mode** setting.
- **Orthogonal Mode** is overruled by entity snaps.
- Turning on **Ortho Mode** automatically disables Polar Tracking (1.15.16) and vice versa.

To toggle Orthogonal Mode

Do one of the following:

- Click the **ORTHO** field in the Status Bar (1.12.2).
- Press the **F8** function key.
- Press and hold the Shift key.
- Type **ortho** in the command bar, then choose the appropriate setting.
- Set the **Orthogonal Mode** in the **Settings** dialog.

1.15.3 Direct Distance Entry

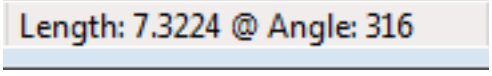
Direct Distance Entry (DDE) means that you can type the length of lines, polyline segments, circle radii and diameters, movement vectors in commands such as Move , Copy and Stretch , etc. in the command bar. When combined with either Orthogonal Mode (1.15.2) or Polar Tracking (1.15.16), **Direct Distance Entry** is a particularly efficient method to draw lines of a specified length and direction, and to move or copy objects over a specified distance. To help you keep track of the position of the cursor it is recommended to set the readout of the coordinate field in the Status Bar (1.12.2) to **Relative** .

To define the Coordinates setting

Do one of the following:

- Right click the then choose **Relative** in the context menu.
- Click the **Coordinates** field in the status bar (1.12.2) to cycle the **Coordinate** setting. The sequence is **Off** , **Relative** , **Absolute.**
- Press the **F6** function key or use the **Ctrl + I** keyboard shortcut to cycle the **Coordinate** setting. The sequence is **Off**, **Relative**, **Absolute.**
- Type **coords** in the command bar, then press Enter. Type 2 and press Enter.
- Open the Settings dialog (1.13.11) and expand the **Display/Viewing** settings sub-category under **Drafting** in the **Drawing** settings class. Set the **Coordinates** setting to **Coordinates in polar form for point, distance and angle selection** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/08S

Figure 1.437



1.15.4 Unit Settings

Commands: UNITS (6.30.8) and -UNITS (6.30.9) **Overview:** The **Units** command sets units of linear and angular measurements through the Settings (1.13.11) dialog box. The **-Units** command sets units of linear and angular measurements in the command bar. The INSUNITS (1.22.6) (Insertion Units) system variable defines a drawing units value for automatic scaling when inserting blocks (1.22.6) or attaching Xrefs (1.22.16). Any length, area or volume property can be displayed using the unit(s) preferred by the user. The unit defined by INSUNITS is used as the base unit for any requested unit conversions. If INSUNITS = 0 (undefined) the PROPUNITS mechanism is disabled because the base drawing unit is undefined.

NOTES

- When LUNITS = 3 (Engineering) or 4 (Architectural), property values will be formatted in imperial notation.
- Automatic unit conversion during length input can be used when bitcode 1 of PROPUNITS (Format length properties) is set. E.g. when INSUNITS is set to ”mm”, entering ”2m” in a length property field will result in entering [2000.0000 mm]. Entering ”1ft” will give [304.8000 mm].

Name	Title	Description
INSUNITS	Insertion Units	Defines the value of 1 Drawing Unit (DU), which is used for automatic scaling when inserting or attaching blocks, images or Xrefs. When both INSUNITS and PROPUNITS are ON, Length, Area, Volume and/or Inertia properties will be formatted with a dedicated unit.
MEASUREMENT	Measurement	Sets drawing units as Imperial or Metric. The MEASUREMENT system variable controls whether ANSI or ISO hatch pattern and linetype files are used.
PROPUNITS	Property Units	Defines which property values are automatically formatted with a dedicated unit when INSUNITS is not zero. The value is stored as a bitcode, using the sum of the values of all selected options.
LUPREC	Linear Unit Precision	Sets the number of decimal places displayed for linear units.
AREAPREC	Area Precision	Sets the number of decimal places displayed for areas when the Area bit of PROPUNITS is ON. If set negative, the value of LUPREC is used.
VOLUMEPREC	Volume Precision	Sets the number of decimal places displayed for volumes when the Volume bit of PROPUNITS is ON. If set negative, the value of LUPREC is used.
LENGTHUNITS	Length Units	Defines a list of units used for displaying lengths when the Length bit of PROPUNITS is ON. If empty, all lengths display in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.

AREAUNITS	Area Units	Defines a list of units used for displaying areas when the Area bit of PROPUNITS is ON. If empty, all lengths display in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.
VOLMEUNITS	Volume Units	Defines a list of units used for displaying volumes when the Volume bit of PROPUNITS is ON. If empty, all lengths display in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.
UNITMODE	Unit Mode	Specifies how Imperial units display.

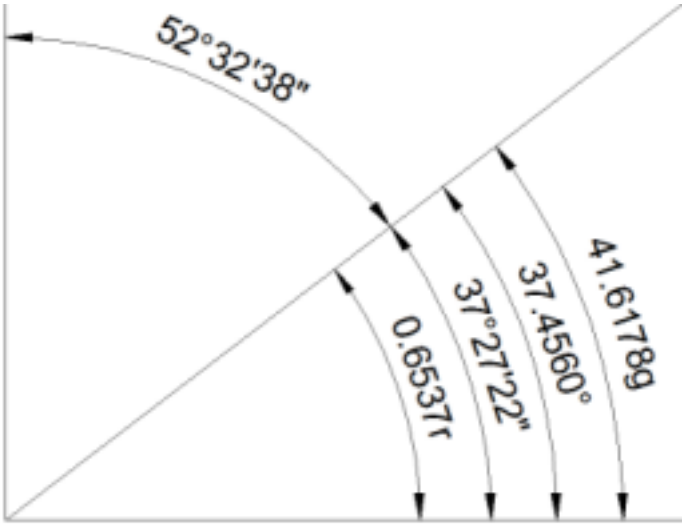
Entering angles

The **AUNITS** system variable defines the angular unit type. You can enter angles in any of 5 formats:

- Decimal degrees : 37.456 (AUNITS = 0)
- Degrees / Minutes / Seconds : 37°27’22” or 37d27’22” (AUNITS = 1)
- Gradians : 41.6178g (AUNITS = 2)
- Radians : 0.6537r (AUNITS = 3)
- Surveyor’s Units : N52°32’38”E or N52d32’38”E (AUNITS = 4) Surveyor’s units are measured from the North (N) or the South (S) to the East (E) or to the West (W).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/03EnteringAngles.png})

Figure 1.438



1.15.5 Coordinate Reference System

Command: GeographicLocation (6.15.15) **Overview:**A Coordinate System is a set of mathematical rules for specifying how coordinates are to be assigned to points. The coordinate system is unrelated to the Earth. A Coordinate Reference System (CRS) is a coordinate system related to the Earth through a ”Datum”. A Geodetic Datum is a model of the Earth that specifies a reference surface (Ellipsoid or Spheroid). A projected coordinate reference system is the result of the application of a map projection to a geographic coordinate reference system. A map projection is a type of coordinate conversion. It uses an identified method with specific formulas and a set of parameters specific to that coordinate conversion method. Coordinates may be changed from one CRS to another through the coordinate operation. Two types of coordinate operation may be distinguished:

- **coordinate conversion**, where no change of datum is involved and the parameters are chosen and thus error free.
- **coordinate transformation**, where the target CRS is based on a different datum to the source CRS. Transformation parameters are empirically determined and thus subject to measurement errors. (Geodetic, Seven paramets, Molodenksy & etc.)

Geographic datasets are defined in **geodatabase.xml**that is placed in <User folder>\AppData\Roaming\Bricsys\BricsCAD\<RE
for Windows. The content of this file is composed of the 5 sections:

- CoordinateReferenceSystems
- Datums
- Ellipsoids
- Transformations
- ProjectionCode and Methods

1. CoordinateReferenceSystems

This section is the main xml node where all Coordinate Reference Systems are defined. It has many sub-nodes called **CRS**, each xml node must define a unique projected coordinate reference system and the necessary projected parameters. The supported CRS projection types are described in the **ProjectionCode** and **Methods** sections. **CRS**node attributes: Follow Table 1 of OGP Geomatics Guidance Note No 7, part 2

Attribute name	Description	Units
epsg	Unique EPSG database id.	Integer
name	Human-readable name of CRS, Example: ”WGS 84 -- WGS84 - World Geodetic System 1984, used in GPS”	String
codeSpace	Owner of CRS. It is option.	String
x, **y**	Orientation of projected direction. Possible values: ”Westing”, ”Easting”, ”Southing” and ”Northing”. Directions MUST be defined.	String
xAxis, yAxis	Orientation of geographic direction. Possible values: ”Latitude” and ”Longitude”.	String
Greenwich	Greenwich Meridian relation.	Degree
proj	Projection method of CRS. Possible values: ”LL”, ”LatLon”, ”LonLat” is a Geodetic Latitude-Longitude. ”MercA”, ”Mercator_1SP” is a Mercator (variant A). ”MercB”, ”Mercator_2SP” is a Mercator (variant B). ”MercC” is Mercator (variant C). ”MercSp” is a Mercator (Spherical). ”MercPv” is a Popular Visualisation Pseudo Mercator. ”TMerc” is a Transverse Mercator. ”TMercSO” is a Transverse Mercator South Orientated. ”LCC_1SP” is a Lambert Conic Conformal 1SP. ”LCC_2SP” is a Lambert Conic Conformal 2SP. ”LCC_2SP_BE” is a Lambert Conic Conformal 2SP Belgium.	String

AngleSG	Angle from Rectified to Skew Grid	Degree
Azimuth	Azimuth of initial line	String
FE	Easting at false origin, False easting	Float-Point value in CRS units
EC	Easting at projection center	Float-Point value in CRS units
FN	Northing at false origin, False northing	Float-Point value in CRS units
FC	Northing at projection center	Float-Point value in CRS units
LonI	The longitude of the western limit of the first zone of a Transverse Mercator zoned grid system	Degree
Lat	Latitude of natural origin, Latitude of standard parallel. It depends on Projection method.	Degree
Lat1	Latitude of 1st standard parallel	Degree
Lat2	Latitude of 2nd standard parallel	Degree
LatF	Latitude of false origin	Degree
LatC	Latitude of projection center	Degree
Lon	Longitude of natural origin, Longitude of origin	Degree
LonF	Longitude of false origin	Degree
LonC	Longitude of projection center	Degree
SF	Scale factor at natural origin	Float-Point value, unity
SFIL	Scale factor on initial line	Float-Point value, unity
SFPSP	Scale factor on pseudo standard parallel	Float-Point value, unity
W	Zone width in longitude	Degree
zone	Zoned Grid System	Degree
units	Units of conversion result from geographic coordinates to projected. Example: ”Meter”, unit ”Degree” means no conversion, Geographic Coordinate System	String

Example: common attributes for all map projections based on EPSG:31468.

```
<CRS epsg="31468" codeSpace="OGP" name="DHDN / 3-degree Gauss-Kruger zone 4" y="Easting" x="Northing" C
```

For each node **CRS** there should be at least one sub-node **Datum** that refers via **epsg** or **alias** attributes to sub-node **Datum** in the **Datums** section. The sub-node **Datum** MUST have the **id** attribute of a unique CRS name. At least one of the epsg code and the alias of Datum must be valid.

Attribute name	**Description**	**Units**
epsg	Unique EPSG database id. Example: ”4326”	Integer
alias	Unique Datum name. Example: ”WGS 84”	String
id	Unique CRS name: combination of Projected method and Datum. Example: ”WORLD-MERCATOR”. It related to CS-MAP, AutoCAD names.	String
pjcode	Indicate code for the coordinate reference system projection method types. Corresponded to AutoCAD projection codes. For instanse, 3 - Transverse Mercator, 44 - UTM, 45 - Transverse Mercator of Snyder formulation and etc.	Integer

Example: there are unique CRS definitions according to Datums inside **CRS** node:

```
<Datum epsg="6314" alias="DHDN/3" id="DHDN/3.Gauss3d-4" pjcode="3" /> <Datum epsg="6314" alias="DHDN/2"
```

2. Datums

This section is a main node where all Datums are stored. Datum is a combination of the Earth model (Ellipsoid or Spheroid) and the transformation method to model WGS84. A **Datum** node specifies common parameters and contains at leat one inner sub-nodes **Datum** exactly one sub-node **Ellipsoid**. Each inner sub-node **Datum** should define a unique name within the **id** attribute and may specify transformation parameters to WGS84 model via sub-node **Transformation** that refers via **epsg** or **alias** attributes to a sub-node **Transformation** in the **Transformations** section.

Attribute name	Description	Units
alias	Unique id. Example: ”DHDN/3”. It related to CS-MAP, AutoCAD names.	String
epsg	Unique EPSG database id. Example: ”6314”	Integer
name	Human-readable name of Datum, Example: ”Deutsches Hauptdreiecksnetz”	String
codeSpace	Owner of CRS. It is option.	String

Example:

```
<Datums> <Datum epsg="6314" codeSpace="OGP" name="Deutsches Hauptdreiecksnetz "> <Datum id="DHDN/3"> <Tr
```

3. Ellipsoids

The Ellipsoids section is the main section where models of the Earth are defined. Each **Ellipsoid** node should define an identification by a unique name (**alias**) and a semi-major axis (**a**) and one of the following attributes at least: flattering (**f**), semi-minor axis (**b**) or eccentricity (**e**).

Attribute name	Description	Units
alias	Unique id. Example: ”WGS84”. It related to CS-MAP, AutoCAD names.	String
epsg	Unique EPSG database id. Example: ”7030”	Integer
name	Human-readable name of Datum, Example: ”WGS 84”	String
codeSpace	Owner of CRS. It is option.	String
a	Length of the semi-major axis of the ellipsoid, the radius of the equator	Float-Point value, strongly in Meter
b	Length of the semi-minor axis of the ellipsoid, the distance along the ellipsoid axis between equator and pole	Float-Point value, strongly in Meter
f	Flattening	Float-Point value, unity
e	Eccentricity	Float-Point value, unity

Example:

```
<Ellipsoid epsg="7008" alias="CLRK66" name="Clarke 1866, Benoit Ratio" a="6378206.4000000004" b="6356583
```

4. Transformations

This section is a main node where all transformation methods are defined. Supported Transformation methods are described in the **ProjectionCode** and **Methods** sections.

Attribute name	Description	Units
alias	Unique id. Example: ”DHDN_to_WGS84”. It related to CS-MAP, AutoCAD names.	String
epsg	Unique EPSG database id. Example: ”1673”	Integer
codeSpace	Owner of Transformation. It is option.	String
source	Source Datum	String

target	Target Datum	String
use	Transformation method. Supported transformations using geocentric methods: Geocentric translations, Four/Six/Seven Parameter Transformation, Similarity Transformation, Position Vector transformation and Coordinate Frame rotation.	String
method	Methods of building of rotation matrix, if applicable, ”PVT” is Position Vector Transformation, ”CFR” is Coordinate Frame Rotation	String
tX	X-axis translation	Meter
tY	Y-axis translation	Meter
tZ	Z-axis translation	Meter
rX	X-axis rotation	Degree
rY	Y-axis rotation	Degree
rZ	Z-axis rotation	Degree
dS	Scale difference	Unity
xp	Coordinate 1 of evaluation point	Meter
yp	Coordinate 2 of evaluation point	Meter
zp	Coordinate 3 of evaluation point	Meter
dtX	Rate of change of X-axis translation	Unity
dtY	Rate of change of Y-axis translation	Unity
dtZ	Rate of change of Z-axis translation	Unity
drX	Rate of change of X-axis rotation	Unity
drY	Rate of change of Y-axis rotation	Unity
drZ	Rate of change of Z-axis rotation	Unity
ddS	Rate of change of scale difference	Unity
t0	The reference epoch for time-dependent parameters.	Float point value
fallback	Alias of transformation from target datum to WGS84 datum.	String

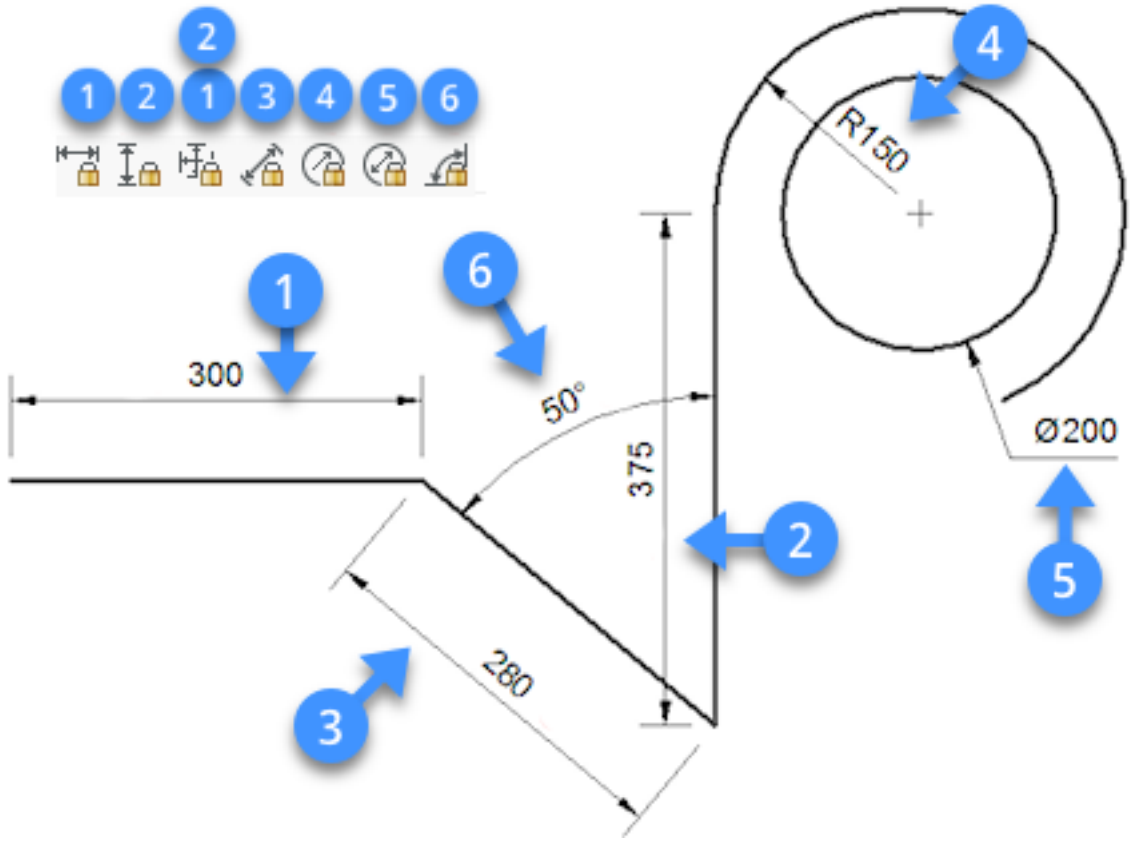
Example:

```
<Transformation epsg="1679" alias="Pulkovo42/2_to_WGS84" src="Pulkovo42/2" trd="WGS84" use="Param7" meth
```

5 . ProjectionCode and Methods

These sections describe the **ProjectionCode** and **Methods** that are mapped to AutoCAD definitions of projections and transformations types. They are used to store the definitions in an XML file of coordinate reference system definition inside the **AcDbGeoData** object that represents a geographic location.

Figure 1.439



1.15.6 Using Dimensional Constraints

Commands: DIMCONSTRAINT (6.10.29), DCALIGNED (6.10.5), DCANGULAR (6.10.6), DCCONVERT (6.10.7), DCDIAMETER (6.10.8), DCDISPLAY (6.10.9), DCHORIZONTAL (6.10.10), DCLINEAR (6.10.39), DCRADIUS (6.10.11), DCVERTICAL (6.10.12), DELCONSTRAINT (6.10.24) **Overview:**The dimensions of the geometry can be controlled by dimensional constraints:

- Horizontal distance between points (1).
- Vertical distance between points (2).
- Distance between points or the length of lines and polyline segments (3).
- Radius of arcs, circles and arc polyline segments (4).
- Diameter of arcs, circles and arc polyline segments (5).
- Angle between entities or between points and entities (6).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/02DimensionalConstraints
Note: Dimensional constraints are created using thencurrent dimension style.

To edit dimensional constraints

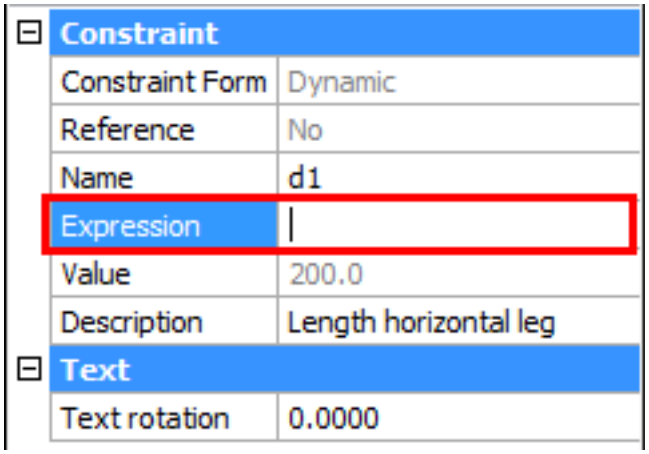
Dimensional constraints can be edited:

- using the DDEDIT (6.10.14) command
- in the Properties (6.24.49) bar
- in the Mechanical Browser (3.2.2)

To edit dimensional constraints in the Properties bar.

1. Select the dimensional constraint. The properties of the entity display in the Properties bar.
2. Type a new value or expression (1.15.6) in the **Expression** field, then press Enter. The readout of the **Value** field changes accordingly. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately
3. (option) Edit the **Name** field.

Figure 1.440



4. (option) Edit the **Description** field.
5. (option) Edit the **Text rotation** field.
6. Press the Esc key to unselect.

Note: Select multiple dimensional constraints simultaneously to set their value equally.

To edit dimensional constraints in the Mechanical Browser:

1. If not already open, open theMechanical Browser (3.2.2). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
2. Select a parameter. The properties the selected parameter display in bottom part of the Mechanical Browser (3.2.2).
3. (option) Select the **Name** field, then type a new name.
4. (option) Select the **Expression** field, then type a new value or expression.
5. (option) Select the **Description** field, then type a new description.
6. (option) Set the **Exposed** field, which specifies whether the parameter is exposed when the drawing is inserted in another drawing using **BmInsert**. The options are:
   - Auto : the parameter is exposed only if it does not depend on other parameters.
   - ON = the parameter is always exposed.
   - OFF = the parameter is never exposed.
7. (option) Set the **Units** property: Undefined, 1-dimensional (linear), 2-dimensional (planar) or 3-dimensiona (volume)l. Select the field, then click the down-arrow button and choose an option.

To edit a single dimensional constraint:

1. Double click the dimensional constraint. You are prompted: _ddedit The Edit Text dialog box displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/02EditTextDialog.png})
2. Type a new value in the **Text** field, then press Enter or click the **OK** button. The new value is applied and the geometry is rebuild accordingly.

Figure 1.441

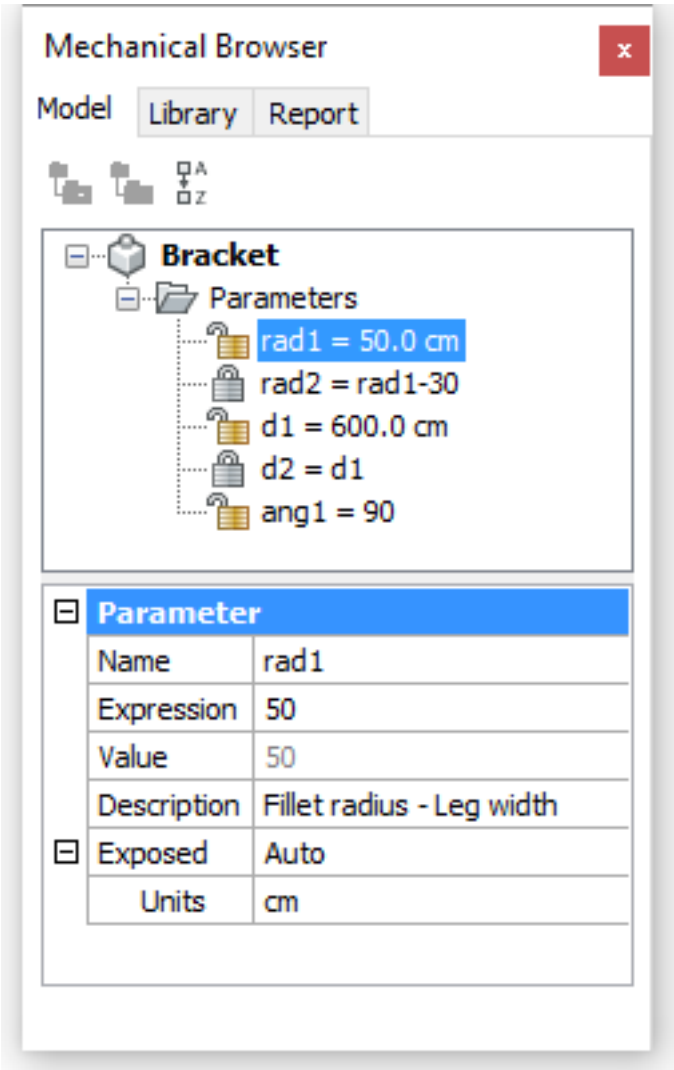
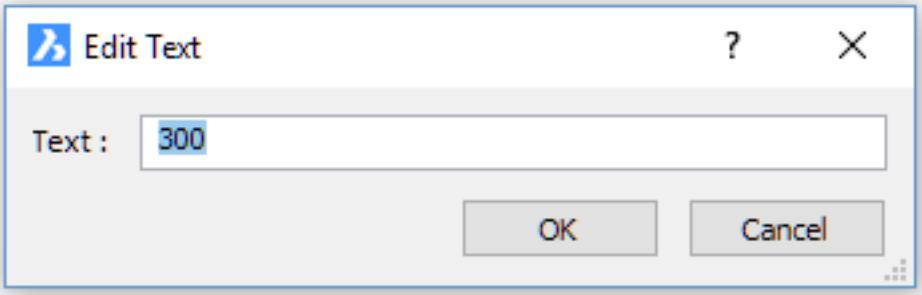


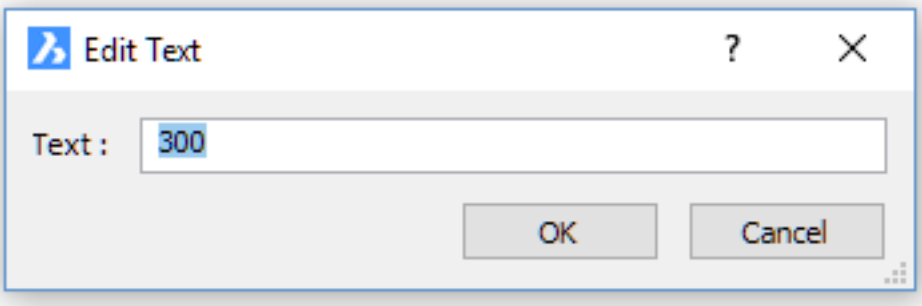
Figure 1.442



To edit multiple dimensional constraints:

1. Type **ddedit** or **ed** in the command bar, then press Enter. You are prompted: Select entity to modify:
2. Click the a dimensional constraint. The Edit Text dialog box displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc
3. Type a new value in the **Text** field, then press Enter or click the **OK** button.
4. (option) Repeat steps 2 and 3 to edit more dimensional constraints.
5. Press Enter or right click to stop editing dimensional constraints. All edited dimensional constraints are updated simultaneously. The geometry is rebuild accordingly.

Figure 1.443



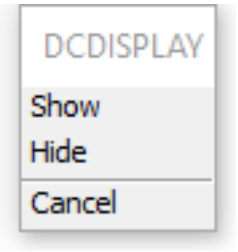
To set the visibility of dimensional constraints

Tools to control the display of dimensional constraints sit on the Dimensional Constraint toolbar. Dimensional constraints are hidden when a drawing is closed and reopened.

To set the display of dimensional constraints of a selection of entities

1. Do one of the following:
   - Click the **Show/Hide Dimensional Constraint** tool button ( ) on the 2D Constraints or Dimensional Constraint toolbar.
   - Choose **Show/Hide Dimensional Constraint** in the Parametric menu.
   - Type **dcdisplay** in the command bar.

   You are prompted: Select Entities.
2. Select the entities, then right click or press Enter. BricsCAD reports the number of selected entities. You are prompted: Enter an option [Show/Hide] <Show>: A prompt menu displays:



3. Do one of the following:
   - Press Enter to display the dimensional constraints of the selected entities.
   - Choose **Hide** or **Show** in the context menu.
   - Type **H** or **S** in the command bar, then press Enter.

To set the visibility of the dimensional constraints of all entities Do one of the following:

- Click the **Show All** tool button ( ) on the Dimensional Constraint toolbar to display the dimensional constraints of all entities.
- Click the **Hide All** tool button ( ) on the Dimensional Constraint toolbar to hide the dimensional constraints of all entities.

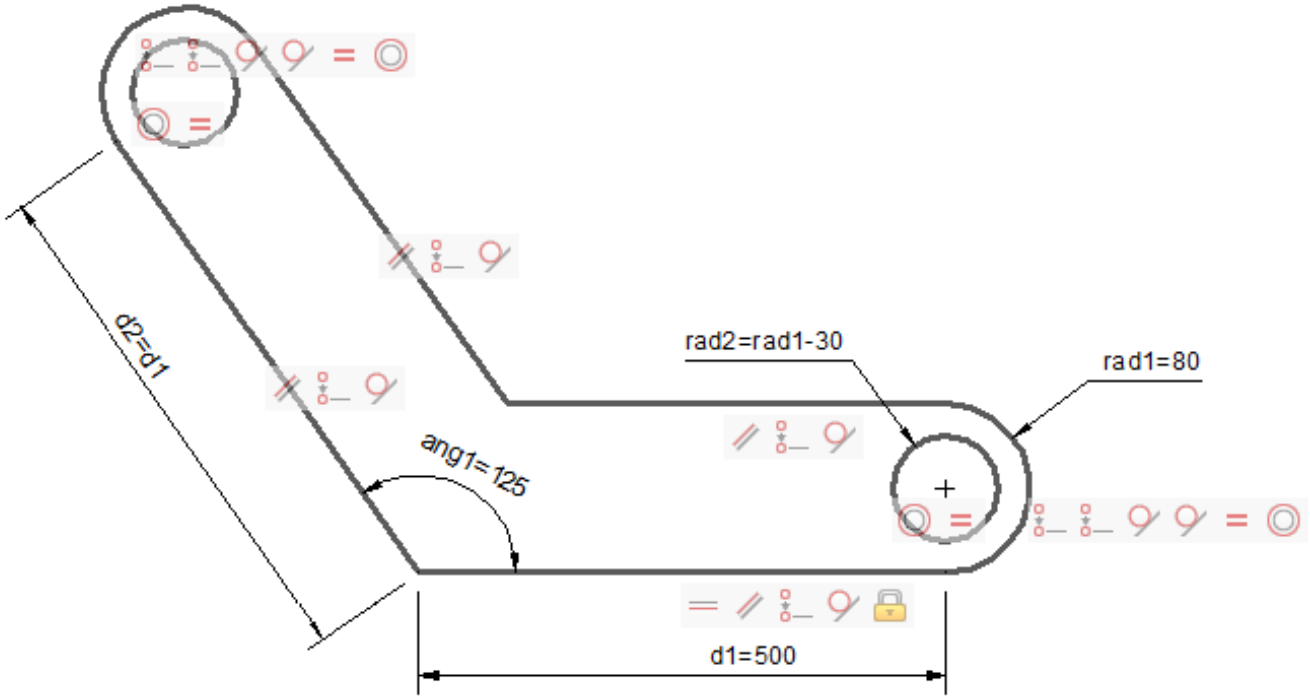
NOTES

The DYNCONSTRAINTMODE (1.13.11) system variable controls the visibility of hidden dimensional constraints when selecting entities:

- ON: Display dimensional constraints when entities are selected.
- OFF: Dimensional constraints remain hidden when entities are selected.

The DYNCONSTRAINTMODE (1.13.11) system variable controls the visibility of hidden dimensional constraints when the constrained entity is selected. When hovering over an entity with a dimensional constraint, the blue constraint glyph ( ) displays if the SELECTIONPREVIEW (1.23.5) system variable is set to 1 or 3

Figure 1.444



Using expressions to define dimensional constraints

Each dimensional constraint is named automatically upon its creation. The naming is as follows: distance: d1, d2, d3, ... radius: rad1, rad2, rad3, ... diameter: dia1, dia2, dia3, .. angular: ang1, ang2, ang3, .. The name of a constraint can be used as a parameter in a mathematical expression to define another dimensional constraint. The dimensions of the bracket below are controlled through the **rad1** radius, **d1** distance and **ang1** angular dimensional constraints. The radius of the hole (**rad2**) and the length of the second leg (**d2**) are defined by expressions. Geometrical constraints force the width of the second leg and the hole in it to be equal to the first leg. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/02UsingExpressionsExample.png})

Creating a new parameter

1. Select either the part name or **Parameters** in the Mechanical Browser (3.2.2), then right click. A context menu displays.
2. Select **Add new parameter** in the context menu. A new parameter is created.
3. Edit the properties of the new parameter. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/

Figure 1.445

Parameter	
Name	v6
Expression	1
Value	1
Description	
Exposed	Auto
Units	

NOTES

- The constants **Pi=3.14...** and **e=2.72...** can be used in expressions. The constants names are not allowed to be used as parameter name and name of 3D constraint.
- The CLEANUNUSEDVARIABLES (6.9.23) command purges parameters that are not used in constraint expressions or linked to dimensions.

Tutorial: 2D Dimensional Constraints Click here (1.5.5) to watch

Using operators in expressions The following operators can be used in expressions:

Operator	Description
+	Addition
-	Subtraction or Negative
	Multiplication
/	Division
^	Exponentiation
%	Modulo or Remainder operator The expression ”5 % 2” would evaluate to 1, because 5 divided by 2 leaves a quotient of 2 and a remainder of 1 .

Expressions are evaluated according to the standard mathematical rules of precedence:

1. Expressions within brackets; innermost sets first.
2. Standard operations order:
   (a) exponent
   (b) multiplication and division
   (c) addition and subtraction
3. Operators of equal precedence from left to right.

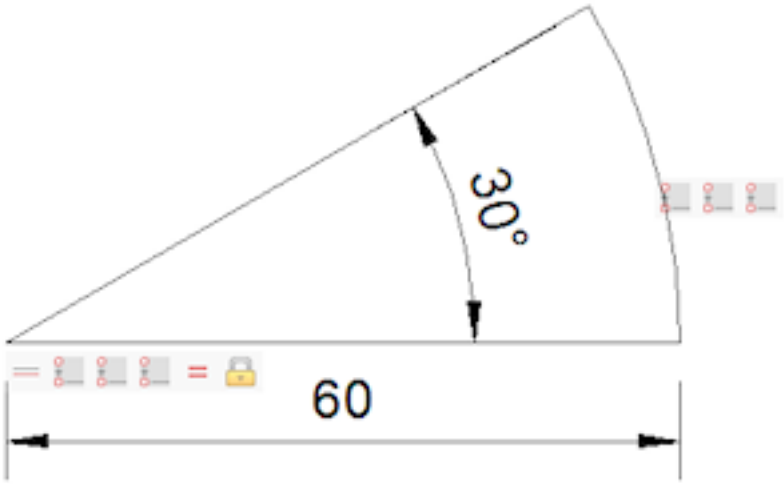
Using functions in expressions The following functions can be used in expressions:

Function	Syntax
Cosine	cos(expression)
Sine	sin(expression)
Tangent	tan(expression)
Arc cosine	acos(expression)
Arc sine	asin(expression)
Arc tangent	atan(expression)
Hyperbolic cosine	cosh(expression)
Hyperbolic sine	sinh(expression)
Hyperbolic tangent	tanh(expression)
Arc hyperbolic cosine	acosh(expression)
Arc hyperbolic sine	asinh(expression)
Arc hyperbolic tangent	atanh(expression)
Square root	sqrt(expression)
Signum function (-1,0,1)	sign(expression)
Round to nearest integer	round(expression)
Truncate decimal	trunc(expression)
Round down	floor(expression)
Round up	ceil(expression)
Absolute value	abs(expression)
Largest element in array	max(expression1;expression2) 1
Smallest element in array	min(expression1;expression2) 1
Degrees to radians	d2r(expression)

Radians to degrees	r2d(expression)
Logarithm, base e	ln(expression)
Logarithm, base	10 log(expression)
Exponent, base e	exp(expression)
Exponent, base 10	exp10(expression)
Power function	pow(expression1;expression2) 1
Random decimal, 0-1	Random

(1) Use the list separator character as defined on your system: , (comma) or ; (semicolon).

Figure 1.446



1.15.7 2D Constraints Overview

Tutorial: 2D Dimensional Constraints Click here to watch Parametric constraints are used to maintain relationships and control dimensions of 2D geometry. There are two types of constraints:

- **Geometrical** constraints control relationships between entities.
- **Dimensional** constraints are used to control the dimensions of an entity, such as length, angle, radius or diameter.

In the example below:

- The width and opening angle are controlled by dimensional constraints.
- Coincident geometrical constraints ( ) are applied to the endpoints of the arc and the polyline and to the center point of the arc and the polyline vertex.
- The polyline segments have an equal ( ) geometrical constraint.
- One segment of the polyline has a horizontal ( ) geometrical constraint.
- The polyline vertex has a fix ( ) geometrical constraint.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/00DoorSwing.png})
Parametric constraints allow to:

- Maintain geometric design intent.
- Make multiple versions of a design by applying different values to dimensional constrains.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/00Roundabout1.png})
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/00Roundabout2.png})
Two versions of the same fully constrained drawing representing a roundabout

NOTE	It is recommended to first apply geometric constraints, then dimensional constraints.

When using constraints a drawing can be in one of three states:

- **Unconstrained**: no entities are constrained. The drawing can be modified using modification commands and procedures.
- **Underconstrained**: some entities are constrained. It might be impossible to apply some changes using modification commands and procedures.
- **Fully constrained**: all possible and relevant geometric and dimensional constraints are applied to the design geometry. The drawing can be modified only by changing the values of the dimensional constraints.

Figure 1.447



Figure 1.448

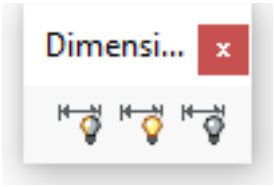
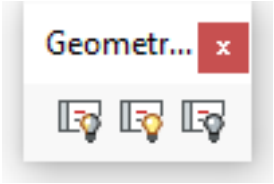


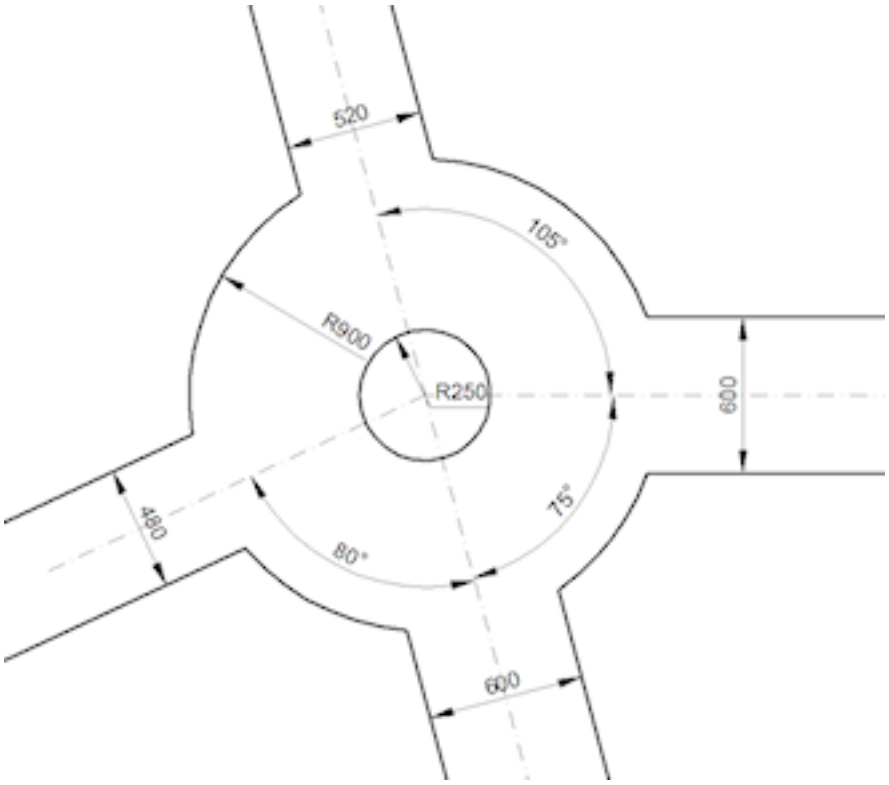
Figure 1.449



Commands and toolbars

2D parametric constraints tools sit on the **2D Constraints** toolbar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/
Constraints.png}) Tools to control the display of constraintbars and geometrical constraints sit on two flyout tool-
bars: **Dimensional Constraint** : (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccura
1DimensionalConstraint.png}) and **Geometric Constraint:** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
2GeometricalConstraint.png}) **Command Icon Description** Geometric Constraints DcDisplay (6.10.9) Allows to set dimensional constraints to be displayed or hidden. Shows all dimensional constraints. Hides all dimensional constraints. GeomConstraint (6.15.16) Applies geometric relationships between entities and/or valid constraint points on entities. GcFix (6.15.6) Constrains points and entities at a fixed position. GcCoincident (6.15.1) Applies a coincident geometrical constraint to two points or constraints a point to an entity. GcConcentric (6.15.3) ===NO VALID URL FOR IMAGE=== Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcCollinear (6.15.2) Forces lines or polyline segments to be collinear. Parallelel (6.15.8) Forces two lines or linear polyline segments to be parallel to each other. GcPerpendicular (6.15.9) Constrains two lines or polyline segments to lie perpendicularly to each other. GcTangent (6.15.12) Constrains two entities to maintain a point of tangency to each other or their extensions. The following entities are accepted: lines, polyline segments, circles, arcs, ellipses or elliptical arcs. GcHorizontal (6.15.7) Constrains lines, linear polyline segments or pairs of points to lie parallel to the X-axis of the current coordinate system. GcVertical (6.15.13) Constrains lines, linear polyline segments or pairs of points to lie parallel to the Y-axis of the current coordinate system. GcSmooth (6.15.10) Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) Constrains two entities or points to lie symmetrically with respect to a selected line. GcEqual (6.15.5) Constrains arcs and circles to the same radius, or lines and polyline segments to the same length. Dimensional Constraints ConstraintBar (6.9.62) Controls the display of the constraint bar of entities. Constraint bars are always hidden when opening a drawing. Shows all constraintbars. Hides all constraintbars. Dimconstraint (6.10.29) Applies a dimensional constraint to an entity or between constraint points on entities. Converts associative dimensions to dynamic dimensions. DcHorizontal (6.10.10) Constrains the horizontal distance (X-distance) between two points with respect to the current co-

Figure 1.450



NOTE	The PARAMETERCOPYMODE (1.13.11) system variable controls copying of constraints and related parameters when entities are copied. 0: Do not copy 2D constraints. 1: Replace all expressions with constants. 2: Use existing parameters; if a parameter is missing, replace it with a constant. 3: Use existing parameters; create a new parameter if a parameter is missing. 4: Use existing parameters; create a new parameter if a parameter is missing, or has a different value.

Figure 1.451

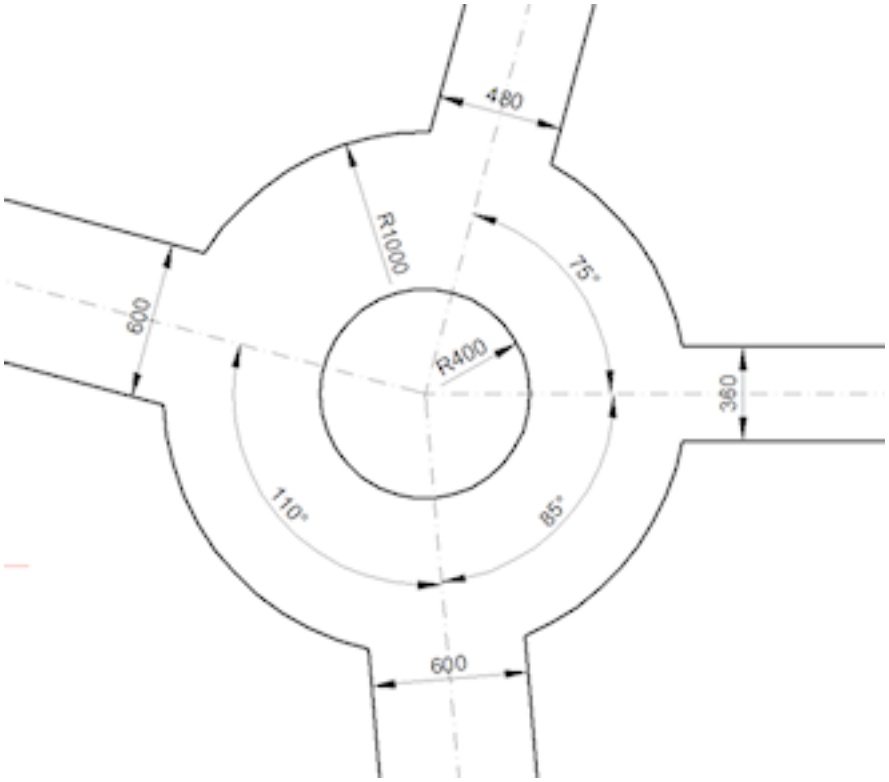
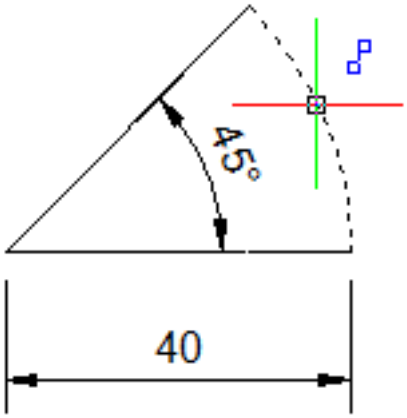


Figure 1.452



1.15.8 Using Geometric Constraints

Commands : GEOMCONSTRAINT (6.15.16), GCCOINCIDENT (6.15.1), GCCOLLINEAR (6.15.2), GCCONCENTRIC (6.15.3), GCCENTER (6.15.4), GCEQUAL (6.15.5), GCFIX (6.15.6), GCHORIZONTAL (6.15.7), GCPARALLEL (6.15.8), GCPERPENDICULAR (6.15.9), GCSMOOTH (6.15.10), GCSYMMETRIC (6.15.11), GCTANGENT (6.15.12), GCVERTICAL (6.15.13) A geometric constraint is a non-numerical relationship between the parts of a geometric figure. Geometric constraints:

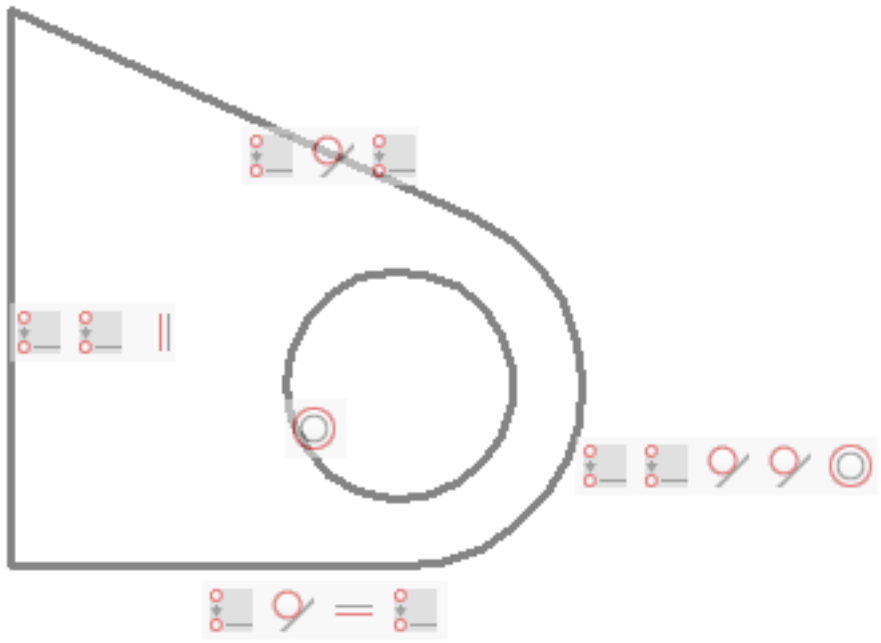
- Associate geometric entities together two by two (coincident, concentric, collinear, parallel, perpendicular, tangent, smooth, symmetric, equal).
- Specify a fixed angle (horizontal, vertical).
- Specify a fixed location (fix).

When a geometric constrained is applied to an entity:

- The position of the entity is adjusted according to the applied constraint.
- An icon displays next to the entity, to indicate the applied constraint. If multiple geometric constraints are applied, the icons are joined to a constraint bar.
- If the SELECTIONPREVIEW (1.23.5) system variable is set to 1 or 3, a blue constraint glyph ( ) displays next to the cursor when you move the cursor over entities that have constraints applied to them.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/01CursorIcon.png})
In the image below the endpoints of the three lines and the arc are joined by coincident constraints. The midpoint of the circle and the arc are made concentric. Two lines are forced to be tangent to the arc. One line has a vertical constraint (= parallel to the Y-axis of the current coordinate system), one line has a horizontal constraint (= parallel to the X-axis of the current coordinate system) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Drawir

Figure 1.453



To set the display of constraint bars

Tools to control the display of constraintbars sit on the Geometric Constraint toolbar. Constraint bars are hidden when a drawing is closed and reopened.

To set the display of constraint bars of a selection of entities

1. Do one of the following:

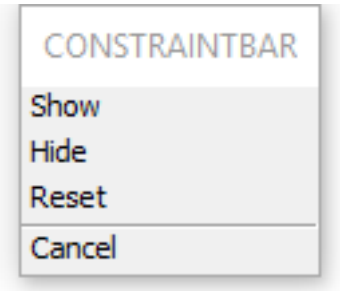
   - Click the **Show/Hide Geometric Constraint** tool button ( ) on the 2D Constraints or Geometric Constraint toolbar.
   - Choose **Show/Hide Geometric Constraint** in the Parametric menu.
   - Type **constraintbar** in the command bar.

   You are prompted: Select Entities.

2. Select the entities, then right click or press Enter. BricsCAD reports the number of selected entities. You are prompted: Select option for constraints [Show/Hide/Reset] <Show>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/01ConstraintBarPromptMe

3. Do one of the following:
   - Press Enter to display the constraint bar of the selected entities.
   - Choose **Hide** or **Show** in the context menu.
   - Type **H** or **S** in the command bar, then press Enter.

Figure 1.454



To set the display of constraint bars of all entities Do one of the following:

- Click the **Show All** tool button ( ) on the Geometric Constraint toolbar to display all constraint bars.
- Click the **Hide All** tool button ( ) on the toolbar Geometric Constraint to hide all constraint bars.

NOTE

The value of the CONSTRAINTBARDISPLAY (1.13.11) system variable controls the visibility of geometric constraints:

- 1: Display constraint bars when geometrical constraints are added.
- 2: Display hidden dimensional constraints when the constrained entities are selected.

To control the position of a constraint bar

By default constraint glyph bars are created close to the midpoint of the entity and are kept at that relative position when the entity position changes. You can drag the constraint bar to a different location. This new relative position is then maintained until the **Reset** option of the CONSTRAINTBAR (6.9.62) command restores the default position of the constraint bar.

To relocate a constraint bar:

1. Place the cursor on the constraint bar.
2. Press and hold the left mouse button to move the constraint bar.
3. Release the left mouse button at the desired location.

To restore the default position of the constraint bars:

1. Do one of the following:

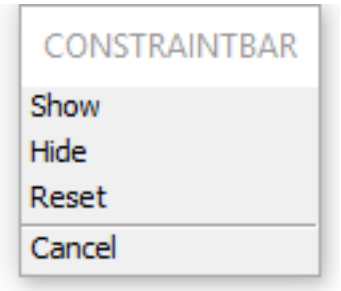
   - Click the **Show/Hide Geometric Constraint** tool button ( ) on the 2D Constraints or Geometric Constraint toolbar.
   - Choose **Show/Hide Geometric Constraint** in the Parametric menu.
   - Type **constraintbar** in the command bar.

   You are prompted: Select Entities.

2. Select the entities, then right click or press Enter. BricsCAD reports the number of selected entities. You are prompted: Select option for constraints [Show/Hide/Reset] <Show>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/01ConstraintBarPromptMe

3. Do one of the following:
   - Choose **Reset** in the context menu.
   - Type **R** in the command bar, then press Enter.

Figure 1.455



Working with the constraint bar

Using the constraint bar you can:

- Control a constraint. Move the cursor over a constraint icon. A tooltip displays, indicating the constraint type (1) The associated entity highlights (2) The corresponding icon on the constraint bar of the associated entity highlights (3) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/01Cons
- Delete a constraint. Move the cursor over the constraint icon, then right click. Click **Delete** . The constraint is deleted and the icon is removed from this constraint bar and from the constraint bar of the corresponding entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/01ConstraintBar
- Hide the constraint bar of an entity. Move the cursor of the constraint bar, then click the **Close** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/01ConstraintBar03.png})

Figure 1.456

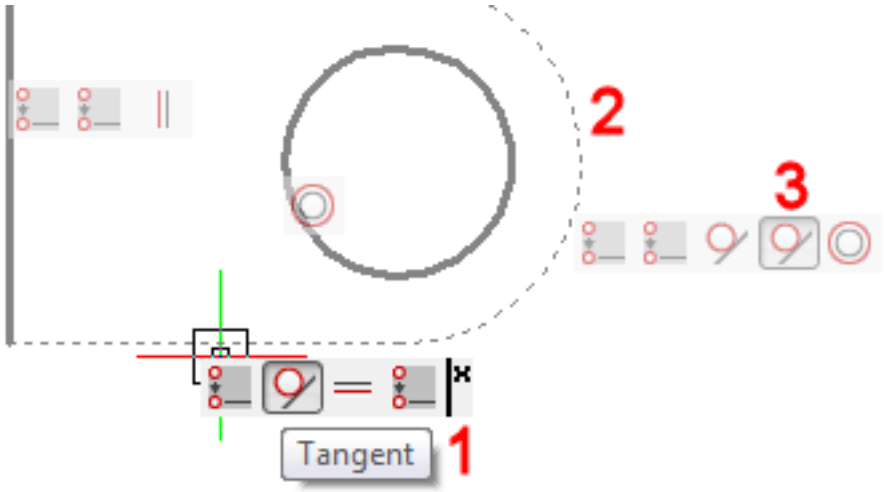


Figure 1.457

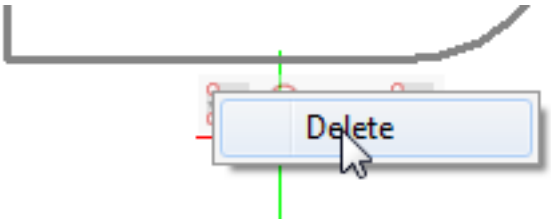
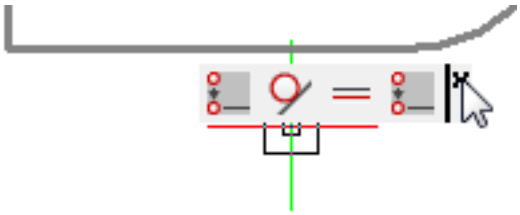


Figure 1.458



Deleting constraints

To delete a single constraint: right click the constraint icon, then click the **Delete** button. To delete all constraints from a selection set:

1. Click the **Delete 2D Constraints** icon ( ) on the 2D Constraints toolbar. You are prompted: Select entities to delete all constraints:
2. Select (1.23.6) the entities you want to delete the constraints from.
3. Right click to stop selecting entities and delete all constraints from the selection set.

Using fix constraints

A fix geometric constraint ( ) forces a point or an entity to a fixed location.

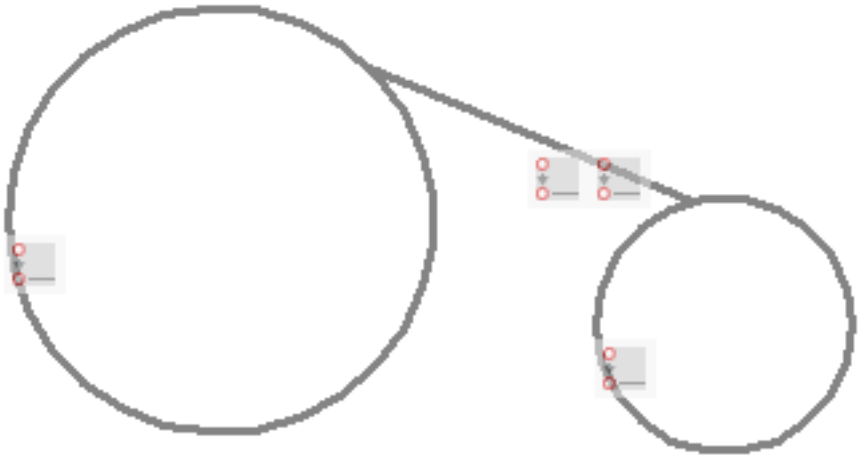
- If a fix geometric constraint is applied to a line or a polyline segment, the angle of the line or polyline segment is fixed. The endpoints can still be moved.
- If you fix an arc or a circle, the center point is fixed too.

Note: It is recommended to apply a fix constraint to important geometric features in your design. This prevents the geometry from moving unexpectedly when you edit the design.

Using coincident constraints

A coincident geometric constraint can be applied to two points or to a point and an entity. The endpoints of lines, polylines, splines, rays and arcs (circular and elliptical) and the center point of circles, ellipses and arcs (circular and elliptical) are accepted. These points can be made coincident (= to lie on the entity or on the extension of the entity) with a line, a polyline segment, a circle or an arc (circular and elliptical). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/01Coincident.png}) Figure: Coincident constraints between the endpoint of a line and two circles. If the endpoints or center points of entities already coincide, the **Autoconstraint** option of the **GcCoincident** command automatically applies coincident constraints to such points.

Figure 1.459



To apply a coincident constraint between a point and an entity:

1. Click the **Coincident** tool button ( ) on the 2D Constraints toolbar. You are prompted: Select first point or [Entity/Autoconstrain] <Entity>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS

2. Do one of the following:

   - Press Enter.
   - Choose **Entity** in the prompt menu.

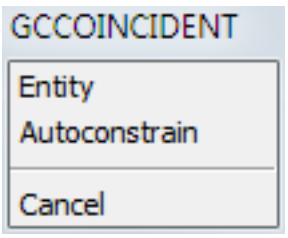
   You are prompted: Select an entity:

3. Select the entity. The entity highlights. You are prompted: Select point or [Multiple]:

4. Snap to the endpoint of an entity or the center point of a circle or an arc. The selected point is forced to lie on the selected entity (or its extension).

Note: If you choose **Multiple** in step 3, you can select multiple points.

Figure 1.460



Usage of a coincident constraint between a point and an entity (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/
By applying coincident constraints between the pentagon vertices and the circle, and equal constraints between one side and the four other sides, the circle radius defines the size of the pentagon. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/01TangentLines.png}) Coincident constraints between the endpoints of the tangent lines and the circles prevent the tangent lines to extend beyond the tangent points. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/01AxisLine
Coincident constraints are used to:

- force the endpoints of the red line to lie on the bold lines;
- force the midpoint of the red line to lie on the dash-dot line.

The bold lines and the dash-dot line have a perpendicular constraint with the red line. As a result the dash dot line will always lie in the middle of the two bold lines.

Figure 1.461

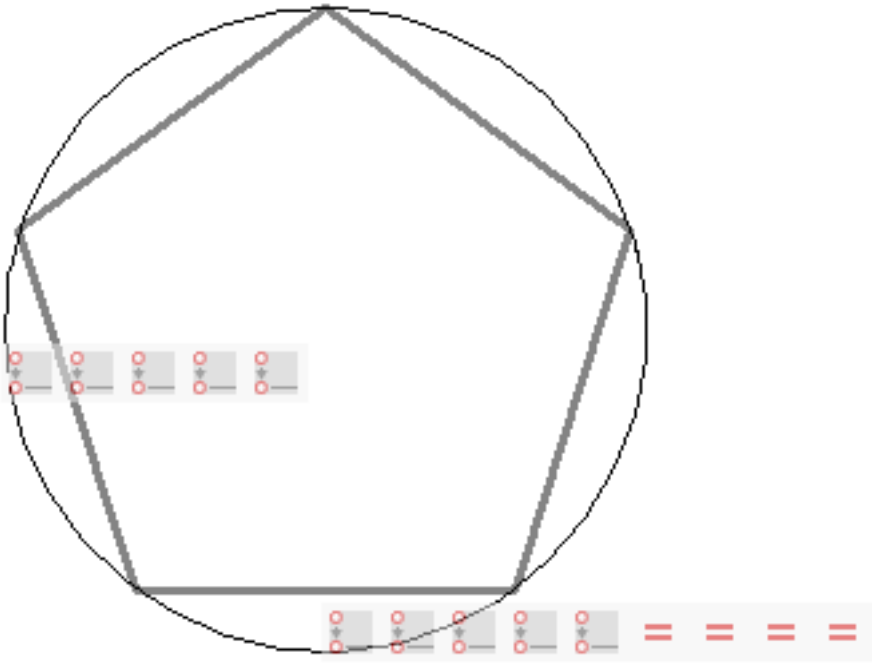


Figure 1.462

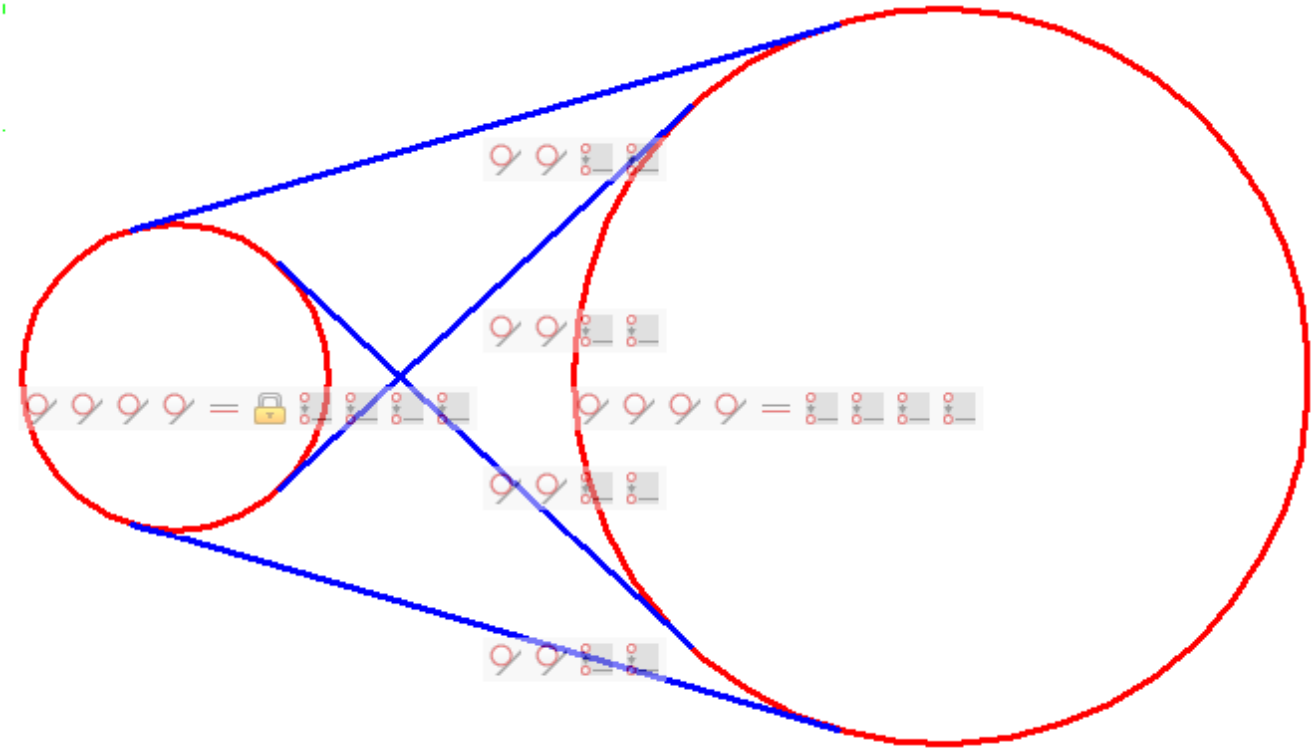
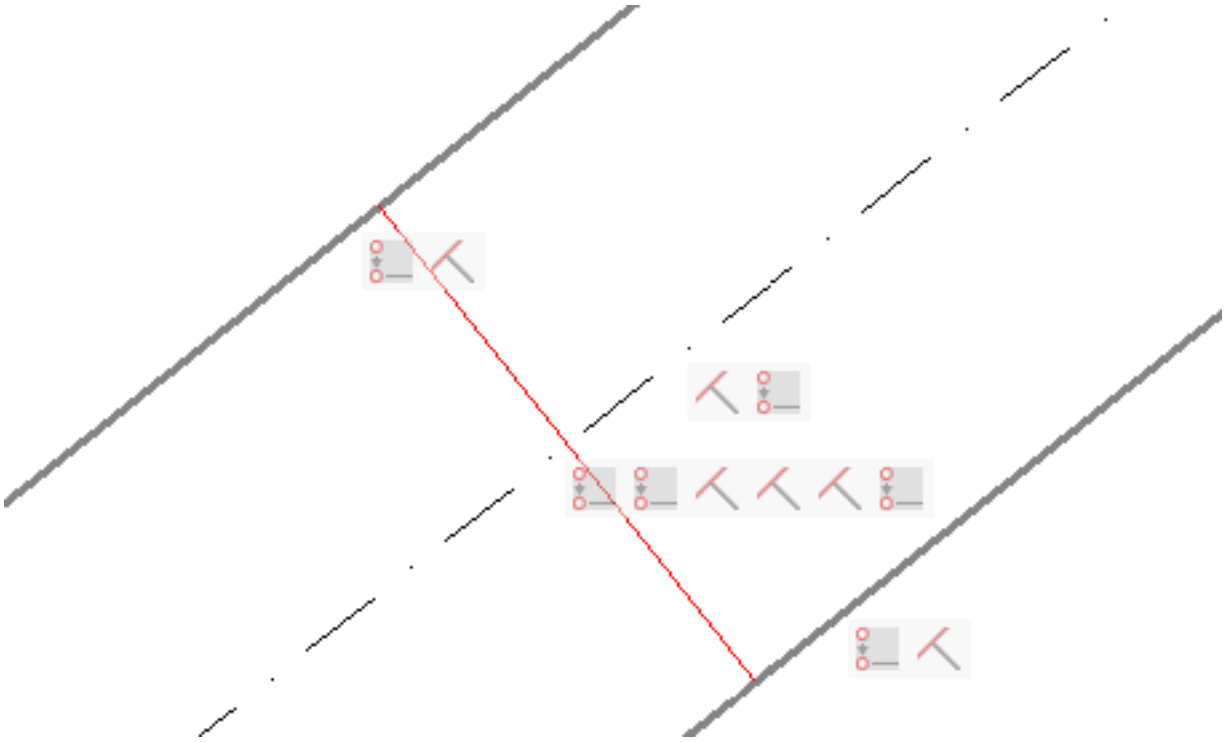


Figure 1.463



Using horizontal and vertical constraints

Horizontal and vertical geometric constraints force two points, a line or a polyline segment to be parallel to the X-axis (horizontal) or the Y-axis (vertical) of the current coordinate system. Therefore these constraints can be used to keep lines at a fixed angle, rather than only horizontal or vertical.

To constrain a line to a fixed angle

1. Align the X-axis of the coordinate system to the desired angle.
   - Type **UCS** in the command bar, then press Enter.
   - Choose **Z** in the prompt menu or type **Z** , then press Enter.
   - Type the angle, then press Enter.

2. Click the **Horizontal** tool button ( ) on the 2D Constraints toolbar. You are prompted: Select an entity or [2Points] <2Points>:

3. Click the line or polyline segment. The selected entity is constrained to be parallel to the X-axis of the current UCS.

4. (option) Restore the WCS: Type **UCS** in the command bar, then press Enter twice.

Note: Use the **Vertical** tool in the above procedure to constrain entities to be parallel to the Y-axis of the current UCS.

1.15.9 Dynamic Dimensions

Dynamic dimensions provide an interface at the cursor position to specify the length and angle of the linear entity being created or grip-edited. As such dynamic dimensions help you to focus on the drawing area. The behavior and appearance of dynamic dimensions is controlled through a series of system variables and user preferences.

Defining the Dynamic Dimensions Settings

1. Right click the **DYNMODE** field in the Status Bar (1.12.2), then choose **Settings** in the context menu. The **Dynamic input** settings in the Settings (1.13.11) dialog display: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help

2. Define the **DYNMODE** system variable, to switch the display of dynamic dimensions and/or dynamic input on/off. Please notice that dynamic input at the pointer is not implemented yet.

3. Define the **DYNDIGRIP** system variable, to select the dynamic dimension types to be displayed. Distance fields have a blue background; angle fields have a green background.
   - Resulting length: total length of the line or polyline segment (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/
   - Extended length: incremental length of the line (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
   - Absolute angle: angle relative to the X-axis of the current UCS (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom
   - Relative angle: angle relative to the original angle of the linear entity (\vref {fig:C:/Dev/Task-2448/images/staticbrics
   - Arc radius: radius of arcs and circles (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGu

4. Define the **DYNDIVIS** system variable to choose the number of dynamic dimensions that display simultaneously.
   - Only the first dynamic dimension (**DYNDIVIS** =0): hit the TAB key to display and edit the next dynamic dimension, as defined by the **DYNDIGRIP** system variable.
   - Only the first two dynamic dimension (**DYNDIVIS** =1): hit the TAB key to edit the next dynamic dimension, as defined by the **DYNDIGRIP** system variable and display the next plus one.
   - All dynamic dimensions as defined by **DYNDIGRIP** (**DYNDIVIS** =2): hit the TAB key to edit the next dynamic dimension

5. Set the dynamic dimensions user preferences to define the appearance (color, linetype and position) of dynamic dimensions.

NOTE	If bitcode 16 (= format dynamic dimensions) of the PROPUNITS (1.15.4) system variable is set, dynamic dimensions are formatted using a dedicated unit. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03 Automatic unit conversion during length input can be used when bitcode 1 of PROPUNITS (**??**) (Format length properties) is set. E.g. when INSUNITS (**??**) is set to ”mm”, entering ”2m” in a length property field will result in entering [2000.0000 mm]. Entering ”1ft” will give [304.8000 mm]. If no unit is specified in the entry field, the unit as specified by INSUNITS (**??**) is used.

Figure 1.464

Drawing	
Drafting	
Drawing units	
Coordinate input	
Dynamic input	
Dynamic input mode	0x0003 (3)
negative	☐ Switch all off temporarily
1	☑ Dynamic input at pointer (not supported)
2	☑ Editable dynamic dimensions
4	☐ Tracking dynamic dimensions
Show dynamic dimensions	0x001F (31)
1	☑ Resulting length
2	☑ Extended length
4	☑ Absolute angle
8	☑ Relative angle
16	☑ Arc radius
Dynamic dimension visibility	[1] Only the first two dynamic dimensions
Dynamic dimension hover color	142
Dynamic dimension hot color	142
Dynamic dimension distance	1
Dynamic dimension linetype	[0] Continuous
Transparency of dynamic input fields	90
Dynamic dimension aperture	20

Figure 1.465

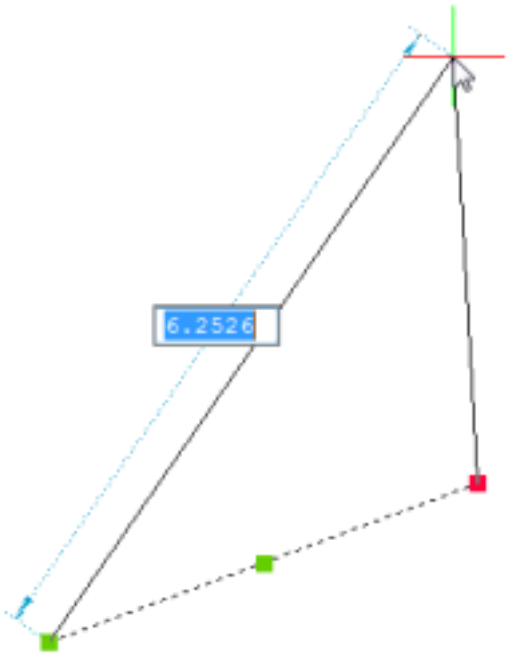


Figure 1.466

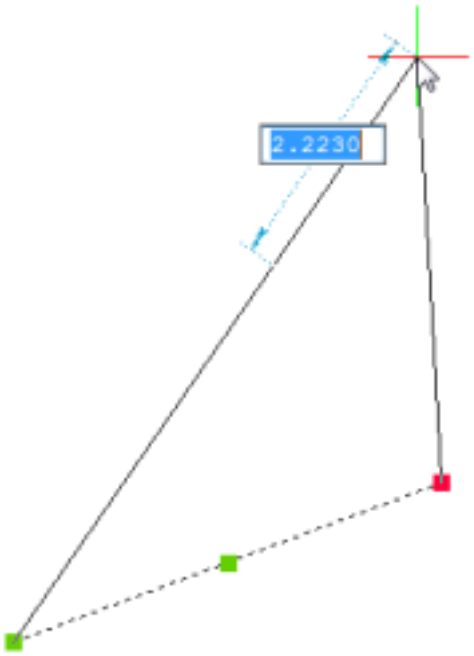


Figure 1.467

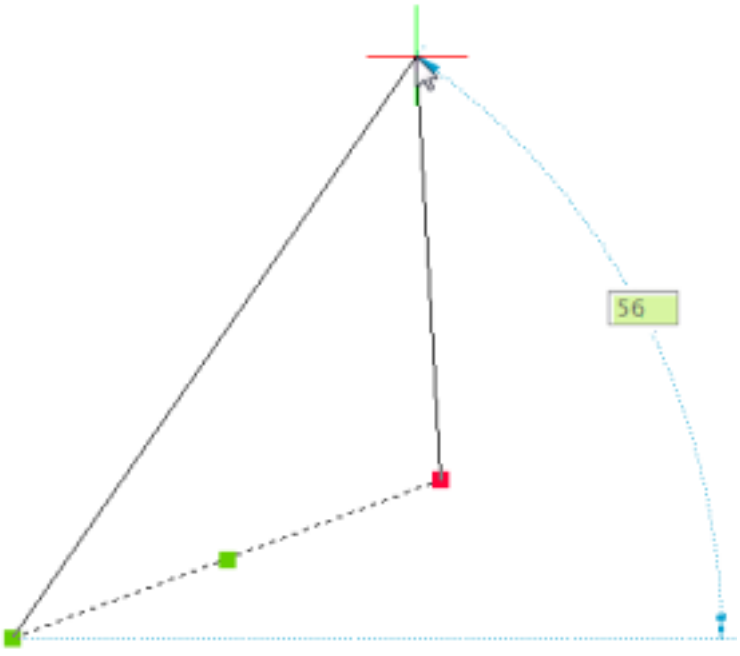
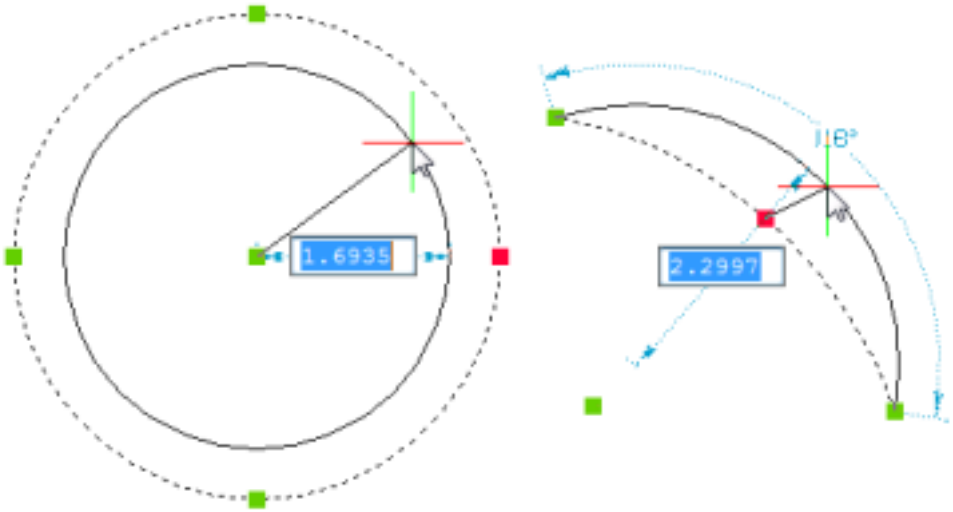


Figure 1.468



Figure 1.469



Using dynamic dimensions when creating entities

1. Launch the command to create a 2D entity (e.g. Line and **DYNDIVIS** = 1 or 2).
2. Specify the first point, then move the cursor to display the dynamic dimensions. The **Length** dynamic dimension highlights.
2. Do one of the following:
   - Type the desired length of the line, then hit the TAB key to jump to the **Angle** dynamic dimension. The **Length** dynamic dimension is locked and displays in red and the **Angle** dynamic dimension is selected.
   - Type the desired length of the line, then click to create the line at the angle defined by the cursor position.
   - Hit he TAB key to jump to the **Angle** dynamic dimension.
4. Do one of the following:
   - Type the desired angle, then hit the Enter key to create the line.
   - Type the desired angle, then hit the TAB key to return to the **Length** dynamic dimension.
   - Hit the Enter key to create the line at the angle defined by the cursor position.
   - Hit the TAB key to return to the **Length** dynamic dimension.

NOTES

- If **DYNDIVIS** = 0, only one dynamic dimension displays. Hit the TAB key to display the next field.
- If you type a value in the **Length** field, and add a comma (,) the content of the **Length** field is copied to the command line and the @-character is placed in front automatically, which allows you to specify the next point using relative coordinates (1.15.20) with respect to the previous point.

Using dynamic dimensions to grip-edit entities

1. Select the entity or entities. All grips on the selected entities display.
2. Click the grip you want to edit, then move the cursor. Depending on the value of the DYNDIVIS (1.13.11) system variable one, two or all dynamic dimensions display simultaneously . The first dynamic dimension highlights.
3. Do one of the following:
   - Type the desired value for the currently highlighted dynamic dimension, then hit the TAB key to jump to the next dynamic dimension. The dynamic dimension is locked and displays in red.
   - Type the desired value for the currently highlighted dynamic dimension, then press Enter to stop.
   - Hit the TAB key to jump to the next dynamic dimension.
4. Repeat step 3 or press Enter to stop.

Tutorial: Dynamic Grips and Tips Click here (1.4.5) to watch

Using dynamic dimensions to measure entities

1. Select the entity or entities. All grips on the selected entities display.
2. Hover the cursor over a grip. The grip highlights and all dimensions related to the selected grip display. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/11DynDim
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/11D
Dynamic dimensions used to measure a rectangle (left) and a line (right).

   (c) (option) Click to start editing the selected grip.

   Note: Dynamic dimensions when hovering over a non-selected grip display in 2DWireframe visual style only.

Figure 1.470

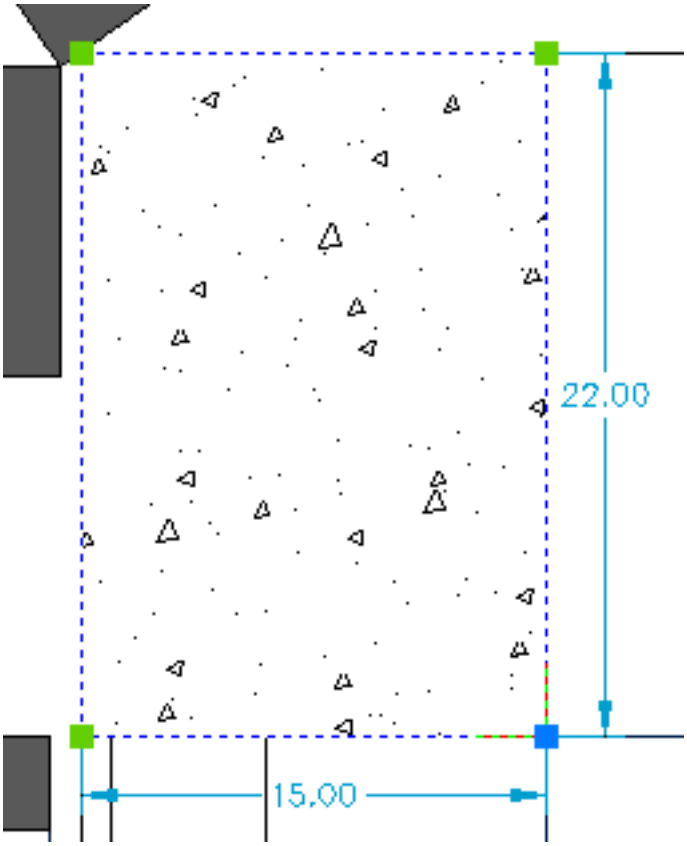
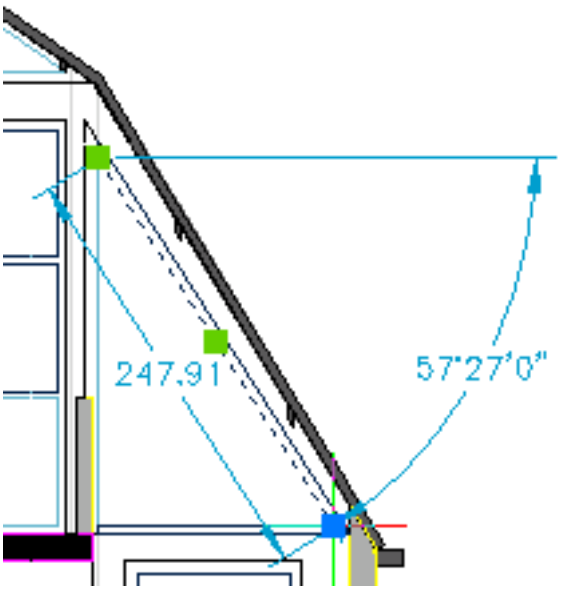


Figure 1.471



Using dynamic dimensions with tracking points

If the **Tracking dynamic dimensions** option of the DYNMODE (1.13.11) system variable is set, dynamic dimensions display when entity snap tracking points (1.15.17) or temporary tracking points (1.15.15) are acquired.

Tutorial: Drawing Accurately: Dynamic dimensions Click here (1.4.3) to watch

Figure 1.472

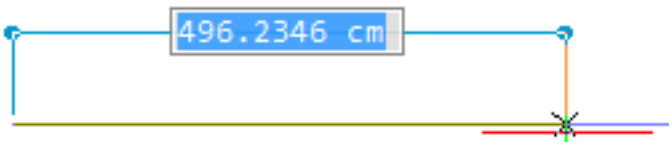


Figure 1.473

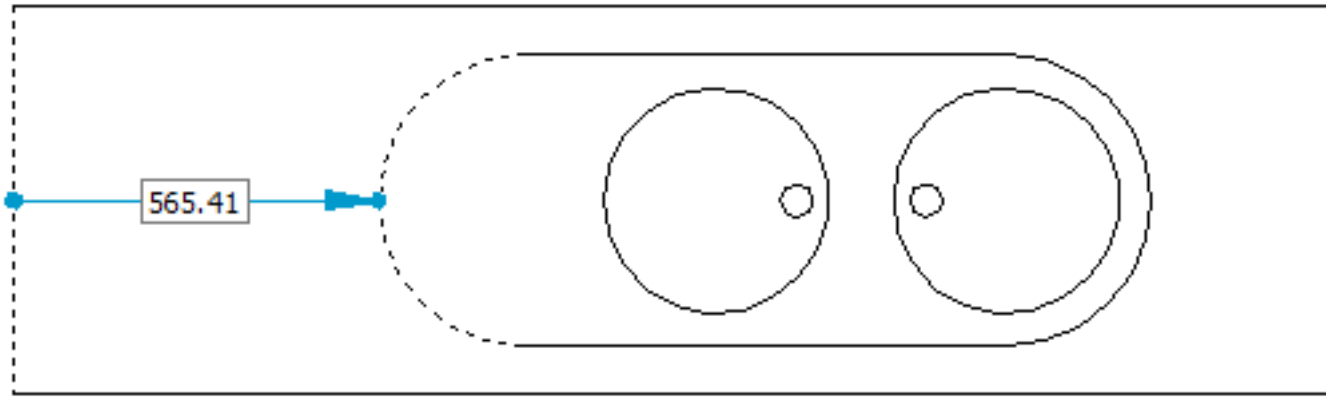


Figure 1.474



1.15.10 Nearest distance

The Nearest Distance feature allows to dynamically view and modify the distance between two selected entities or subentities. Do the following:

1. Select the first entity or subentity*.
2. Select the second entity or subentity*. The nearest distance displays. The arrowhead points to the second entity.
3. (option) Double click the distance field and type a value. The second entity is moved.

* Press and hold the Ctrl-key to select a subentity. The Nearest Distance feature can be used to:

- Measure the distance between two entities or subentities.
- Adjust the distance between a (sub)entity and a block. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS 1.png})
- Modify the dimensions of a rectangle. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Dra 2.png})
- Adjust the distance between 3D solids. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/D 1.png})
- Edit the width of a 3D solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccu 2.png})
- Measure and edit the height of 3D solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/D 3.png})
- ...

Figure 1.475

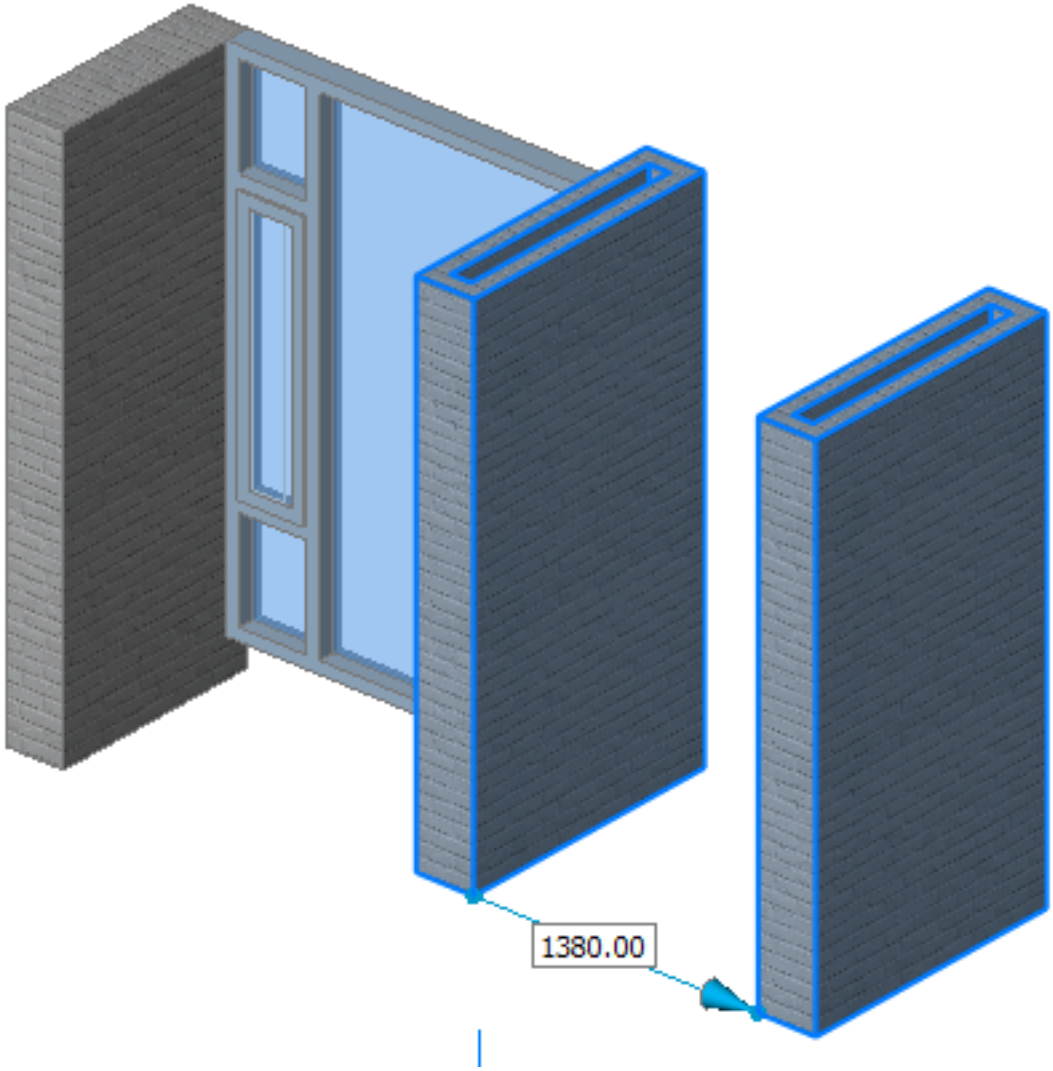


Figure 1.476

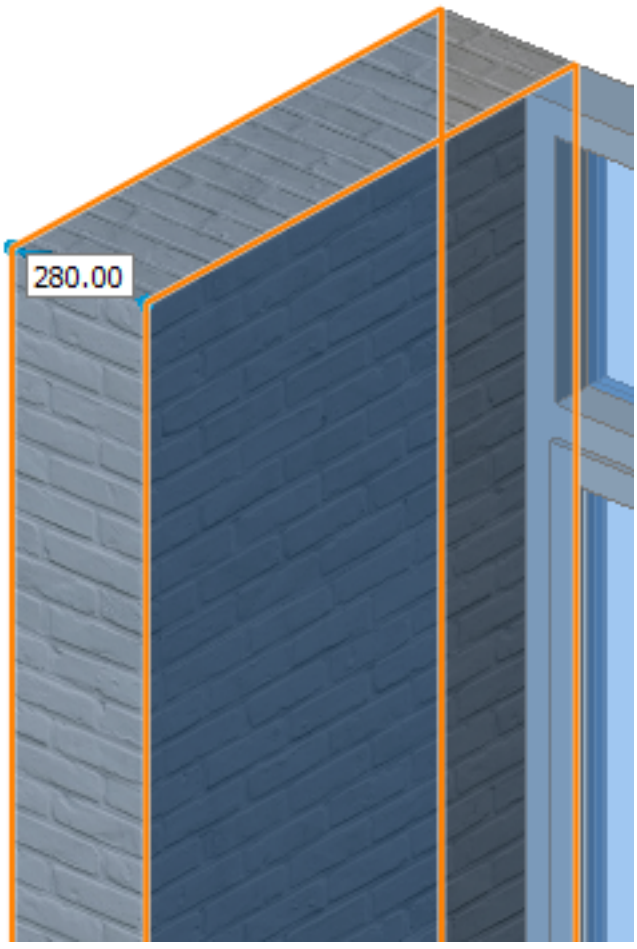
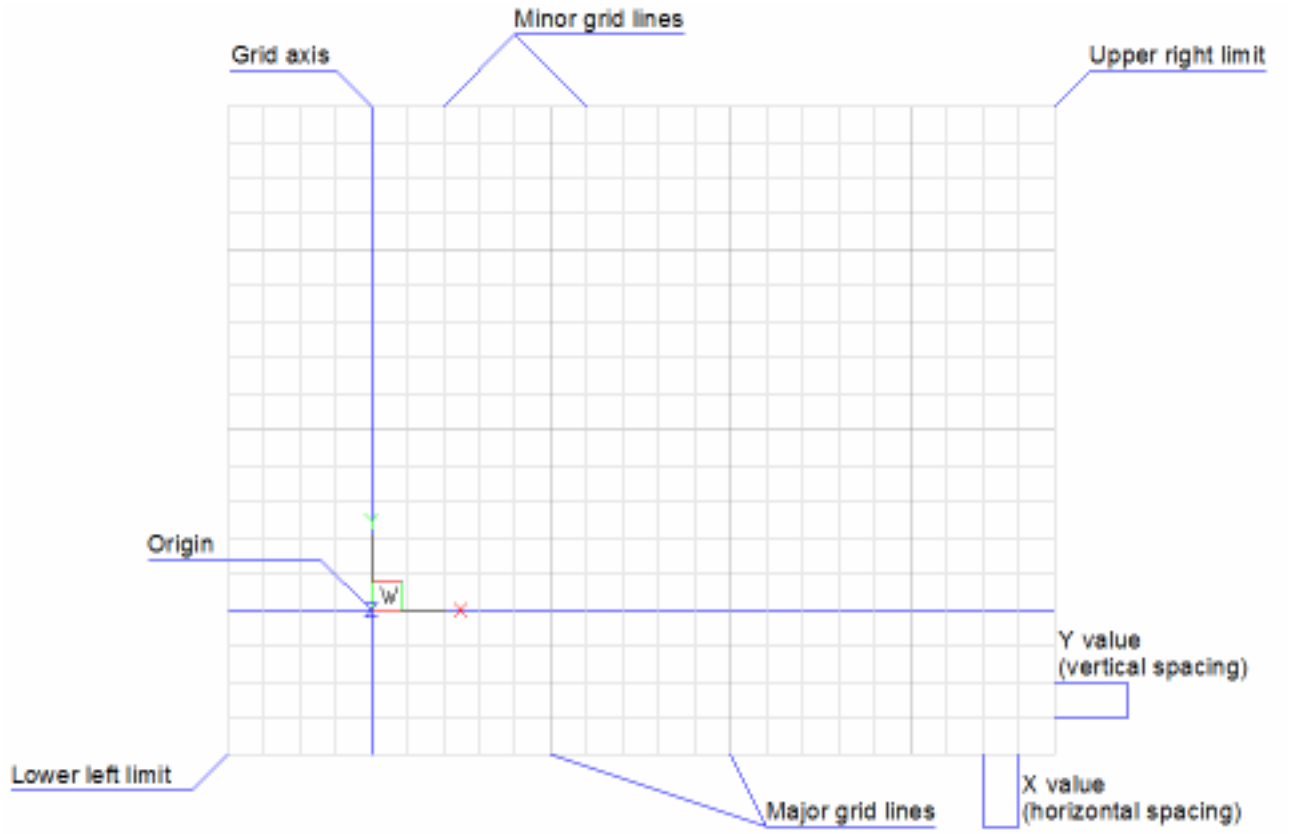


Figure 1.477



Figure 1.478



1.15.11 Using Snap and Grid

Commands : SNAP (6.27.38), GRID (6.15.21) and LIMITS (6.20.30) Grid and snap help you to draw fast and accurately. A grid is a set of evenly spaced lines or dots that serve as a visual distance reference. The GRIDSTYLE (1.13.11) system variable controls whether grid lines or dots (6.15.21) display. The GRIDDISPLAY (1.13.11) system variable controls the display of the grid (6.15.21). The grid can also indicate how far the drawing limits extend. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/02Grid.png}) The snap feature creates a set of evenly spaced, invisible magnetic points**,** which make the crosshairs move in even increments. Both grid and snap are like the intersection points of the lines on a piece of grid paper. Grid points are for visual reference only and they do not print. Snap constrains the points that you can pick with the mouse.

- Click the **Grid** tool ( ) on the Settings toolbar or the **Grid** field in the Status Bar (1.12.2), or press **Ctrl+G** to toggle grid display on/off.
- Click the **Snap** tool ( ) on the Settings toolbar or the **Snap** field in the Status Bar (1.12.2), or press **Ctrl+B** to toggle snap on/off. The setting is saved through the SNAPMODE system variable. If SNAPTYPE = 2, adaptive grid snap (1.15.12) is used and SNAPMODE is denied.
- Both **snap** and **grid** can be set differently in each viewport (1.16.13).
- The **Snap** option of the **Grid** command equals the grid spacing to the current snap spacing.
- Grid lines or dots do not print.

Tutorial: Drawing Accurately: Snap and Grid Click here (1.4.1) to watch

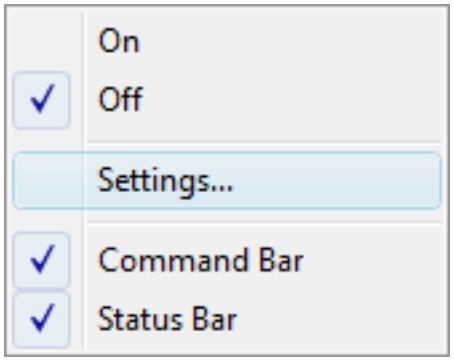
Using the adaptive grid

When SNAPTYPE = 2, the movement of the cursor is controlled by an adaptive grid of which the spacing depends on the current zoom factor of the view and the value of AdaptiveGridStepSize user preference. The value of this preference is expressed in pixels. As a result the step size of the adaptive grid increases when you zoom out, and decreases when you zoom in. The step size of the ruler in the Manipulator (1.23.10) also depends on theAdaptive-GridStepSize user preference.

Display the Snap and Grid settings

Do one of the following:

Figure 1.479



- In the Status Bar (1.12.2), right click on the **Snap** field, then choose **Settings** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/02SnapSe

- Open the Settings (1.13.11) dialog, then click the **Drawings** button. Under **Drafting** , expand **Coordinate Input** and **Snap/Grid** .

To synchronize snap and grid spacing

1. Type **Grid** in the command bar, then press Enter. You are prompted: Grid is on: OFF/Aspect/<Grid spacing (synced with snap)>:
2. Type **s** , then press Enter.
3. Type **Snap** in the command bar, then press Enter. You are prompted: Snap is on, x and y = 0.2000: OFF/Rotate/Style/Aspec spacing>:
4. Type a new value for the snap spacing, then press Enter. The value is applied to both snap and grid.

Note: The grid spacing remains in sync with the snap spacing until you set the grid spacing differently using the **Grid** command or in the **Settings** dialog.

To set the Drawing Limits

See the LIMITS (6.20.30) command.

To toggle the Drawing Limits

Click the **Drawing Limits** tool button ( ) on the Settings toolbar to toggle the LIMCHECK (1.13.11) system variable on/off. If LIMCHECK = ON, it is not possible to draw outside the drawing limits. The appearance (pressed or unpressed) of the **Drawing Limits** tool button reflects the current status of the **Drawing Limits.** The **Display beyond LIMITS area** option of the GRIDDISPLAY (1.13.11) system variable defines wether or not the grid display indicates the drawing limits.

Using Isometric Snap

You can use the Isometric snap and grid option to create two-dimensional isometric drawings. With the isometric option, you can draw a simulated three-dimensional view on a two-dimensional plane, much the same as you might draw on a piece of paper. Do not confuse isometric drawings with three-dimensional drawings. The isometric option always uses one of three preset planes, which are denoted as **Left**, **Right** and **Top**. You cannot alter the arrangement of these planes. If the Snap Angle is 0, the three isometric axes are 30 degrees, 90 degrees, and 150 degrees. When you set the **Snap Style** setting to **Isometric Snap** and then set the **Snap Isometric Pair** setting to **Left**, **Top** or **Right**, the snap intervals, grid, and crosshairs align with the selected plane. The grid is always shown as isometric and uses y-coordinates to calculate the grid spacing. If the Orthogonal Mode (1.15.2) is active, the movement of the crosshairs is constrained to the current isometric plane. Press the **F5** function key to toggle the **Snap Isometric Pair** setting.

1. Display the **Snap/Grid** settings in the Settings (1.13.11) dialog: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he

2. Turn on the **Orthogonal Mode** .

Figure 1.480

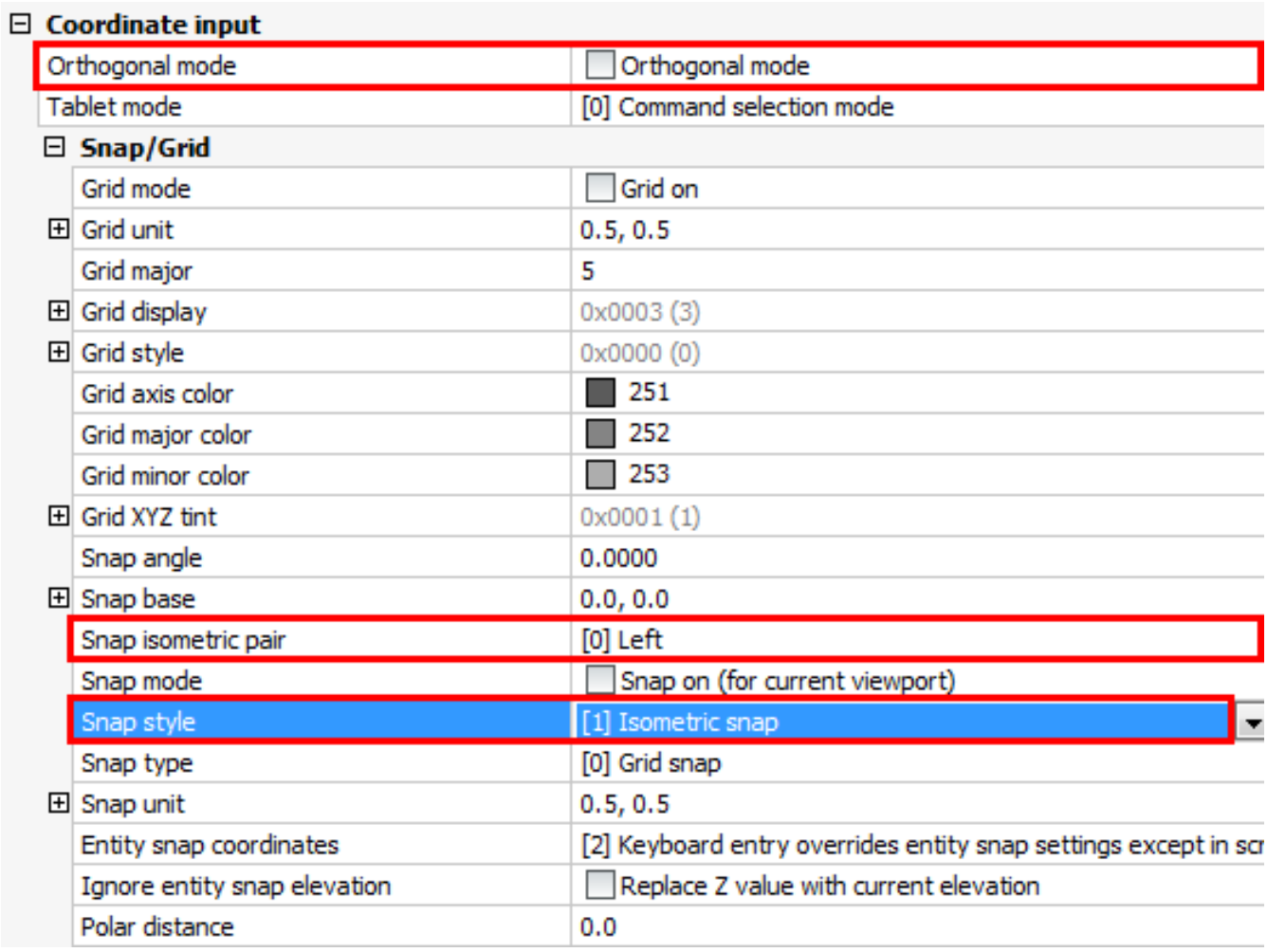
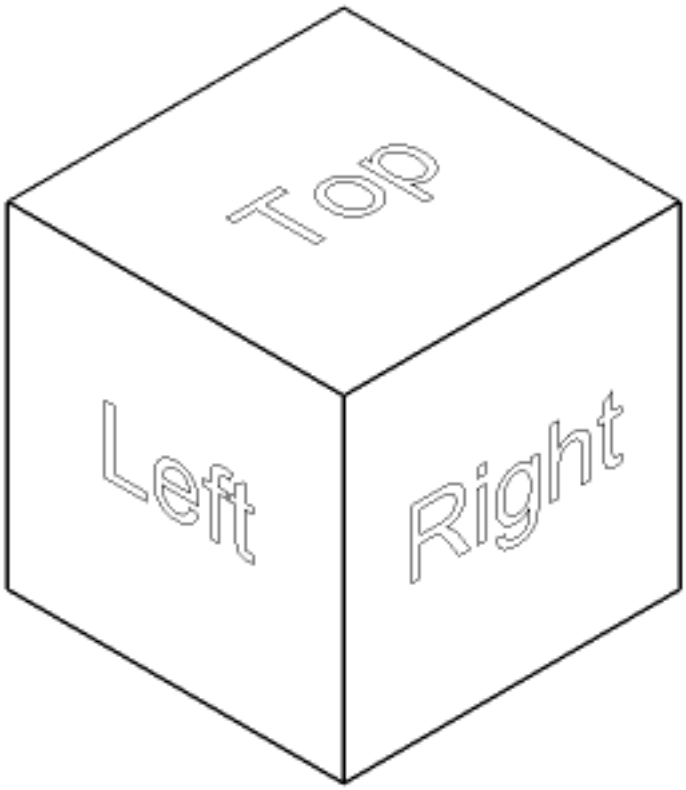

Coordinate input
Orthogonal mode
Orthogonal mode
Tablet mode
[0] Command selection mode
Snap/Grid
Grid mode
Grid on
Grid unit
0.5, 0.5
Grid major
5
Grid display
0x0003 (3)
Grid style
0x0000 (0)
Grid axis color
251
Grid major color
252
Grid minor color
253
Grid XYZ tint
0x0001 (1)
Snap angle
0.0000
Snap base
0.0, 0.0
Snap isometric pair
[0] Left
Snap mode
Snap on (for current viewport)
Snap style
[1] Isometric snap
Snap type
[0] Grid snap
Snap unit
0.5, 0.5
Entity snap coordinates
[2] Keyboard entry overrides entity snap settings except in scr
Ignore entity snap elevation
Replace Z value with current elevation
Polar distance
0.0

Figure 1.481



3. Use the SNAP command to set the **Snap Style** setting to **Isometric Snap** .
4. Press the **F5** function key to set the appropriate drawing plane: **Top**, **Left** or **Right**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/02SnapGridIsometricPlane

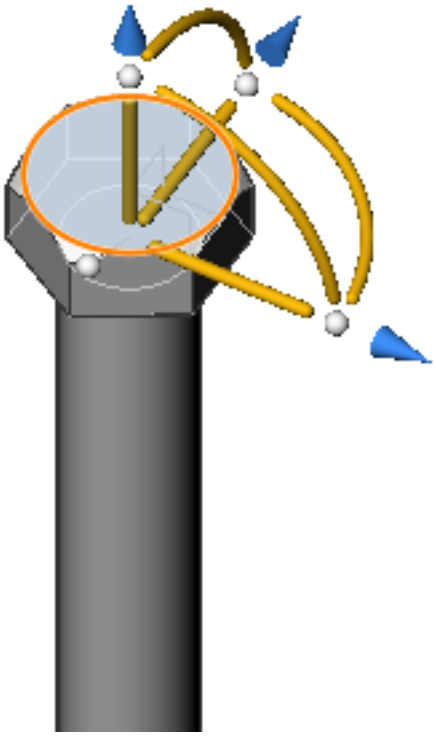
Setting the Snap Angle

You can rotate a dotted grid using the Snap Angle setting (if **Snap Angle** is different from zero, grid lines do not display).

1. Display the **Snap/Grid** settings in the Settings (1.13.11) dialog:
2. Select the **Snap Angle** setting.
3. Type a new value in the Snap Angle setting field.
4. Close the Settings dialog.

Note: You can also set the **Snap Angle** by typing **snapang** in the command bar. When in a command, type **'snapang** (with an apostrophe in front), to set the **Snap Angle** variable transparently (= without interrupting the running command).

Figure 1.482



1.15.12 Adaptive grid snap

The adaptive grid snap feature lets you create, edit, nudge (1.15.13) and manipulate (6.21.2) entities using concise values without keyboard entry.The snap step size is adapted according to the current screen zoom factor. The adaptive grid snap is active if **SNAPTYPE = 2**. This system variable overrides **SNAPMODE**. The AdaptiveGridStep-Size user preference controls the smallest available step size; default = 4 pixels. Do the following:

1. Move the cursor over an entity or subentity, then shortly hold down the left mouse button (= long left click). The Manipulator widget displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAc

2. Move the cursor over the axis in the direction you want to move the entity. The selected axis highlights; the other parts of the Manipulater are faded. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/

3. Move the cursor. The Ruler displays. The current displacement distance displays in a dynamic field. The grid snap resolution automatically adapts according to the zoom factor.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAdaptiveGrid1.png})	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
Ruler divisions at 2 mm	Ruler divisions at 5 mm

4. (option) Zoom in or out to adjust the dynamic grid.

5. Do one of the following:
   - Click to conclude.
   - Type a value in the dynamic field, then press Enter.

Figure 1.483

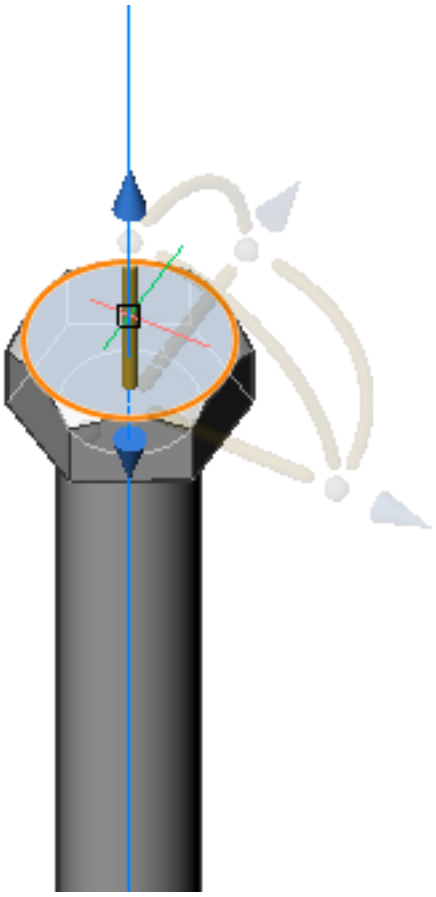


Figure 1.484

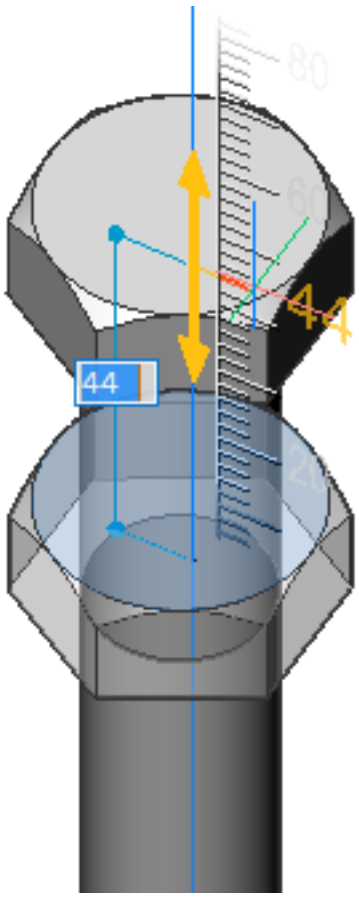
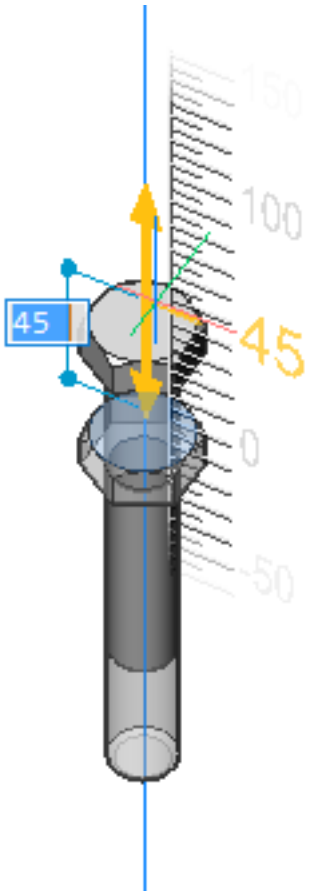


Figure 1.485



1.15.13 Nudge

The Nudge drawing aidlets you move selected entities or subentities in small increments along the X, Y, or Z axis using the keyboard.

Using Nudge

1. Select one or more entities or subentities.
2. (option) Hold the Ctrl key down, then use the left/right arrow keys to move the selection along the X axis of the current coordinate system. The command line reads:NUDGE: offset along the X axis of the UCS: 10*
3. (option) Hold the Ctrl key down, then use the up/down arrow keys to move the selection along the Y axis of the current coordinate system. The command line reads:NUDGE: offset along the Y axis of the UCS: 10*
4. (option) Hold the Shift key down, then use the up/down arrow keys to move the selection along the Z axis of the current coordinate system. The command line reads:NUDGE: offset along the Z axis of the UCS: 10*

*The step size used by the NUDGE functionality is based on the adaptive grid snap (1.15.12) resolution. Zoom in/out to decrease/increase the step size.

Figure 1.486

Entity 3d snap mode	0x000A (10)
1	☐ Turn off all 3d snaps
2	☑ Vertex
4	☐ Midpoint of edge
8	☑ Center of face
16	☐ Knot
32	☐ Perpendicular to planar face
64	☐ Nearest to face
32768	☐ Intersection with face

1.15.14 3D Entity Snaps

Commands: 3DSONAP (6.3.8), -3DOSNAP (6.3.9),ZCENTER (6.35.1),ZINTERSECTION (6.35.2), ZKNOT (6.35.3), ZMIDPOINT (6.35.4), ZNEAREST (6.35.5), ZNONE (6.35.6), ZPERPENDICULAR (6.35.8), ZVERTEX (6.35.9) In addition to the 2D Entity Snaps (1.15.15), a series of 3D entity snap modes apply to points on 3D entities such as solids, surfaces, splines and meshes. The current 3D Entity Snap modes are saved through the 3DOSMODE system variable as the sum of the biticodes of the selected snap modes. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
Pointcloud entity snaps are not yet implemented.

Name	Icon	Marker	Description
Center			Snaps to the center of 3D faces.
Intersection			Snaps to the intersections of linear entities, edges, and polar or entity snap tracking lines with faces.
Knot			Snaps to knots (fit points) onsplines (6.27.65).
Midpoint			Snaps to the midpoint of a face edge.
Nearest			Snaps to a point on the face of a 3D entity that is nearest to the cursor.
None			Disables all 3D snap modes.
Perpendicular			Snaps to a point perpendicular to a face on a 3D entity.

Vertex			Snaps to the closest vertex of a 3D entity and the control vertices of splines (6.27.65).

To set the 3D Entity Snap modes

- Click the buttons on the**3D Entity Snaps**toolbar. The buttons of the currently active entity snap modes are pressed. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Rendering/3DEntitySnapsToolb

  - When no command is running clicking a tool button toggles the snap mode on/off.
  - When executing a command clicking a tool button sets the snap mode for the next point, temporalily disabling all other selected snap modes.

- Run the3DSONAP (6.3.8)or-3DOSNAP (6.3.9)the commands.

Figure 1.487



1.15.15 Entity Snaps

Commands: 2DINTERSECTION (6.2.1), 3DINTERSECTION (6.3.6), NEAREST (6.22.2), CENTER (6.9.3), ENDPOINT (6.13.7), EXTENSION (6.13.20), FROM (6.14.11), GCENTER (6.15.4), INSERTION (6.17.16), INTERSECTION (6.17.20), MIDPOINT (6.21.16), MTP (6.21.36), NONE (6.22.8), PARALLEL (6.24.4), PERPENDICULAR (6.24.22), POINT (6.24.31), QUADRANT (6.25.7), TANGENT (6.29.9) Entity snaps enable you to quickly select exact geometric points on existing entities without having to know the exact coordinates of those points. With entity snaps, you can select the end point of a line or arc, the center point of a circle, the intersection of any two entities, or any other geometrically significant position. You can also use entity snaps to draw entities that are tangent or perpendicular to an existing entity. You can use entity snaps any time you need to specify a point. You can work with entity snaps in one of two ways:

- Enable a running entity snap that remains in effect until you turn it off by choosing an entity snap when no other command is active.
- Enable a one-time entity snap for a single selection by choosing an entity snap when another command is active. You can also use a one-time entity snap to override a running entity snap.

When using entity snaps, the program recognizes only visible entities or visible portions of entities. You cannot snap to entities on layers that have been turned off or to the blank portions of dashed lines. If the **APBOX** system variable is on, the entity Snap Aperture Box is added to the crosshairs when one or more entity snaps are active. When you move the cross hairs, the program snaps to the snap point closest to the center of the Snap Aperture Box. The **Snap Marker** indicates the current snap point. The DRAGSNAP (1.13.11) system variable controls the snap behavior while 'dragging', providing an increased WYSIWIG experience. DRAGSNAP controls whether rubberband dynamics are displayed at the current cursor location or at the current entity snap location. DRAGSNAP applies to all modify commands that display dynamics, such as COPY, PASTECLIP, PASTEBLOCK, MOVE, ROTATE, MIRROR, SCALE and STRETCH.

- **DRAGSNAP = 0** (Default): dragged entities display at the cursor location.
- **DRAGSNAP = 1**: dragged entities display at the current entity snap location.

The OSOPTIONS (1.13.11) system variable suppresses entity snaps on certain entity types. The value is stored as a bitcode using the sum of the values of the selected options.

- 1: Entity snapping is disabled for hatch entities.
- 2: Entity snapping is disabled for geometry with negative Z values when using a dynamic UCS.
- 4: Entity snapping is disabled for endpoints of dimension extension lines.

Note: Press the **TAB** key, without moving the mouse, to cycle through all possible entity snaps. If the **DYNMODE** variable has a positive value (Dynamic Dimensions (1.15.9) switched ON) entity snap cycling is not possible. Click the **DYNMODE** field in the Status Bar (1.12.2) to toggle the display of dynamic dimensions.

To define the Entity Snap settings

1. Open the Settings dialog (1.13.11).
2. In the Settings dialog, expand the **Program Options** settings class.
3. Under **Program Options** expand the **Display** settings group, then scroll down to the **Entity Snap** settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/03EntitySn

User Preference	Title	Description
SnapMarkerSize	Snap marker size	Sets the size of the snap marker. (default size is 6)
SnapMarkerThickness	Snap marker thickness	Sets the thickness of the snap marker. (default thickness is 2)
SnapMarkerColor	Snap marker color	Sets the color of the snap marker. (default color = 20)

3dSnapMarkerColor	3d snap marker color	Sets the color of the 3d snap marker (default color = 5 (Blue). See 3D Entity Snaps (1.15.14). If OSNAPZ = 1 (Ignore entity snap elevation) the 3d snap marker displays also at the the identified 3D point, which is then projected onto the XY plane of the current UCS at the current Elevation. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 The 3d snap marker (1) displays the identified 3d point, the snap marker (2) displays at the current Elevation height.
DisplaySnapMarkerInAllViews	Snap marker in all views	If multiple viewports are open, enables the display of the snap marker in all viewports.
DisplayTooltips	Snap tooltips	Enables the display of the Entity Snap tooltips.

4. Under **Drawing** expand the **Drafting** settings group then go to **Coordinate Input** > **Entity Snaps**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/03EntitySnapSettings2.png

System Variable	Title	Description
OSMODE	Entity snap mode	Sets the entity snap modes and toggles entity snap on/off.
3DOSMODE	3D entity snap mode	Sets the entity snap modes for 3D entity features such as the midpoint of an edge or the center of a face.
OSNAPCOORD	Entity snap coordinates	Determines whether entity snaps override keyboard coordinate entry.
OSNAPZ	Ignore entity snap elevation	Replaces the Z-coordinate of the snapping point with the current value of the ELEVATION system variable.
OSOPTIONS	Entity snap options	Suppresses entity snaps on hatches and/or dimension extension lines or negative z-values in dynamic UCS mode.
APBOX	Entity snap aperture box	Toggles the display of the aperture box (2).
APERTURE	Entity snap aperture	Sets the size of the Autosnap aperture box. (default size is 10).
DGNOSNAP	DGN entity snap	Enables entity snapping to entities in DGN underlay files.
DWFOSNAP	DWF entity snap	Enables entity snapping to entities in DWF underlay files.
PDFOSNAP	PDF entity snap	Enables snapping to the geometry in PDF underlay files.

Tutorial: Drawing Accurately: Snap and Grid Click here (1.4.1) to watch

Figure 1.488

Snap marker size	6
Snap marker thickness	2
Snap marker color	20
3d snap marker color	Blue
Auto tracking vector color	171
Snap marker in all views	Display snap marker in all views
Locked layer fade control	50
Xref database fade control	70
Reference editing fade control	50
Snap tooltips	Enable snap tooltips

Figure 1.489

Entity snaps	
Entity snap mode	0x0003 (3)
Entity 3d snap mode	0x0000 (0)
Entity snap coordinates	[2] Keyboard entry overrides entity snap settings except in scripts
Ignore entity snap elevation	Replace Z value with current elevation
Entity snap options	0x0007 (7)
1	Entity snap ignores hatches
2	Entity snap ignores negative Z values in Dynamic UCS mode
4	Entity snap ignores end points of dimension extension lines
Entity snap aperture box	Display Aperture box
Entity snap aperture	10
Dgn entity snap	Enable DGN entity snap
Dwf entity snap	Enable DWF entity snap
Pdf entity snap	Enable PDF entity snap

To set the Entity Snaps

Do one of the following:

- Click the buttons on the Entity Snaps toolbar. The buttons of the currently active entity snap modes are pressed. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/0

- Press and hold the Shift key, then right click and select the Entity Snap mode in the context menu. The icons of the currently active Entity Snap modes are outlined. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V

When no command is active, the above procedures toggle the Entity Snap modes on/off. When a command is active, the above procedures set a 'one shot' override of the current Entity Snap modes. Alternatively you can type a single letter shortcut as indicated in the table below.

Figure 1.490

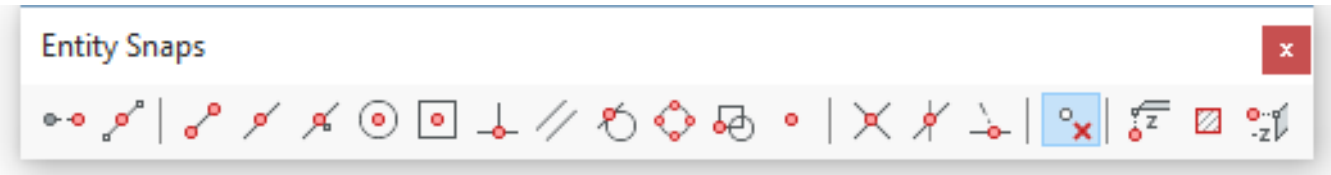
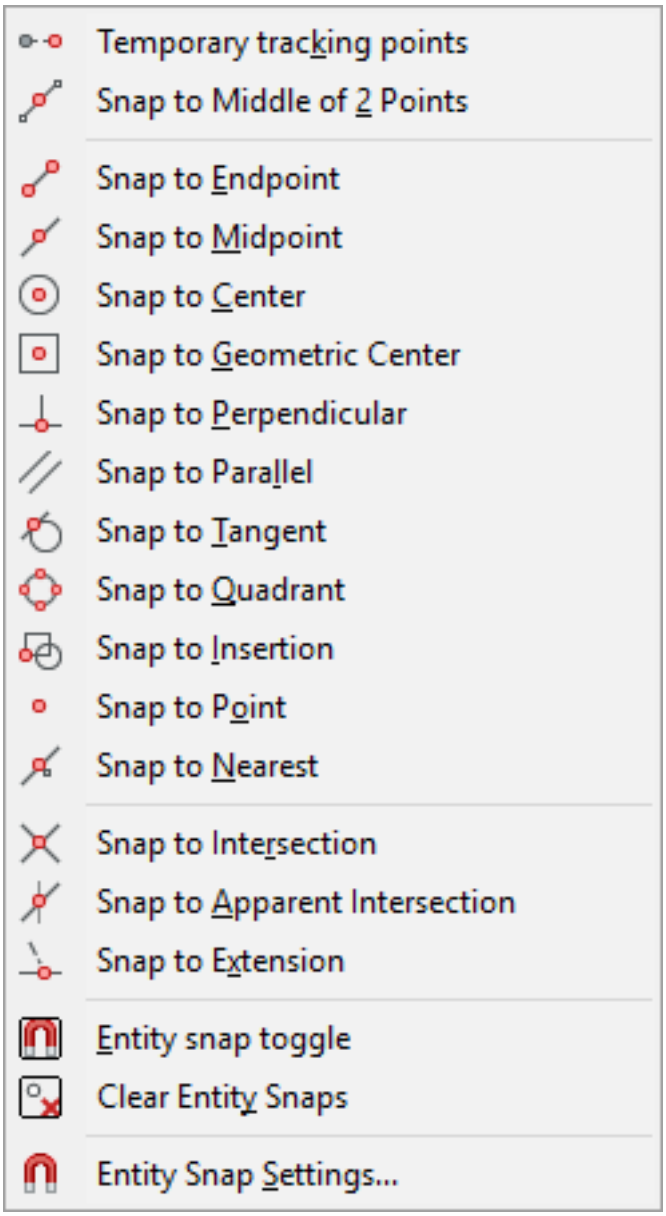


Figure 1.491


Temporary tracking points
Snap to Middle of 2 Points
Snap to Endpoint
Snap to Midpoint
Snap to Center
Snap to Geometric Center
Snap to Perpendicular
Snap to Parallel
Snap to Tangent
Snap to Quadrant
Snap to Insertion
Snap to Point
Snap to Nearest
Snap to Intersection
Snap to Apparent Intersection
Snap to Extension
Entity snap toggle
Clear Entity Snaps
Entity Snap Settings...

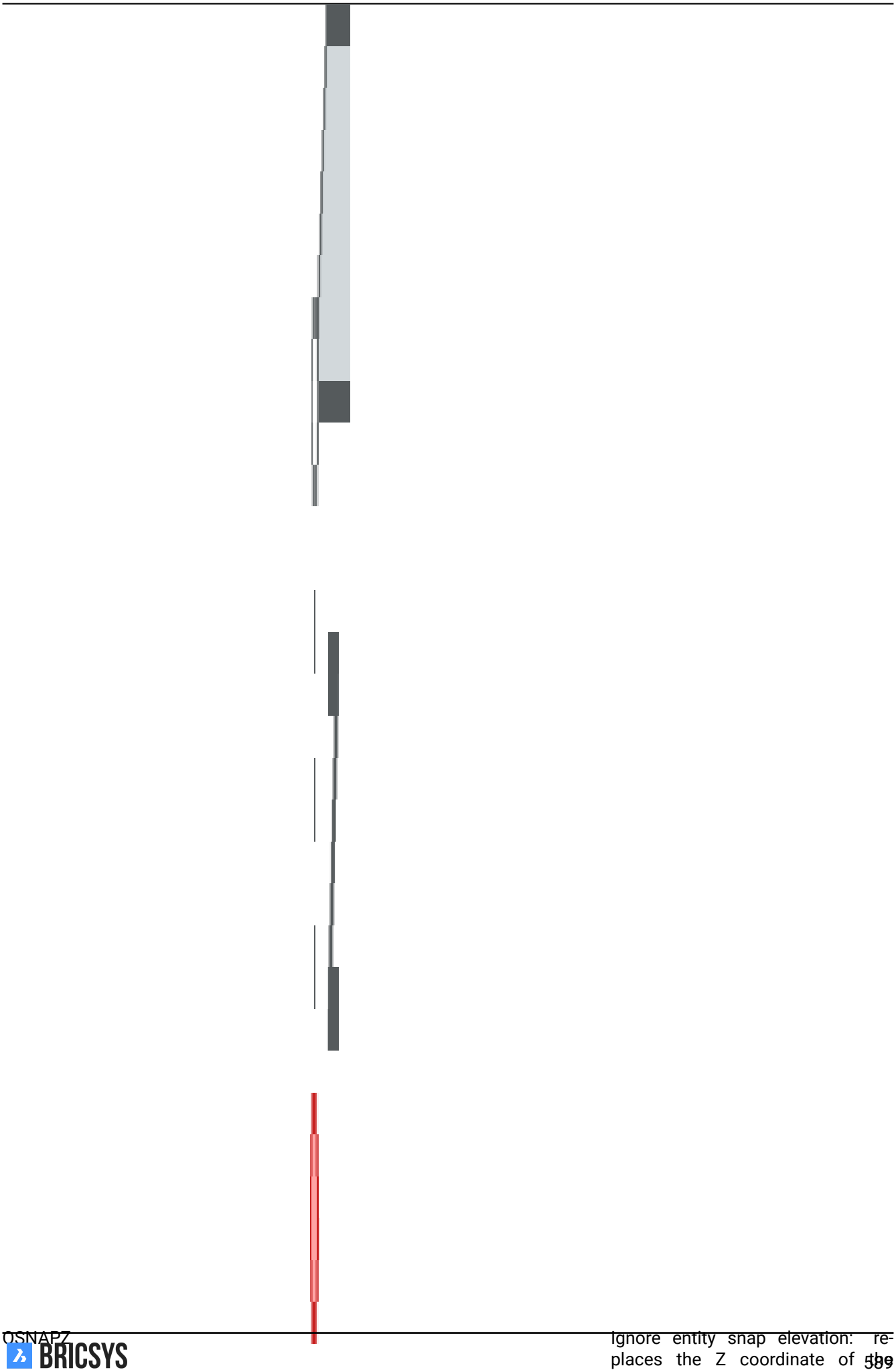
Name	Icon	Marker	Context menu shortcut*	Transparent shortcut**	Description
TK				tk	Use temporary tracking points. Prompts you to identify temporary points; press Enter to accept the last temporary point.
MTP			2	mtp	Snaps to a point in the middle between two points. Prompts you to identify two points.
ENDPOINT			E	end	Snaps to the nearest endpoint of an entity or polyline segment.
MIDPOINT			M	mid	Snaps to the midpoint of an entity or polyline segment.
NEAREST			N	nea	Snaps to the nearest point on an entity.
CENTER			C	cen	Snaps to the center point of an arc, circle, polygon, ellipse or elliptical arc. Snaps to the center of gravity of a closed polyline.
GEOMETRIC CENTER			G	gce	Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids.

PERPENDICULAR			P	per	Snaps to the perpendicular point of another entity. You can snap to an arc, circle, ellipse, line, polyline, infinite line, ray, spline or edge of a plane to form a perpendicular alignment with that entity or with an extension of that entity.
PARALLEL			L	par	Displays a tracking line parallel to the selected entity.
TANGENT			T	tan	Snaps to the point on an arc, ellipse, spline or circle that, when connected to the previous point, forms a line tangent to that entity.
QUADRANT			Q	qua	Snaps to the closest quadrant of an arc, circle, ellipse, or elliptical arc.
INSERTION			I	ins	Snaps to the insertion point of an attribute, block or text entity.
POINT			O	poi nod	Snaps to a point entity.
INTERSECTION			R	int	Snaps to the intersection of any combination of entities.

APPARENT INTERSECTION			A	app	Snaps to the apparent intersection in the current view of two entities that do not intersect in three-dimensional space. Snaps also to the intersection of any combination of entities.
EXTENSION			X	ext	Snaps to the extension of an entity or to the intersection of the extension of two entities.
NONE			Y	non	Turns off all entity snap modes.

Entity Snap Modes * Type this letter to quickly select the entity snap mode in the right click context menu. ** Type the transparent shortcut in the command bar when a command prompts for a point. Other running entity snaps are temporarily disabled.

System Variable	Icon	Description



OSNAPZ

Ignore entity snap elevation: replaces the Z coordinate of the snapped point with the current value

OSOPTIONS		Entity snap to hatches: allows to snap to hatches.
OSOPTIONS		Entity snap to negative-z: allows entity snaps with a negative z-value when a **Dynamic UCS** is active,

NOTE

You can toggle the **Entity Snaps** on/off:

- by clicking the **ESNAP** field in the Status Bar (1.12.2).
- by hitting the **F3** function key.

Working with multiple Entity Snap modes

1. Move the cursor to the entity you want to snap. One of the active Entity Snap modes markers display.
2. Do not move the mouse; press the TAB key. The entity the snap point is on highlights.
3. Do one of the following:
   - Click to accept the snap point.
   - Press the TAB key. The next possible Entity Snap mode marker displays. The entity the snap point is on highlights.
4. (option) Repeat step 3 until the right snap point is found.

NOTES

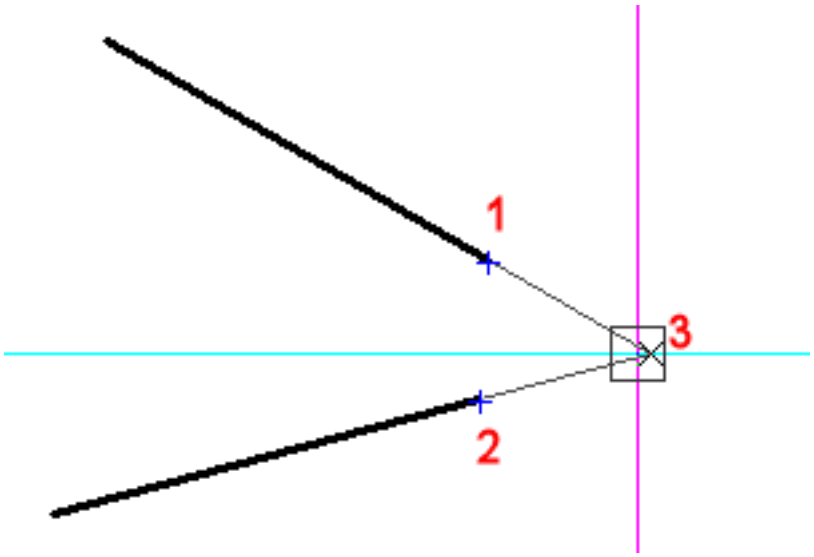
- Do not move the mouse while pressing the TAB key.
- Entity snap cycling is not possible when Dynamic Dimensions (1.15.9) are active. Click the **DYNMODE** field in the **Status Bar (1.12.2)** to toggle Dynamic Dimensions on/off.

To snap to the extension of two entities

1. If not already on, turn on the **Extension** Entity Snap mode ().
2. Launch a drawing tool, e.g. Draw Line.
3. Move the cursor over the endpoint of the first entity (1). A small cross () indicates the entity is marked for extension.
4. Move the cursor over the endpoint of the second entity (2). A small cross () indicates the entity is marked for extension.
5. Move the cursor near the intersection of the extensions of the two entities. An X indicates the intersection (3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/03E
6. Click to accept the snap point.

Note: You can snap to the extension of lines, polylines, arcs and elliptical arcs.

Figure 1.492



Using temporary tracking points

When using temporary tracking points, Polar Tracking (1.15.16) must be on and it is recommended to have dynamic dimensions (1.15.9) active as well.

1. Whenever a command prompts for a point, do one of the following:

   - Click the mouse wheel.
   - Click the **Temporay tracking points** tool button ( ) on the Entity Snaps toolbar.
   - Hold down the Shift key, then right click and choose **Temporay tracking points** in the context menu.
   - Type **TK** in the command bar, then press Enter.

   The command is suspended temporarily. You are prompted: Specify first temporary tracking point:

2. Enter a point, e.g. use an entity snap to specify a point. You are prompted: Specify next temporary tracking point (press ENTER to end)

3. Do one of the following:

   - Enter a point.
   - Press Enter to accept the last temporary tracking.

   The command resumes.

Using 'Middle of 2 points' snap

1. When prompted to specify a point, do one of the following:

   - Click the **Snap to the middle of 2 points** button ( ) on the Entity Snaps toolbar.
   - Type **mtp** or **m2p** in the command bar, then press Enter.
   - Press and hold the Shift key, then right click and choose **Snap to middle of 2 points** in the context menu.

   You are prompted: First point of mid:

2. Specify the first point You are prompted: Second point of mid:

3. Specify the second point. The point in the middle of the two points is snapped.

Using the Parallel entity snap

1. Make sure Entity Snap Tracking (1.15.17) is active.

2. Launch an entity create command to create a linear entity (e.g. Line).

3. Specify the first point of the entity. The line displays dynamically.

4. Do one of the following:

- Press and hold the Shift key, then right click and choose **Snap to parallel** in the context menu.



- Click the **Snap to parallel** tool button ( ) in the Entity Snaps toolbar.

5. Hover the cursor over the entity you want to draw parallel to. The **Parallel Entity Snap** marker ( ) displays on the selected entity. Wait until a small cross (+) indicates the entity is recognized.

6. Move the cursor in order to place the dynamics of the line more or less parallel to the selected entity. A tracking line displays from the start point of the line.

7. Do one of the following to specify the second point while the tracking line displays:
   - Click a point.
   - Enter the desired length and press Enter.
   - Use a second entity snap tracking point, then move the cursor to the intersection of the parallel tracking line and the second entity snap tracking line.

Geometric Relations

When dragging a grip (1.23.4) of a line, polyline, arc or a circle the geometric relation (perpendicular or tangent) of the entity being edited with crossing or nearby other lines, polylines, arcs or circles is indicated. A snap marker displays and the other entity highlights when the current position of the dragged entity is perpendicular or tangent within the distance defined by the APERTURE (1.13.11) system variable. This feature is controlled by the GEOMRELATIONS (1.13.11) system variable. In the images below the red grip of the line is dragged. Depending on the position of the dragged grip the line is nearly perpendicular to the other line (top) or the circle (middle) or tangent to the circle (bottom). Click when the desired position is reached. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/GeomRelations2.png})
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/DrawingAccurately/GeomRelations3.png})

Figure 1.493

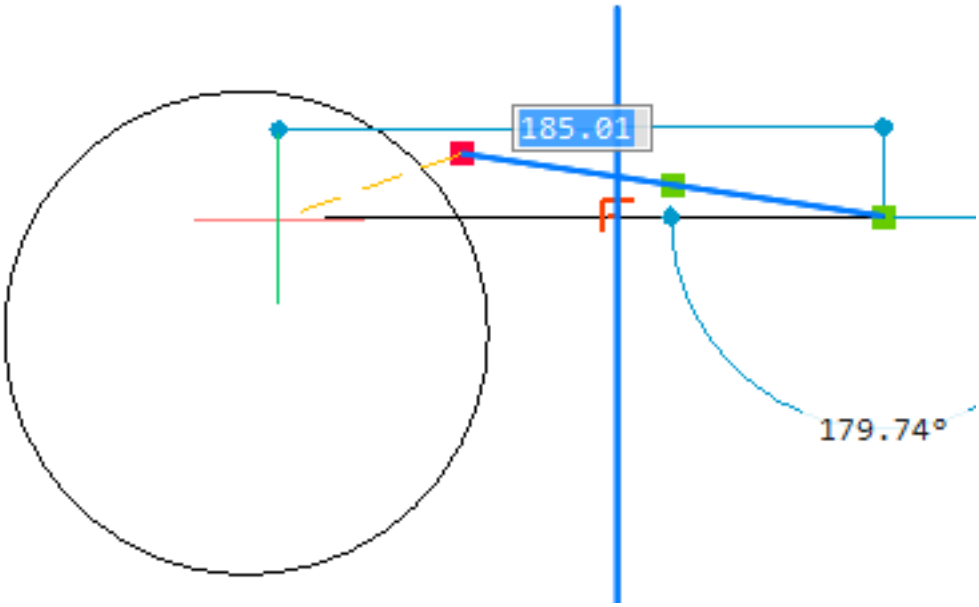


Figure 1.494

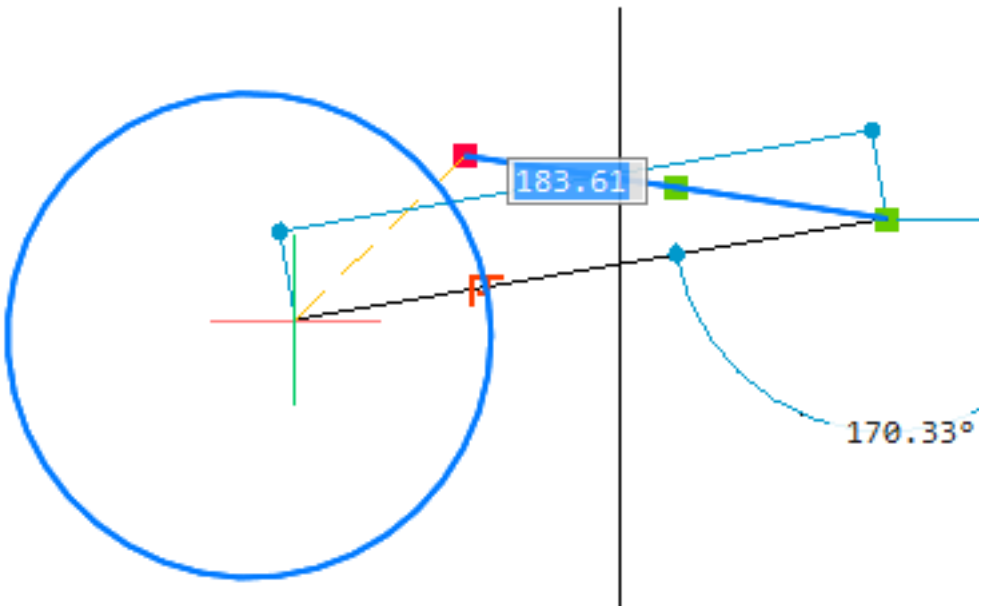


Figure 1.495

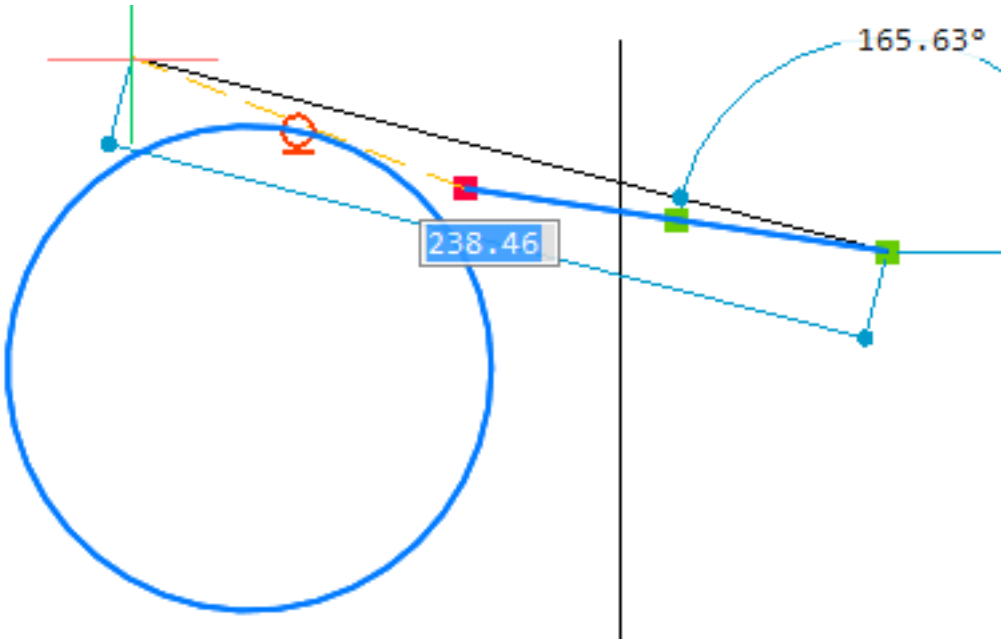
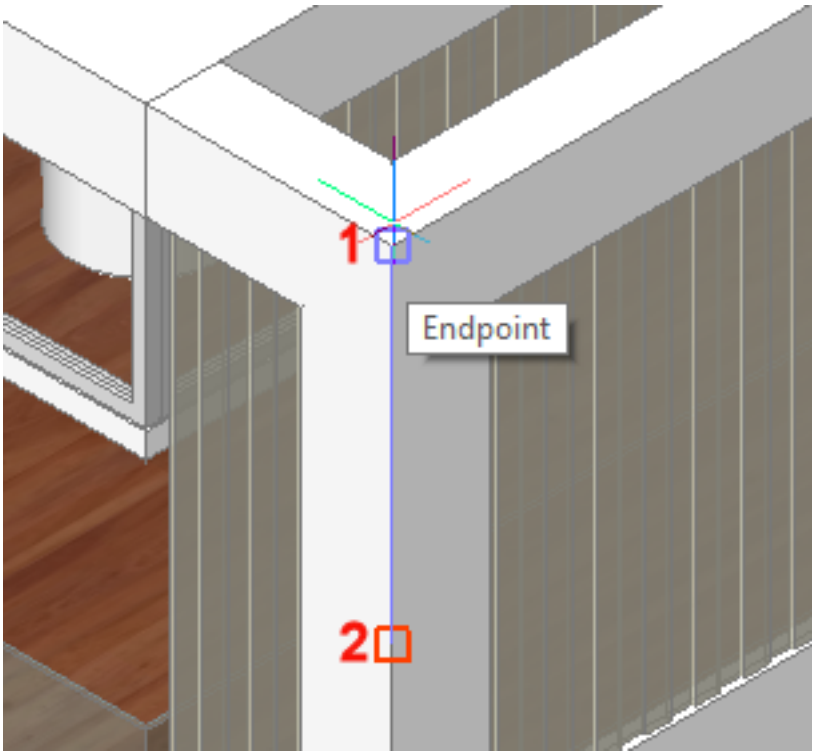


Figure 1.496



1.15.16 Polar Tracking

Tutorial: Drawing Accurately : Polar and Snap Tracking Click here (1.4.2) to watch Polar tracking assists you to draw at exact angles:

- at fixed intervals, starting from 3 O'clock (or East)
- at specific additional angles

A polar tracking path line displays from the origin point when you move the cursor close to one of the polar tracking angles. The tracking path lines display as long as the **Snap Aperture** box overlaps the polar tracking line. In 3D views an additional tracking line displays parallel to the Z-axis of the current UCS. Snap track lock: to lock a snap track path hit the Shift key. To unlock a track path, hit Shift again: it works as a toggle. Locked tracking paths display in the **color defined by the**SnapMarkerColor (1.13.11) user preference..

NOTE	The display of the tracking path line is controlled through the **TRACKPATH** system variable. The color of the tracking path line is controlled through the **AutoTrackingVecColor** user preference. Tracking paths parallel to the X-, Y- or Z-axis of the current coordinate system (WCS or UCS) display in the color of the axis they are parallel with (defined by **COLORX** , **COLORY** or **COLORZ** variables)..

To define the Polar Tracking settings

1. Do one of the following:
   - Right click on the **POLAR** field in the Status Bar (1.12.2), then right click and choose **Settings** in the context menu.
   - Open the **Settings** dialog, then expand the **Coordinate input** sub-category under **Drafting** in the **Drawing** settings class. Expand the **Snap Tracking** settings group. **(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/**

2. Adjust the settings:
   - 1: Autosnap marker: If checked displays markers at the snapped locations.
   - 2: Autosnap tooltips: If checked tooltips display.
   - 4: Autosnap magnet (not used, required for AutoCad® compatibility only)
   - 8: Polar tracking: If checked, **Polar Tracking** is on.
   - 16: Entity snap tracking: If checked, **Object Snap Tracking** is on.
   - 32: Tooltips: If checked, tooltips display for polar tracking and entity snap tracking.

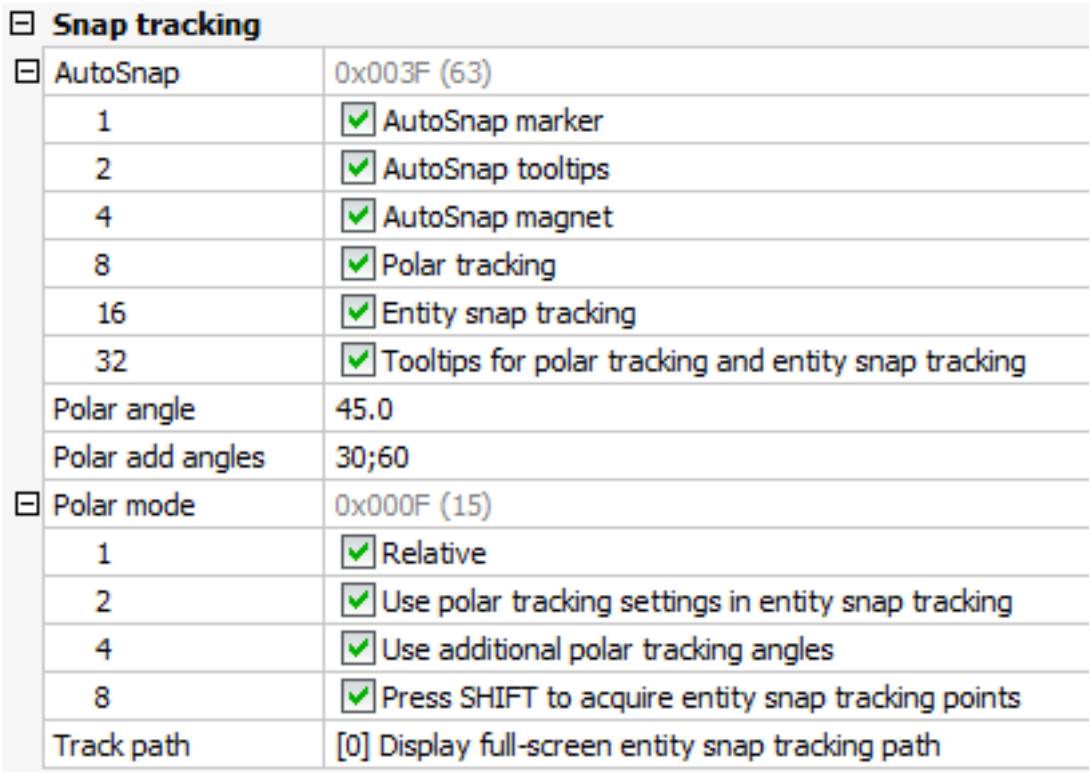
3. Polar angle: Sets the polar angle increment.

4. Polar add angles: Sets the additional polar angles. Separate angles by semicolons.

5. Polar mode: Controles the behavior of Polar Tracking.
   - 1: If checked, polar angles are measured relatively from selected entities.
   - 2: If checked, the polar angles (interval and additional angles) are also used in entity snap tracking.
   - 4: If checked, the additional angles for polar tracking are used.
   - 8: If checked, press and hold the Shift key to acquire entity tracking points.

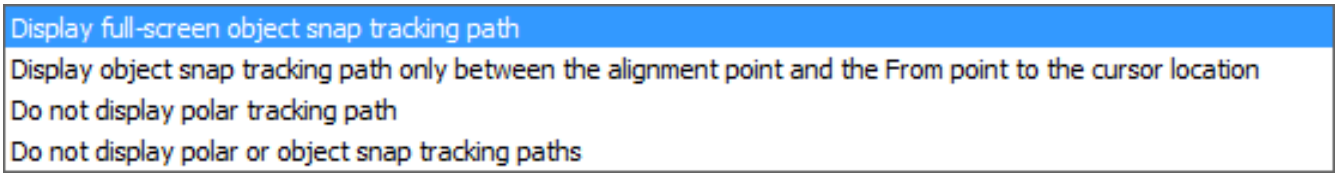
6. Track path: Controls the display of the polar and entity snap tracking paths. (\vref {fig:C:/Dev/Task-2448/images/staticbr

Figure 1.497



Snap tracking	
AutoSnap	0x003F (63)
1	AutoSnap marker
2	AutoSnap tooltips
4	AutoSnap magnet
8	Polar tracking
16	Entity snap tracking
32	Tooltips for polar tracking and entity snap tracking
Polar angle	45.0
Polar add angles	30;60
Polar mode	0x000F (15)
1	Relative
2	Use polar tracking settings in entity snap tracking
4	Use additional polar tracking angles
8	Press SHIFT to acquire entity snap tracking points
Track path	[0] Display full-screen entity snap tracking path

Figure 1.498



Using Polar Tracking to draw a line

1. (option) Check the **Polar Tracking** settings.
2. (option) If not already on, click the POLAR field in the Status Bar (1.12.2).
3. Launch the Draw Line (6.20.31) tool.
4. Specify the start point of the line. Polar tracking lines display at the specified intervals and additional angles. If the cursor is on or next to a tracking line:
   - The current distance and angle from the start point display next to the cursor when
   - A cross marker (X) indicates the current point.
5. When the appropriate tracking line displays, do one of the following to define the endpoint of the line:
   - Click when the cursor is at the desired position.
   - Type the length of the line in the command bar and press Enter.

Note: Turning on **Polar Tracking** automatically disables Orthogonal Mode (1.15.2).

Using Snap Track Lock

Hit the Shift key while a snap track marker line is displayed to lock a tracking path, any (snap)point that is picked will be projected perpendicularly onto the tracking path. Hit the Shift key again to unlock. A locked tracking path changes to the color defined by theSnapMarkerColor (1.13.11) user preference.

Using temporary tracking points

When **Polar Tracking** is ON, temporary tracking points can be used to pick points at locations where no existing geometry is available to snap directly to, while you know the desired offset distance(s) from snap points on existing geometry in the neighborhood. **TT option: whenever prompted to enter a point, type TT** on the command line, then press Enter to define a single temporary tracking point. After the tracking point has been picked, normal command execution continues. Repeat typing **TT** to enter additional temporary tracking points. **TK option** : whenever prompted to enter a point, type **TK** on the command line, then press Enter to define one or more temporary tracking points. When eventually the desired location has been reached, enter the point by pressing the Right Mouse button or the Enter key.

Using the TT option

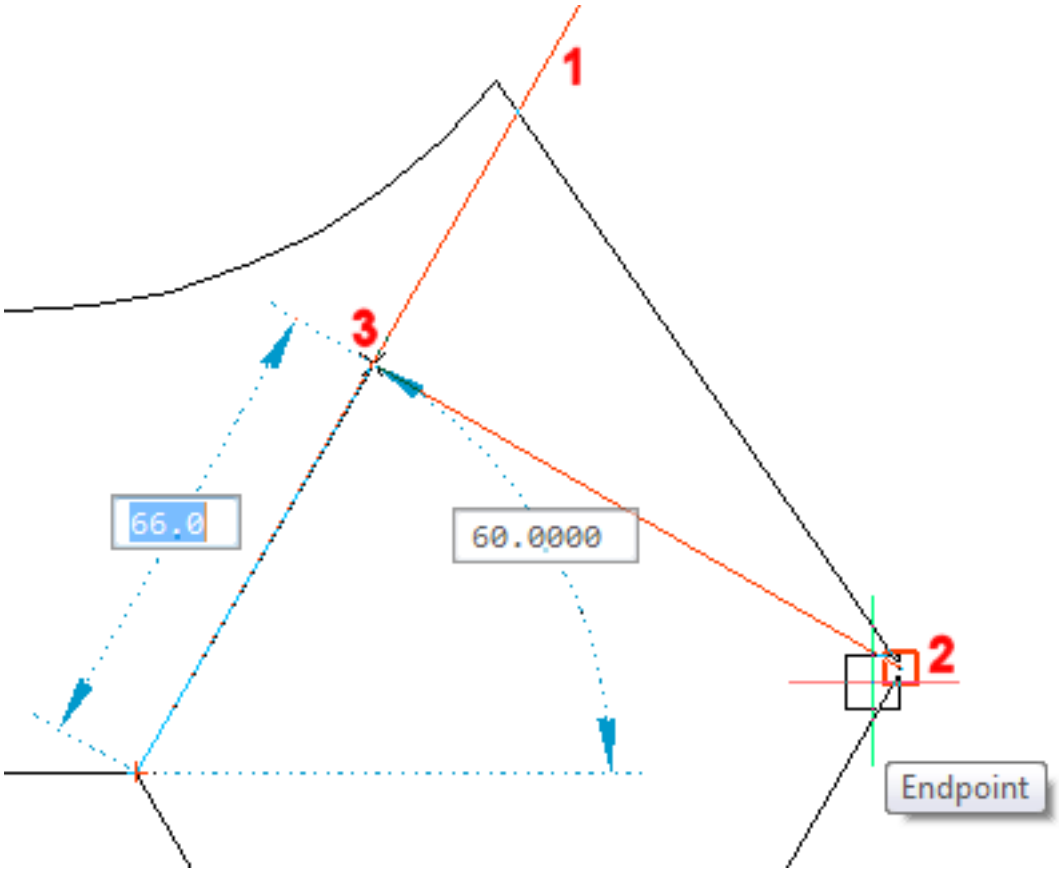
1. Launch a draw command, e.g. Line. You are prompted: ENTER to use last point/Follow/<Start of line>:
2. Type **TT** in the command bar, then right click or press Enter. You are prompted: Specify temporary tracking point:
3. Use anEntity Snap (1.15.15) to pick a point on existing geometry. A small red cross indicates the temporary tracking point.
4. Move the cursor in the desired direction. Depending on the value of thePOLARANG (1.13.11) and POLARADDANG (1.13.11) system variables tracking lines display from the first temporary tracking point.
5. When a tracking line in the appropriate direction displays, type the distance from the temporary tracking point.
6. Specify the endpoint of the line.

Using the TK option

1. Launch a draw command, e.g. Line. You are prompted: ENTER to use last point/Follow/<Start of line>:
2. Type **TK** in the command bar, then right click or press Enter. You are prompted: Specify first temporary tracking point:
3. Use anEntity Snap (1.15.15) to pick a point on existing geometry. A small red cross indicates the temporary tracking point. You are prompted: Specify next temporary tracking point (press ENTER to end):
4. Move the cursor in the desired direction. Depending on the value of thePOLARANG (1.13.11) and POLARADDANG (1.13.11) system variables tracking lines display from the first temporary tracking point.

5. When a tracking line in the appropriate direction displays, type the distance from the temporary tracking point. A small red cross indicates the second temporary tracking point. You are prompted: Specify next temporary tracking point (press ENTER to end):
6. Do one of the following:
7. Right click or press Enter to accept the second tracking point as the start point of the line.
8. Repeat steps 4 and 5 until the desired point is specified, then right click or press Enter to accept the point.
9. Specify the endpoint of the line.

Figure 1.499



1.15.17 Entity Snap Tracking

Entity Snap Tracking assists you to draw entities with respect to other entities. When **Entity Snap Tracking** is turned on, BricsCAD displays temporary alignments based on entity snap points. **Entity Snap Tracking** works in combination with **Entity Snap** tools. At least one **Entity Snap** mode must be on if you want to use **Entity Snap Tracking.** Click the **STRACK** field in the Status bar (1.12.2) to enable/disable **Entity Snap Tracking** . **Entity Snap Tracking** alignments are parallel to the X- and Y-axis of the current UCS by default. If the Polar mode > (1.15.16) 0x0002 setting is checked, the polar angles used in Polar Tracking (1.15.16) (interval and additional angles) are also used in **Entity Snap Tracking** . Alignment points are acquired when you hover over a point, using entity snaps. Acquired points display a small plus sign (+). To remove an acquired point, hover over the point again. The small plus sign (+) is then removed. If the Polar mode > (1.15.16) 0x0008 is checked, you must press and hold the Shift key to acquire entity tracking points. Alignments relative to an acquired point are displayed as you move the cursor over their tracking paths. You can then define points either on the alignments or at the intersection of two alignments. Esnap track markers: to distinguish between regular esnap markers and markers drawn together with related snap tracking lines, esnap track markers display in the color of the tracking lines. Hitting the Shift key when a tracking line displays, locks the tracking line. Locked tracking lines display in red (1). Entity snaps (2) or specified points are projected perpendicularly onto the tracking line (3). Hit the Shift key again to unlock a tracking line. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/06SnapTrackLocked.png}) More options with locked tracking lines are:

- If the Intersection entity snap is active, it is possible to snap to the intersections of a locked tracking line and entities in the drawing.
- Acquire additional tracking lines and snap to the locations where these intersect with the locked tracking line.

Using Entity Snap Tracking

1. Launch a drawing tool, e.g. **Draw Line** .
2. Hover the cursor over the first tracking point (1). A small plus sign (+) displays to indicate the tracking point is active.
3. Hover the cursor over the second tracking point (2) A small plus sign (+) displays to indicate the tracking point is active.

4. Move the cursor close to position 4.

   - Tracking lines display from both tracking points (3).
   - A cross marker (X) indicates the intersection of the tracking lines.
   - Snap markers display at the tracking points.
   - An **Entity Snap Tracking** tooltip at the intersection of the tracking lines shows the angles and distances from the tracking points.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/06S

5. Click to accept the **Entity Snap Tracking** point.

Tutorial: Drawing Accurately: Snap and Grid Click here (1.4.1) to watch

Figure 1.500

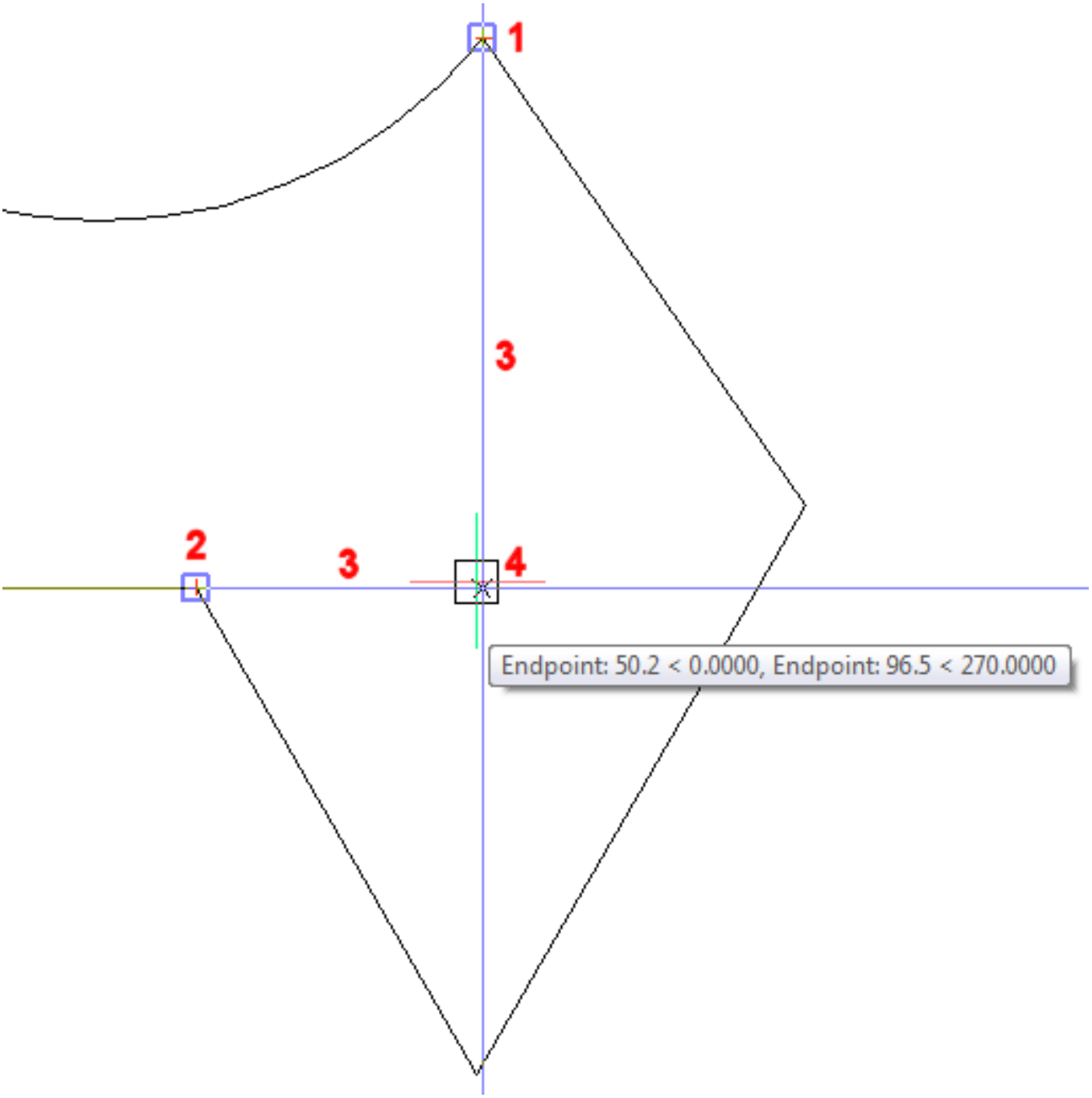
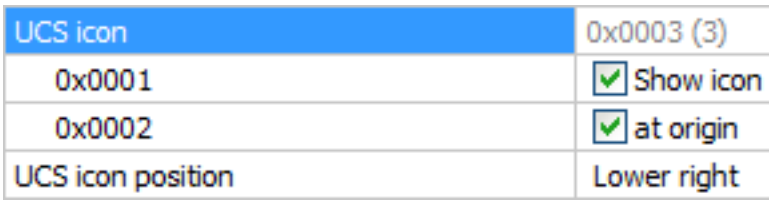


Figure 1.501



1.15.18 User Coordinate Systems

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. The **ELEVATION** system variable specifies the distance from the xy-plane of the current coordinate system. If **ELEVATION** = 0 (zero), entities are created in the xy-plane of the current coordinate system. Every drawing has a fixed coordinate system called the World Coordinate System (WCS). You can also define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS) and can be located anywhere in the WCS and oriented in any direction. You can create as many UCS as you want, saving or redefining them to help you construct three-dimensional entities. By defining a UCS within the WCS, you can simplify the creation of most three-dimensional entities into combinations of two-dimensional entities. When you create a new drawing the WCS is the current coordinate system by default, which is indicated by the letter W in the UCS icon. When you display a drawing in plan view, you see the coordinate system icon from the top, with the z-axis directed straight toward you. When you display a three-dimensional drawing in a view other than plan view, the coordinate system icon changes to reflect your new viewpoint. **Note:** You cannot delete or modify the WCS. To help you keep your bearings in the current coordinate system, BricsCAD displays a coordinate system icon (UCS icon). The visible portions of the axes are the positive directions. **2D Wireframe visual style** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/00UCSiconWorldTop.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/00UCSico (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/00UCSico (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/00UCSico WCS Top View UCS Top View WCS Isometric View UCS Isometric View **Rendered Visual Styles** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/00UCSiconRenderTop.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/00UCSico (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/00UCSico (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/00UCSico Three colors represent the three axes, making it easier for you to recognize the orientation in three-dimensional space:

- x-axis: red
- y-axis: green
- z-axis: blue

The colors of the axes are controlled by the COLORX (1.13.11), COLORY (1.13.11) and COLORZ (1.13.11) system variables. The Plan View (6.24.25) command restores the Plan view or Top view of the current UCS or WCS. The **UCSICON** settings variable controls the display and location of the UCS icon: (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco **Show icon** : Controls whether the icon shows or not. **At origin** : Controls the location of the UCS icon: if on, the icon indicates the origin point of the current coordinate system (UCS or WCS). However, if the origin is not within the viewport borders, the UCS icon moves to the corner of the viewport, as defined by the **UCSICONPOS** (UCS Icon Position) settings variable. When the **at origin** option is not checked, the icon always displays in the corner of the viewport defined by **UCSICONPOS**.

To move the origin of the current coordinate system

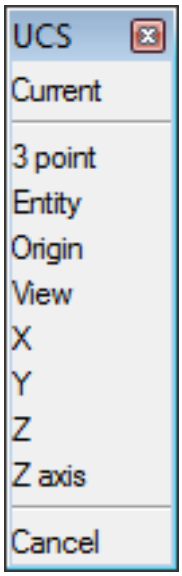
1. Type **ucs** at the command prompt. You are prompted: Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z Axis/Move/World] <World>:

2. Identify the new origin, then press Enter to confirm.

To define a User Coordinate System

1. Choose **Drawing Explorer > Coordinate Systems** in the **Tools** menu. The **Drawing Explorer - Coordinate Systems** dialog opens.

Figure 1.502



2. Click the **New** tool button ( ) on the **Drawing Explorer**dialog. The **BricsCAD Explore**dialog closes. **You are prompted**: Current/Entity/Origin/View/X/Y/Z/ZAxis;3point;:; A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/07UCSPromptBox.png})

3. Press Enter to define the UCS using points. You are prompted: New origin <current origin>:

4. Specify the origin point of the UCS. You are prompted: Point of positive X axis <current point>:

5. Specify a point to define the positive X-axis. The command window reads: Point in X-Y plane with positive Y value <current point>:

6. Specify a point to define the positive Y-axis. The UCS is defined. The **BricsCAD Explorer** dialog reopens.

7. Click in the **UCS Name** field of the newly defined UCS to replace the <NewUCS> default name.

8. Close the **BricsCAD Explorer** dialog.

Command Options

See the EXPUCS (6.13.18) and UCS (6.30.2) commands.

To restore the WCS

1. Type **UCS** in the command bar, then press Enter. You are prompted: ?/3point/Delete/Entity/Origin/Previous/Restore/Save
   A prompt menu displays.
2. Do one of the following:
3. Press Enter to accept the default option.
4. Choose **World** in the prompt menu.
5. Type **W** in the command bar and press Enter.

To restore a UCS

1. Choose **Drawing Explorer > Coordinate Systems** in the **Tools** menu. The **Drawing Explorer - Coordinate Systems** dialog opens.
2. Click the blank tile in front of the UCS name to make it current.
3. Close the **BricsCAD Explorer** dialog.

Note: If no UCS is marked, the WCS is the current coordinate system.

To set a relative UCS

1. Do one of the following:

   - Click the **UCS...** tool button ( ) on the **Settings** toolbar.
   - Choose **UCS...** in the **Settings** menu.

   The **User Coordinate System** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Usr

2. Choose one of the following:

   - Set the selected UCS relative to the **Current UCS** .
   - Set the selected UCS relative to the **World coordinate system (WCS)** .

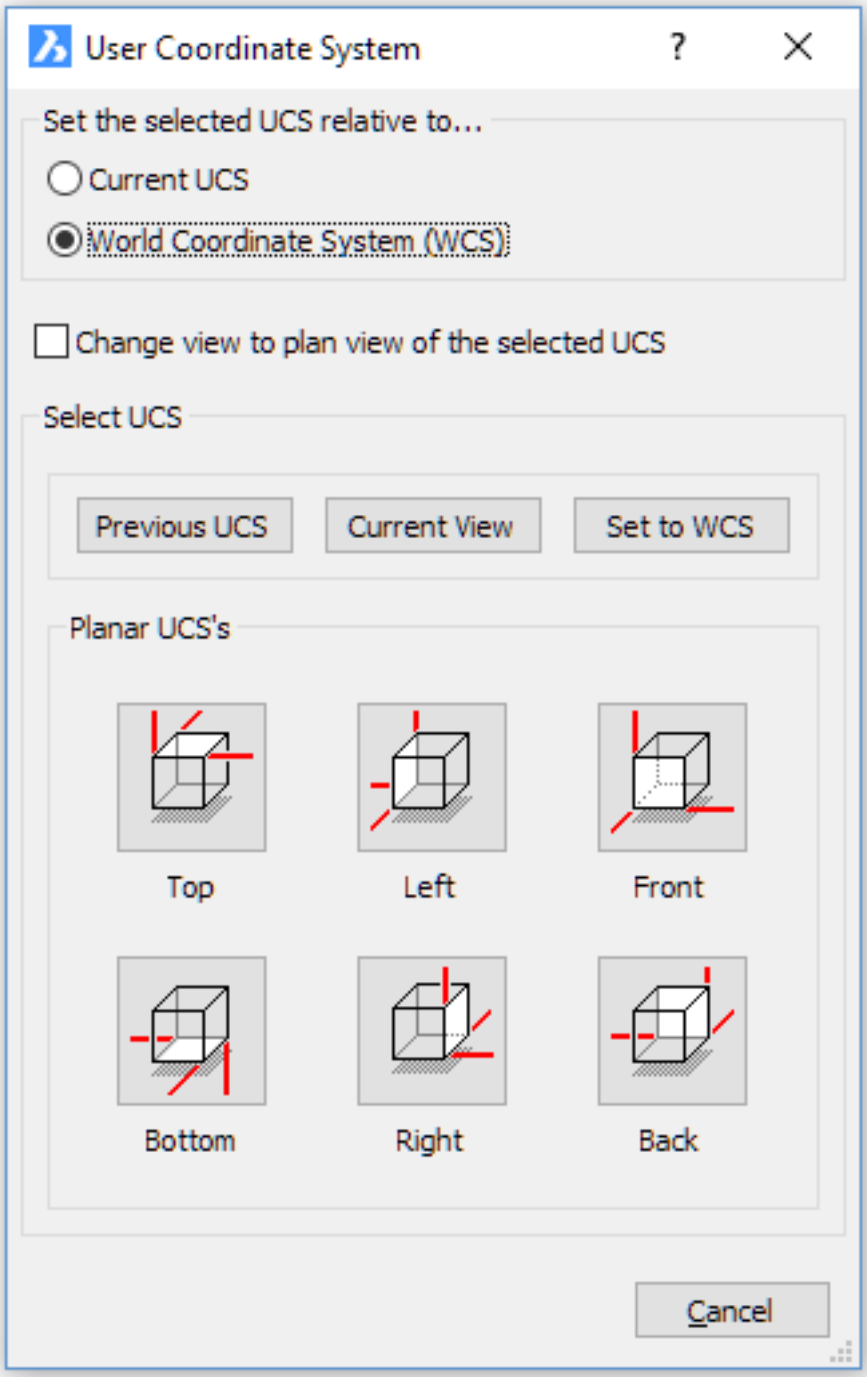
3. (option) Enable the **Change view to plan view of the selected UCS** option.

4. Click one of the **Planar UCS's** buttons. The User Coordinate System dialog closes.

NOTES

- If the UCSORTHO (1.13.11) system variable is ON an orthographic UCS is restored automatically when the related orthographic view is restored by the -VIEW (6.31.2) command or selected on the Look From (1.16.8) control. An orthographic UCS has its origin at the WCS origin.
- The UCSVP (1.13.11) (UCS viewports) system variable controls whether the UCS in a viewport is fixed (UCSVP = ON) or changes to match the UCS in the current viewport (UCSVP = Off). As a result, in a multiple viewport drawing, the UCS of all viewports that have UCSVP = Off matches the orthographic UCS of the current viewport if UCSORTHO = On.

Figure 1.503


User Coordinate System
?
Set the selected UCS relative to...
Current UCS
World Coordinate System (WCS)
Change view to plan view of the selected UCS
Select UCS
Previous UCS
Current View
Set to WCS
Planar UCS's
Top
Left
Front
Bottom
Right
Back
Cancel

Command Options

See the SETUCS (6.27.25) command.

Dynamic UCS (DUCS)

With the Dynamic UCS feature enabled, the coordinate system is aligned automatically with a face of a 3D solid under the cursor when drawing entities. This feature is controlled by the UCSDETECT (1.13.11) system variable. The RedHilite_DUCSLocked_Face_Color (1.13.11) user preference controls the highlight color for the face to which the Dynamic UCS has been locked by pressing the Shift key. The RedHilite_DUCSLocked_Face_Alpha (1.13.11) user preference controls the highlight transparency for the face to which the Dynamic UCS has been locked by pressing the Shift key.

To toggle the dynamic UCS Do one of the following:

- Press the **F6** function key.
- Click the **DUCS** field in the status bar (1.12.2).

Using the Dynamic UCS

1. With the Dynamic UCS feature active, launch a Draw command.
2. Move the cursor over the face of a 3D solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG

   - The face highlights.
   - If ON, the grid is aligned with the selected face.
   - If the UCSICON (1.13.11) system variable = 3 (On at Origin), the UCS icon displays in the selected face.
   - The dynamic UCS is oriented according to the edge by which the cursor 'entered' the hovered face, thus providing users a way to manipulate the orientation of the DUCS. The origin of the DUCS is set to the start point of this edge and the X-axis is chosen codirectional to the tangent at the start point of the edge.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGuiTsc/UCSDrawingAccurately/Bottom	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGuiTsc/UCSDrawingAccurately/Left	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGuiTsc/UCSDrawingAccurately/Curved
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGuiTsc/UCSDrawingAccurately/Top	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGuiTsc/UCSDrawingAccurately/Right	

3. (option) Hit the Shift key to lock the DUCS, which allows you to start drawing outside the highlighted face. Hit the Shift key again to unlock the DUCS.
4. Pick a point to start drawing.
5. When the drawing command concludes, the WCS or previously defined UCS is restored.

Tutorial: DUCS on Curved Surfaces Click here (1.6.27) to watch

Figure 1.504

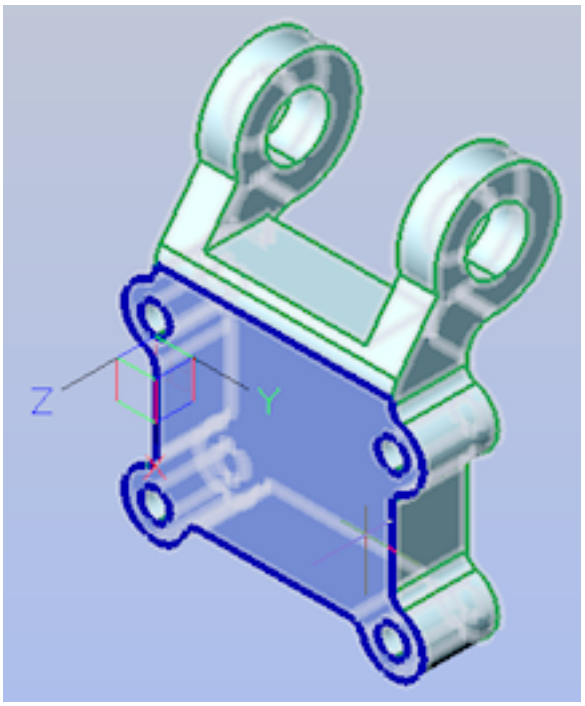


Figure 1.505

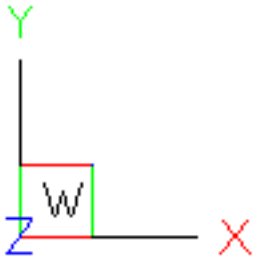


Figure 1.506

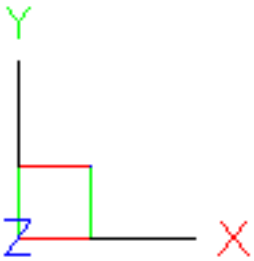


Figure 1.507

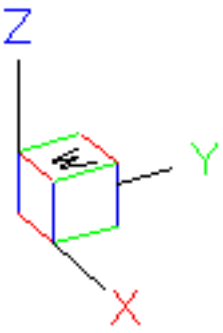


Figure 1.508

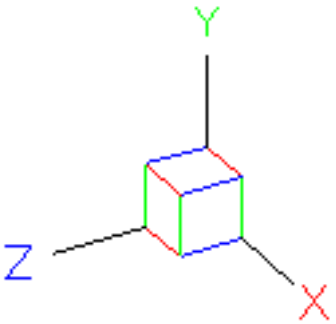


Figure 1.509

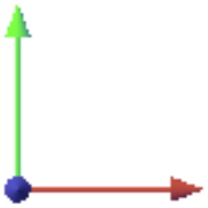


Figure 1.510

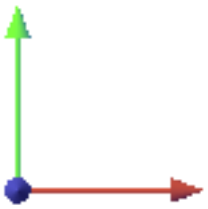


Figure 1.511

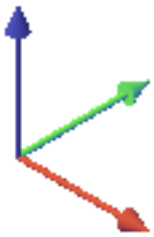


Figure 1.512

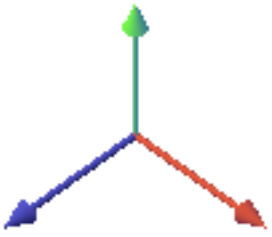


Figure 1.513

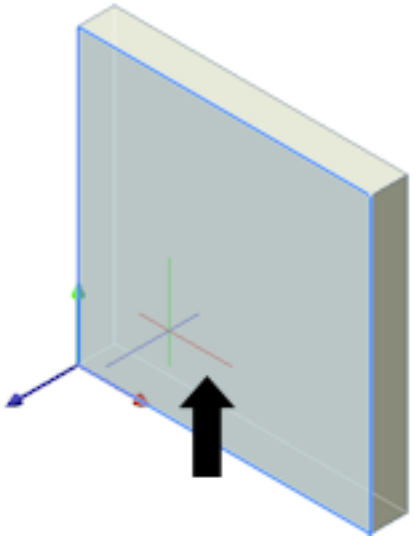


Figure 1.514

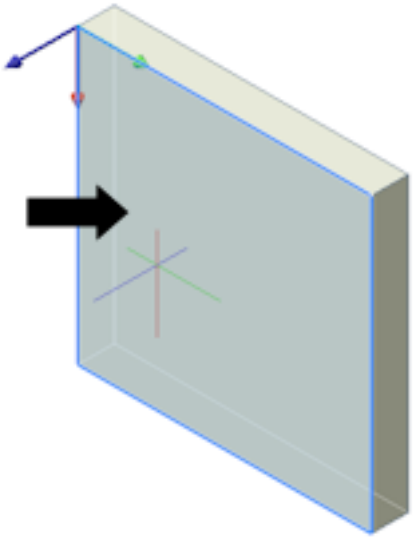


Figure 1.515

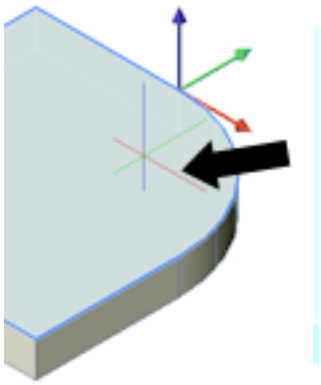


Figure 1.516

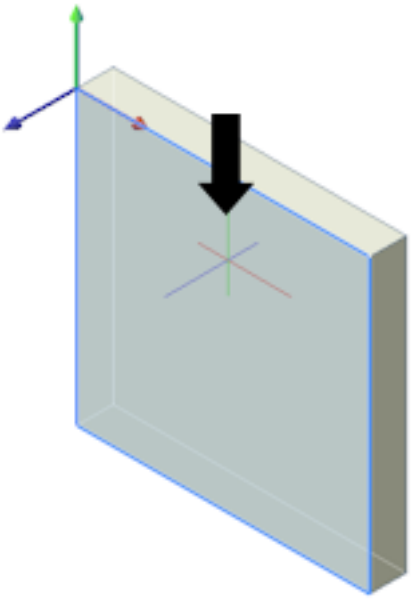
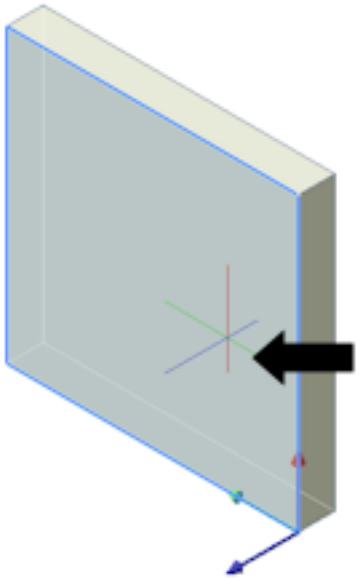


Figure 1.517



1.15.19 Measuring

Commands: DIST (6.10.31), AREA (6.4.13), MASSPROP (6.21.4), ID (6.17.1) and LIST (6.20.35) The **Dist** command reports the distance and angle between two points. The **Area** command finds the area and perimeter (or length) of 2D entities. The **Massprop** command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties"). The **ID** command reports the x,y,z coordinates of picked points. The **List** command lists the properties of selected entities.

Measuring distances

To measure a distance between two points:

1. Do one of the following:

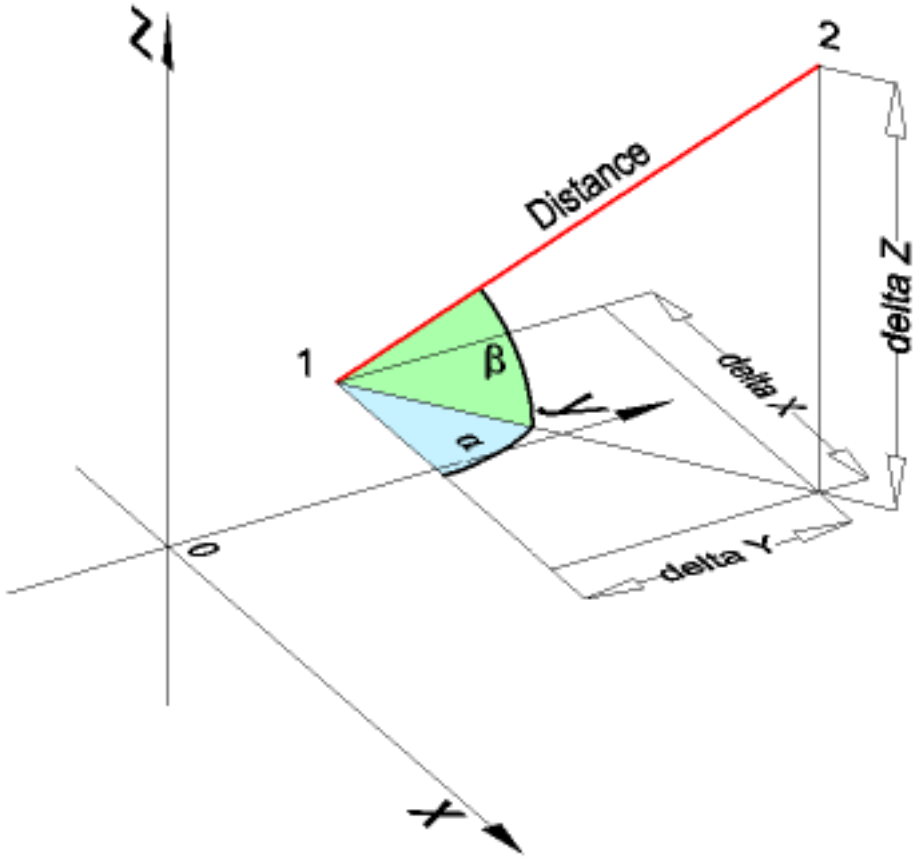
   - Click the **Distance** tool button ( ) on the Inquiry toolbar.
   - Type **dist** in the command bar and press Enter.

   You are prompted: Starting point for distance.

2. Identify the first point. You are prompted: End point.

3. Identify the second point. If DRAGSNAP (1.15.15) = 1, the distance displays in the dynamic dimension (1.15.9) field. The following information displays in the command bar (see image below): Distance, Angle in XY Plane (alpha), Angle from XY Plane (beta), Delta X, Delta Y, Delta Z. The values are calculated with respect to the current **coordinate system**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03Draw

Figure 1.518



Measuring lengths

To measure the length of single linear entity, click the entity then read the **Length** field in the **Properties Bar**. To add the lengths of a series of linear entities:

1. Do one of the following:

   - Click the **Area** tool button ( ) on the Inquiry toolbar.
   - Type area in the command bar and press Enter.

   You are prompted: Entity/Add/Subtract/<First point>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/sta

2. Do one of the following:

   - Choose **Add areas together** in the prompt menu.
   - Type **A** in the command bar and press Enter.

   You are prompted: Adding: Entity/Subtract/<First point>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/10AreaPromptBox2.png})

Figure 1.519

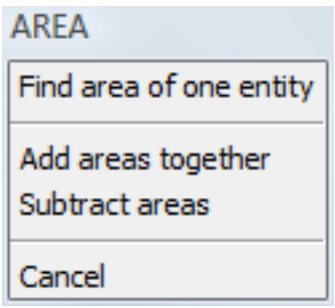
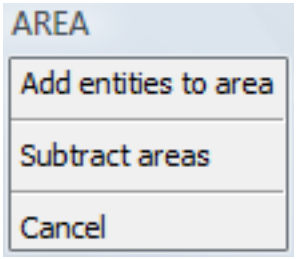


Figure 1.520



3. Do one of the following:

   - Choose **Add entities to area** in the prompt menu.
   - Type **E** in the command bar and press Enter.

   You are prompted: Adding area<Select entities>:

4. Click the first entity. The area and length of the first entity display in the command bar. You are prompted: Adding area<Select entities>:

5. Click the second entity. The area and length of the second entity and the total area and total length display in the command bar. Optionally press the **F2** function key to display the Prompt History window; press **F2** again to close the Prompt History window. You are prompted: Adding area<Select entities>:

6. Repeat step 5 to add more entities or press the **Esc** key to stop.

Measuring areas

To measure the area and length of single closed entity, click the entity then read the **Area** and **Length** fields in the **Properties Bar**. To measure areas and perimeters by specifying points in a drawing:

1. Do one of the following:

   - Click the **Area** tool button ( ) on the Inquiry toolbar.
   - Type area in the command bar and press Enter.

   You are prompted: Entity/Add/Subtract/<First point>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/sta

2. Do one of the following:

   - Choose **Add areas together** in the prompt menu.
   - Type **A** in the command bar and press Enter.

   You are prompted: Adding: Entity/Subtract/<First point>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/10AreaPromptBox2.png})

3. Click the first corner point of the area. A dashed line displays dynamically from the first corner point. You are prompted: Adding: <Next point>:

Figure 1.521

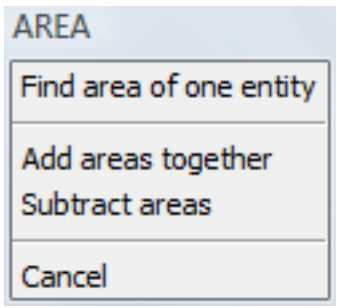
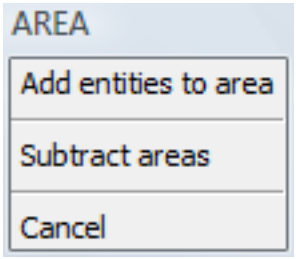


Figure 1.522



4. Click the second corner point of the area. A dashed polygon displays dynamically, indicating the currently defined area. You are prompted: Adding: <Next point>:
5. Click the third corner point of the area. You are prompted: Adding: <Next point>:
6. Repeat step 5 to add more corner points or Right click to stop adding corner points. The **Total length** and **Total area** display in the command bar. Optionally press the **F2** function key to display the Prompt History window; press **F2** again to close the Prompt History window.
7. Do one of the following:
   - Repeat steps 3 through 6 to add another area.
   - Press the **Esc** key to stop.

1.15.20 Coordinate Input

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. Every drawing has a fixed coordinate system called the **World Coordinate System** (WCS). You can also define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (1.15.18) (UCS) and can be located anywhere in the WCS and oriented in any direction. To specify points and distances using the keyboard you can use the following formats:

- Cartesian coordinates: x,y,z
- Cylindrical coordinates: R<alpha,z
- Spherical coordinates: R<alpha<beta

Relative coordinates

If you place the @-character in front of the entry, the coordinates are calculated with respect to the previous point. This technique is called**Relative Coordinates** . When Dynamic Dimensions (1.15.9) are active and you type a value in the **Length** field and add a comma (,) the content of the **Length** field is copied to the command line and the @-character is placed in front automatically, which allows you to specify the next point using relative coordinates with respect to the previous point.

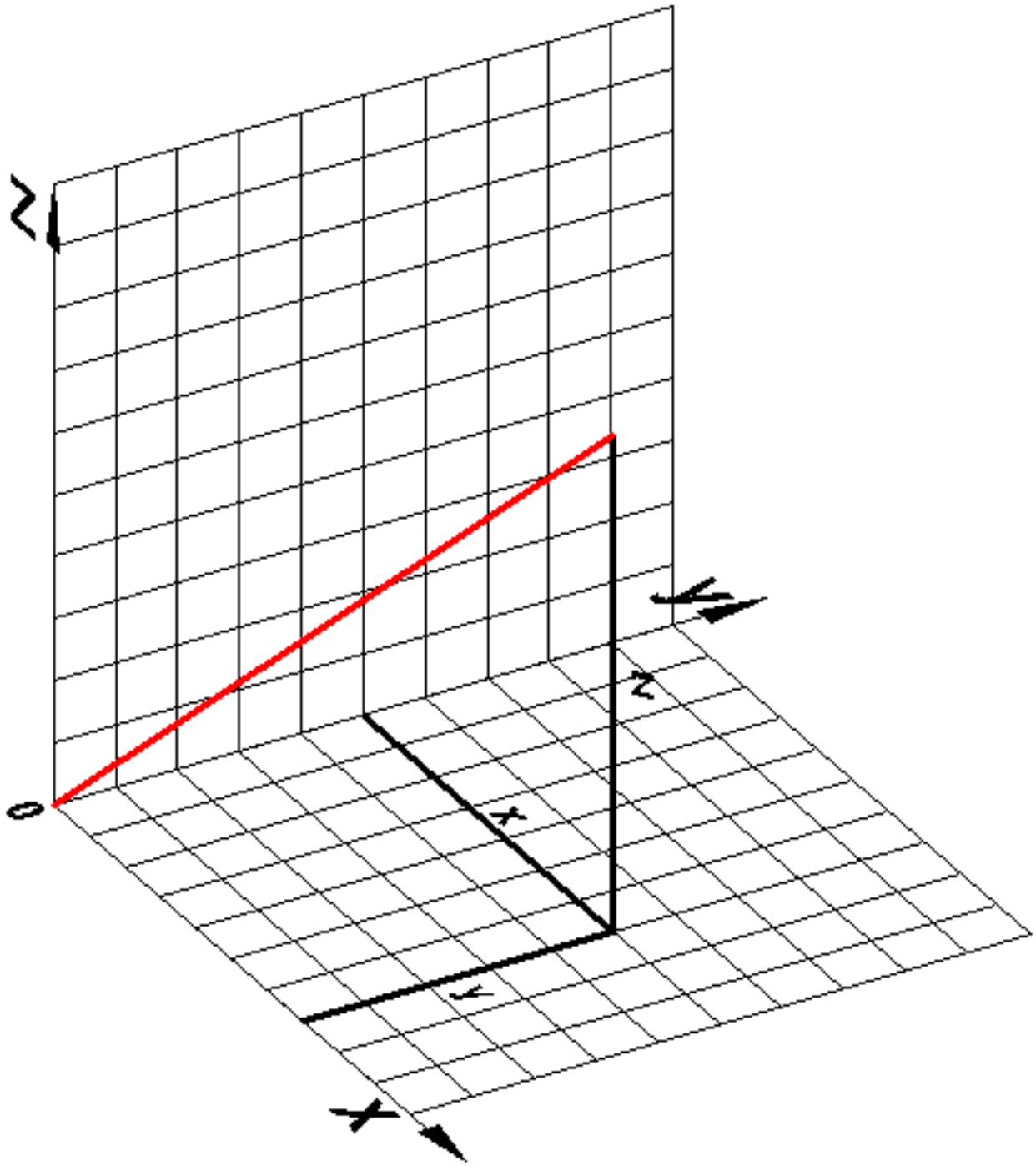
Absolute coordinates

When a user defined coordinate system is active you can enter absolute coordinates (World coordinates) if you place an asterisk (*) in front. E.g. *0,0 refers to the origin of the WCS (World Coordinate System).

Working with Cartesian coordinates

In the Cartesian coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes. A point is defined by its distances to the yz-, xz- and xy- planes. These distances are called the xyz-coordinates of a point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/01CoordinatesCartesian.png}) If you want to enter the absolute Cartesian coordinates of a point, type the x-, y- and z- coordinates separated by commas: 45.5,57.3,60 If you omit the z-coordinate, the point is placed in the xy-plane (Z = 0). If you place the @-character in front of the entry (@x,y), the coordinates are calculated with respect to the previous point. This technique is called **Relative Cartesian coordinates** .

Figure 1.523



Using relative Cartesian coordinates to draw a rectangle

1. Launch the RECTANGLE (1.17.19) command.
2. Specify the first corner of the rectangle.
3. In the command bar type: @<width>,<height>
4. <width> = the width of the rectangle in drawing units, measured along the x-axis
5. <height> = the height of the rectangle in drawing units, measured along the y-axis

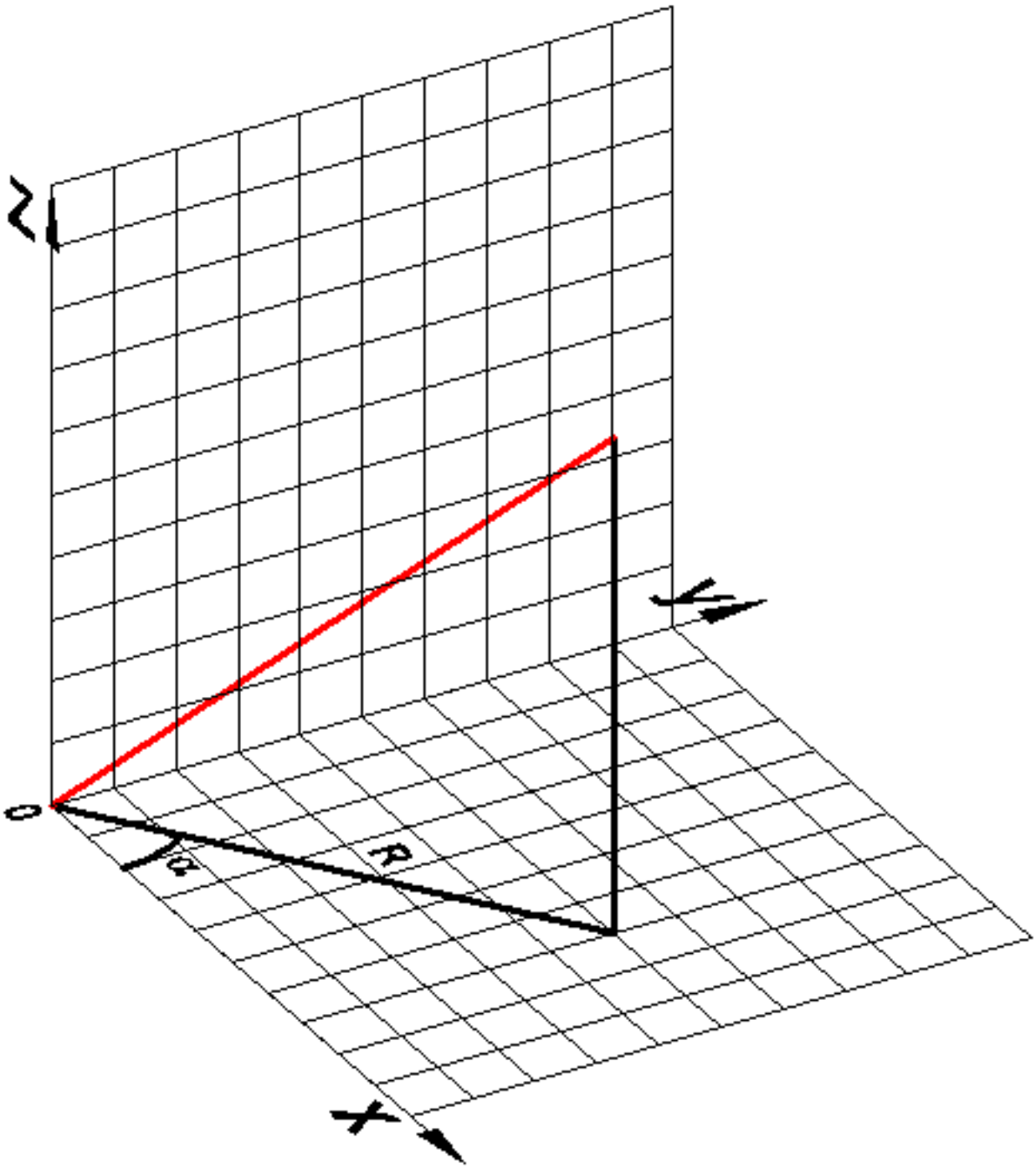
Working with cylindrical coordinates

In a cylindrical coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes. A point is defined using the following format: R<alpha, z.

- R = distance to the origin in the xy-plane
- <alpha = the angle between R and the x-axis (positive angles are measured counter clockwise)
- z = the height above the xy-plane.

If the z-coordinate is omitted, cylindrical coordinates are referred to as polar coordinates. If you place the @-character in front of the entry, the coordinates are calculated with respect to the previous point. This technique is called **Relative Cylindrical coordinates** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sourc

Figure 1.524



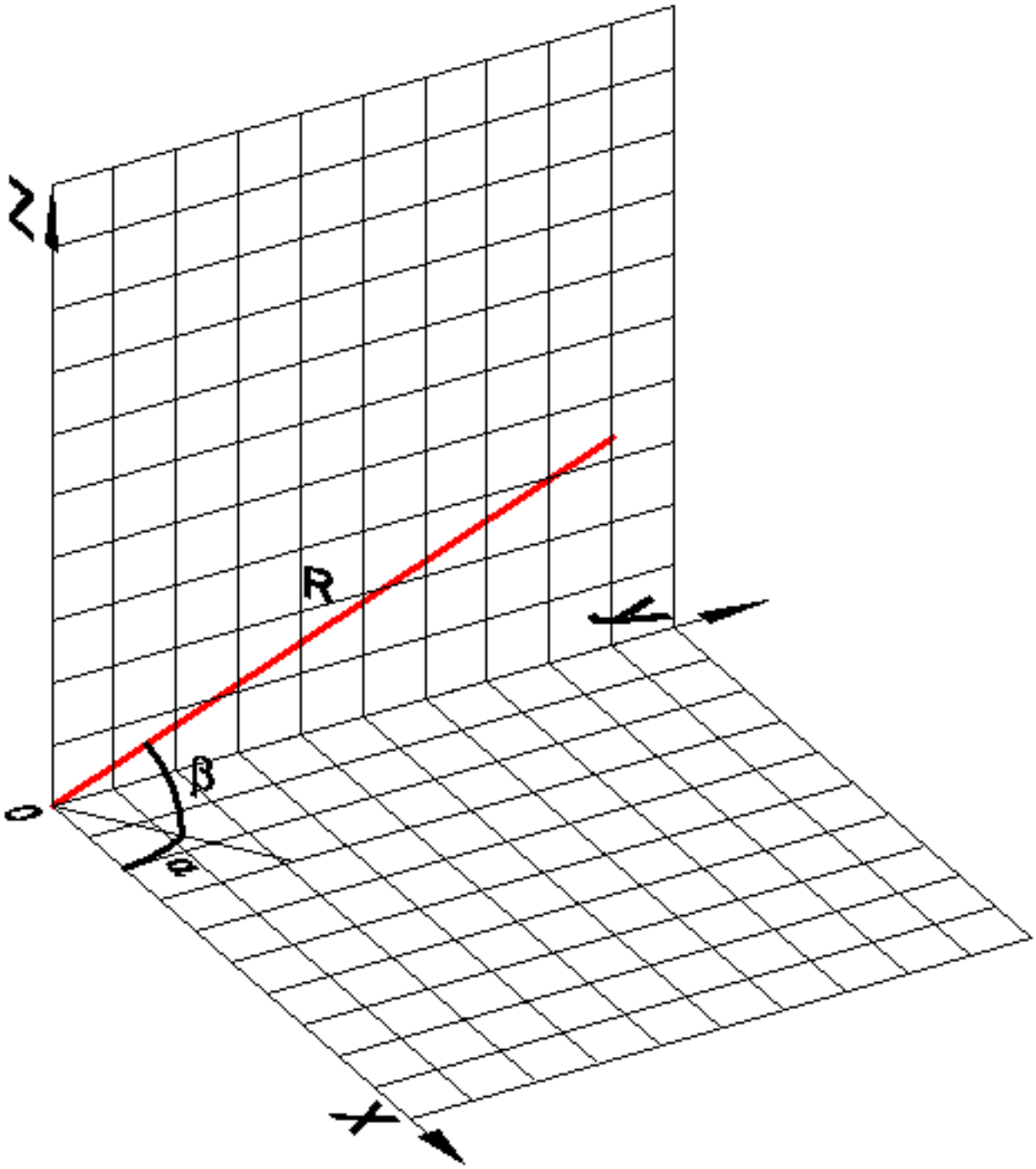
Working with spherical coordinates

In a cylindrical coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes. A point is defined using the following format: R<alpha<beta

- R = distance from the origin
- <alpha = angle in the xy-plane (positive angles are measured counter clockwise)
- <beta = angle measured from the xy-plane (positive angles are measured counter clockwise, above the xy-plane)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/03DrawingAccurately/ART/01Coordin

Figure 1.525



1.16 Viewing your Drawing

1.16.1 View Commands Overview

BricsCAD provides various tools to control the display and view orientation of your drawing. These tools can be found either in the **View** menu or the **View** toolbar or both.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar05Vie
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar05-1R
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar05-2Z
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar05-3R
View toolbar and its flyouts



The Look From control (1.16.8)

Icon	Tool name	Command	Description
	Redraw (1.16.4)	REDRAW (6.26.9)	Refreshes the screen display in the current viewport.
	Redraw All (1.16.4)	REDRAWALL (6.26.10)	Refreshes the screen display in all open viewports.
	Regenerate (1.16.4)	REGEN (6.26.15)	Recalculates the screen display in the current viewport.
	Regenerate (1.16.4) All (1.16.4)	REGENALL (6.26.16)	Recalculates the screen display in all open viewports.
	Update Fields (1.21.23)	UPDATEFIELD (6.30.10)	Updates the selected fields.
	Real Time Pan (1.16.5)	RTPAN (6.26.39)	Pans the view dynamically.
	Pan (1.16.5)	PAN (6.24.2)	Pans the view.
	Real Time Zoom (1.16.6)	RTZOOM (6.26.44)	Zooms in/out dynamically.
	Zoom (1.16.6) In (1.16.6)	ZOOM (1.16.6) **+ I(*)**	Zooms in on the center of the window by a factor of 2.
	Zoom Out (1.16.6)	ZOOM (1.16.6) **+ O(*)**	Zooms out from the center of the window by a factor of 1/2.
	Zoom Extents (1.16.6)	ZOOM (1.16.6) + **E(*)**	Displays all the entities in the drawing (referred to as the drawing extents).
	Zoom Window (1.16.6)	ZOOM (1.16.6) **+ W(*)**	You are prompted to pick two corners of a box on the existing view in order to enlarge that area to fill the display.
	Zoom Previous (1.16.6)	ZOOM (1.16.6) **+ P(*)**	Restores the displayed view prior to the current one.

	**Zoom All (1.16.6)**	ZOOM (1.16.6) **+ A(*)**	Displays the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two).
	**Zoom Left (1.16.6)**	ZOOM (1.16.6) **+ L(*)**	Pick the lower left corner of the next view, then specify the magnification factor or the view height.
	**Zoom Center (1.16.6)**	ZOOM (1.16.6) **+ C(*)**	Pick the center of the next view, then specify the magnification factor or the view height.
	**Zoom Right (1.16.6)**	ZOOM (1.16.6) **+ R(*)**	Pick the upper right corner of the next view, then specify the magnification factor or the view height.
	Look Around **Look Up and Down**	RTLOOK (6.26.37) RTUPDOWN (6.26.42)	Look around in a 3D scene. Move up / down and left / right in a 3D scene.
	Walk	RTWALK (6.26.43)	Walk left / right and forward / backward in a 3D scene.
	Real-Time Constrained Sphere (1.16.8)	RTROT (6.26.38)	Rotates the view dynamically in 3D. • Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the world z-axis. • Moving the mouse vertically (parallel to the screen y-axis) rotates the screen x-axis.
	Real-Time Sphere Center (1.16.8)	RTROTCTR (6.26.47)	Rotates the view dynamically in 3D about a user-defined center point. Behaves like RTROT.

	Real-Time Free Sphere (1.16.8)	RTROTF (6.26.48)	Rotates the view dynamically in 3D. • Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the screen y-axis. • Moving the mouse vertically (parallel to the screen y-axis) rotates the view about the screen x-axis.
	Real-Time X (1.16.8)	RTROTX (6.26.49)	Rotates the view dynamically about the screen x-axis.
	Real-Time Y (1.16.8)	RTROTY (6.26.50)	Rotates the view dynamically about the screen y-axis.
	Real-Time Z (1.16.8)	RTROTZ (6.26.41)	Rotates the view dynamically about the screen z-axis.
	Set View Point (1.16.8)	**DDVPOINT (6.10.22)**	Opens the **Preset Viewpoints** dialog.
	Plan View (1.16.8)	PLAN (6.24.25)	Restores the plan view with respect to the WCS, the current UCS or a saved UCS.
	Define View (1.16.11)	DVIEW (6.10.45)	Defines parallel and visual perspective views.
	Save/Restore view (1.16.10)	VIEW (6.31.1)	Saves and restores named views.
	Viewports (1.16.13)	VPORTS (6.31.20)	Manages viewports in model space.
	Paper Space Views (1.16.14)	MVIEW (6.21.37)	Creates viewports in paper space.
	Generate Drafting Views	**BMGENDRAFT**	Generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
	Generate Section	**BMGENSECTION**	Creates a cross section view based on a standard drafting view generated by the **BmGenDraft** command in a paper space layout.
===ERROR WITH DOWNLOADED IMAGE===	**2D Context**	**2DCONTEXT**	Switches the drawing to a 2D drafting context.
	3D Context	**3DCONTEXT**	Switches the drawing to a 3D modeling context.

(*) <command> <option>: type zoom followed by Enter, then type the command option letter, followed by Enter.

Figure 1.526



Figure 1.527

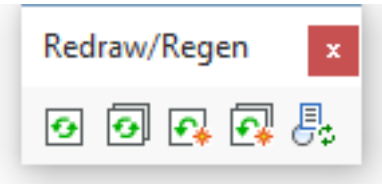


Figure 1.528

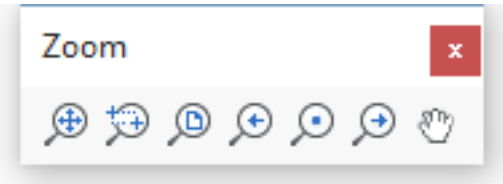


Figure 1.529

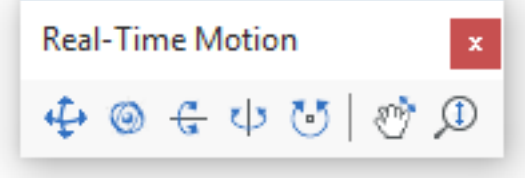
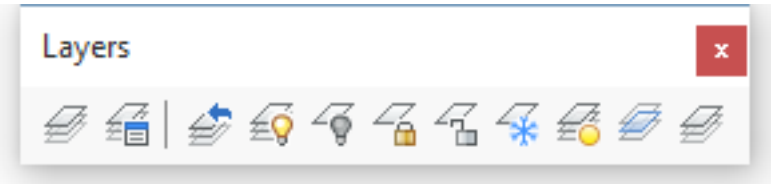


Figure 1.530



1.16.2 Layer Utilities

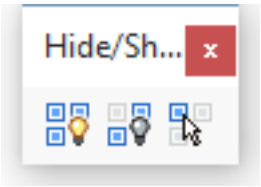
The layer utility commands help you to easily manipulate the **On/Off** , **Thaw/Freeze** and **Unlocked/Locked** layer properties by selecting entities. You can then save the result in a Layer State (1.16.9). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar15-2Layers.png})

Command	Description
LAYON (6.20.13)	Turns on all layers that have been turned off
LAYOFF (6.20.12)	Turns off layers associated with selected entities.
LAYFRZ (6.20.18)	Freezes layers associated with selected entities.
LAYTHW (6.20.16)	Thaws all frozen layers.

LAYLCK (6.20.10)	 Locks layers associated with selected entities.
LAYULK (6.20.17)	 Unlocks layers associated with selected entities.
LAYISO (6.20.9)	 Isolates layers associated with selected entities. If no entity on the current layer is selected, the layer of the entity that is selected first temporarily becomes the current la
LAYUNISO (6.20.18)	 Restores layers isolated by the LAYISO command.

See also Hide and Show Entities (1.16.3).

1.16.3 Hide and Show Entities



The tools on the **Hide/Show Entities** toolbar allow to temporarily hide or isolate a selection of entities.

Command	Icon	Description
UNISOLATEOBJECTS		Brings back all previously hidden entities.
HIDEOBJECTS		Hides the selected entities.
ISOLATEOBJECTS		Displays the selected entities only.

The OBJECTISOLATIONMODE (1.13.11) system variable controls whether hiding and isolating entities is saved or not. See also Layer Utilities (1.16.2).

1.16.4 Redrawing and Regenerating a Drawing

Commands: REGEN (6.26.15), REGENALL (6.26.16), REGENAUTO (6.26.17), REDRAW (6.26.9) and REDRAWALL (6.26.10) The **Regen** command regenerates the current viewport (short for "regenerate"). The **Regenall** command regenerates all viewports (short for "regenerate all"). The **Regenauto** command controls when BricsCAD regenerates the drawing automatically. The **Redraw** command redraws the current viewport to clean it up. The **Redrawall** command redraws all viewports to clean them up. As a matter of fact the screen display of a drawing is a simplified version of the drawing database. From time to time it is necessary to synchronise the screen display and the drawing database. The **REGENMODE** system variable controls whether BricsCAD regenerates the drawing automatically. You can change the status of **REGENMODE** through the **RegenAuto** command. If **REGENMODE** is ON BricsCAD regenerates the display automatically, but in a few cases a forced regeneration of the drawing might be necessary. This is done by the **Regen** command. Don't confuse the **Regen** command with the **Redraw** command, which simply repaints the screen, without attempting to synchronize the screen display with the drawing database.

Regenerate the current viewport

Do one of the following:

- Click the **Regen** tool button () on the **View** toolbar.
- Choose **Regen** in the **View** menu.
- Type **re** or **regen** in the command bar, then press Enter.

The current viewport is regenerated.

Regenerate all viewports

Do one of the following:

- Click the **Regenall** tool button () on the **View** toolbar.
- Type **rea** or **regenall** in the command bar, then press Enter.

All viewports are regenerated.

Redraw the current viewport

Do one of the following:

- Click the **Redraw** tool button () on the **View** toolbar.
- Choose **Redraw** in the **View** menu.
- Type **r** or **redraw** in the command bar, then press Enter.

The current viewport is refreshed.

Redraw all viewports

Do one of the following:

- Click the **Redraw Al** l tool button () on the **View** toolbar.
- Type **ra** or **redrawall** in the command bar, then press Enter.

All viewports are refreshed.

1.16.5 Panning

Commands: PAN (6.24.2) and RTPAN (6.26.39) The **Pan** command moves the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space. The only change is the portion of the drawing being displayed. The cursor changes to a hand ( ) when a **Pan** tool is active. The **Rtpan** command pans the drawing in real-time. See also : View manipulation using the mouse (1.16.7).

Using the pan command

1. To launch the Pan command do one of the following:

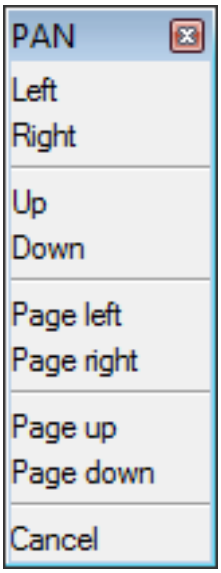
   - Click the **Pan** tool button ( ) on the **View** toolbar (**Zoom** flyout).
   - Choose **Zoom > Pan** in the **View** menu.
   - Type **p** or **pan** in the command bar, then press Enter.

   You are prompted: Left/Right/Up/Down/PGLeft/PGRight/PGUp/PGDown/<Pan base point>: A prompt menu opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/A

2. Specify the **Pan base point** . **You are prompted:** Pan displacement point:

3. Specify the **Pan displacement point** . The display shifts over the specified distance and in the specified direction.

Figure 1.531



Using real time panning

1. To launch the Real-time Pan command do one of the following:

   - Click the **Real-Time Pan** tool button ( ) on the **Real-Time Motion** toolbar
   - Choose **Real-Time Motion** > **Real-Time Pan** in the **View** menu.
   - Type **Rtpan** in the command bar, then press Enter.

   You are prompted: >>ENTER, Right click or Esc to complete.

2. Press and hold the left mouse button to pan the view.
3. To abort the **Real-Time Pan** command, do one of the following:

   - Right click.
   - On the keyboard, press Enter, space bar or Esc.

1.16.6 Zooming

Command: ZOOM (6.35.7) and RTZOOM (6.26.44) The **Zoom** command visually changes the size of the drawing within the current viewport. The **Rtzoom** command zooms the drawing in real-time. You can change the magnification of your drawing at any time by zooming. The cursor changes to a magnifying glass () when a **Zoom** tool is active. Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail. Changing the magnification of the drawing affects only the way the drawing is displayed; it has no effect on the dimensions of the entities in your drawing. See also : View manipulation using the mouse (1.16.7).

Using the zoom command

1. Do one of the following:

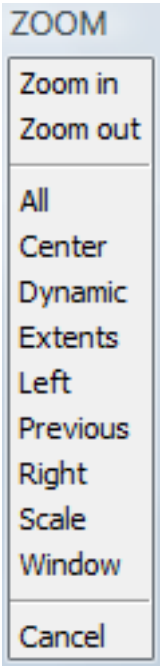
   - Choose **zoom** in the **View** menu.
   - Type **zoom** in the command bar.
   - Type **Z** in the command bar.

   You are prompted: In/Out/All/Center/Dynamic/Extents/Left/Previous/Right/Window/<Scale(nX/nXP)>: A prompt menu opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdra

2. Do one of the following:

   - Type the zoom factor, followed by **X** . E.g. type **2x** to magnify the display 2 times; typing **0.5x** changes the display to half its original size.
   - In paper space viewports (1.16.14), type the zoom factor followed by **XP** to define the zoom factor relative to the viewport, thus defining the scale of the viewport (1.16.14) content.
   - Pick two corners of a box on the existing view in order to enlarge that area to fill the display.

Figure 1.532



Real-time zooming

1. Do one of the following:

   - Click the **Real-Time Zoom** tool button ( ) in the View toolbar.
   - Type **rtzoom** in the command bar, then press Enter.

   You are prompted: >>ENTER, Right click or Esc to complete.

2. Press and hold the left mouse button. Move the mouse forward to zoom in. Move the mouse backward to zoom out.

3. To abort the **Real-Time Zoom** command, do one of the following:

   - Right click.
   - On the keyboard, press Enter, space bar or Esc.

Dynamic zooming

1. Do one of the following:

   - Choose **zoom** in the **View** menu.
   - Type **zoom** in the command bar.
   - Type **Z** in the command bar.

   A prompt menu displays. **You are prompted:**In/Out/All/Center/Dynamic/Extents/Left/Previous/Right/Window/<Scale(n

2. Do one of the following:

   - Choose **Dynamic** in the prompt menu.
   - Type **D** , then press Enter. A view extents or view limits (if larger) is executed first. A view box displays. The original size of the view box matches the size of the previous view. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/04ZoomViewBox.png
     A dashed rectangle indicates the area of the previous view.

3. (option) To resize the view box:

   - Click to switch to resize mode. A right pointing arrow indicates you are in resize mode. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/04ZoomViewBoxRes

Figure 1.533

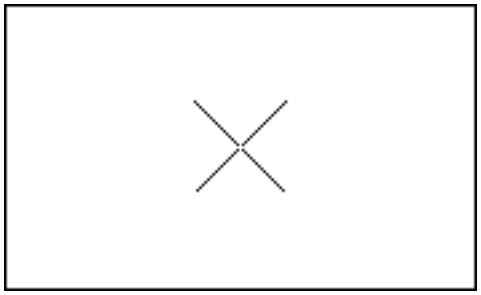
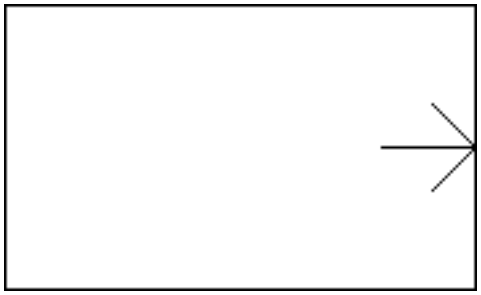


Figure 1.534



- Move the mouse to adjust the size.
- Click to leave resize mode.

4. Move the view box to the area you want to enlarge, then right click. The drawing is zoomed in on the selected area.
5. Repeat steps 1 through 4 to zoom in on another part of the drawing.

1.16.7 View manipulation using the mouse and keyboard

In BricsCAD you can navigate through a 3D model using the mouse and keyboard shortcuts. In perspective mode (1.16.11) you can walk inside the 3D model. In isometric mode you can only zoom in very close, but not go inside.

Mouse / Key	Action	Result
Wheel	scroll	Zoom in / out. The incremental change in zoom with each mouse-wheel action is controlled through the **Zoom Factor** variable (ZOOMFACTOR (1.16.7)). Default = 60. Values between 3 and 100 are accepted. When zooming in the incremental step decreases gradually allowing to focus on a particularly detail easily. In a 3D model the entity under the cursor has the zoom focus. The ZOOMWHEEL (1.13.11) system variable controls the zoom direction: • 0 or OFF: scrolling forwards zooms in, backwards zooms out. • 1 or ON: scrolling forwards zooms out, backwards zooms in.
Middle Button or wheel (**)	press and hold while moving the mouse	Real-time pan (RTPAN (6.26.39) command)
Middle Button or wheel (**)	double click	Zoom extents
Ctrl + Shift + Left Button	press and hold while moving the mouse	Real-time zoom in / out (RTZOOM (6.26.44) command)
Ctrl + Shift + Right Button	press and hold while moving the mouse	Constrained real-time pan (RTPAN (6.26.39) command). Release the Shift key for free panning.
Ctrl + Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time sphere (RTROTF (6.26.48) command)
Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time constrained sphere (RTROT (6.26.38) command)
Ctrl + Right Button	press and hold while moving the mouse	Real-time rotate about the screen Z-axis (RTROTZ (6.26.41) command)
Walk through navigation		
Alt + Left Button	press and hold while moving the mouse	Walk left / right and forward / backward (RTWALK (6.26.43) command)
Alt + Middle Button	press and hold while moving the mouse	Move up / down and left / right (RTUPDOWN (6.26.42) command)
Ctrl + Middle Button	press and hold while moving the mouse	Look around (RTLOOK (6.26.37) command)
Ctrl + Arrow Keys	press the arrow keys while holding the Ctrl key	Walk left / right and forward / backward (RTWALK (6.26.43) command)
Ctrl + Shift + Arrow Keys	press the arrow keys while holding the Alt + Ctrl keys	Look up / down and left / right (RTUPDOWN (6.26.42) command)
Ctrl + Home		Sets the view direction horizontal
Alt + Home		Moves target point to center of scene
Alt + plus/minus		Increase / decrease movement speed

Ctrl + plus/minus	Increase / decrease rotation speed

(**) On condition the **Middle Button Pan** (MBUTTONPAN (1.13.11)) variable is set **ON**.

Setting the Zoom Factor variable

1. Type **zoomfactor** in the command bar, then press Enter. You are prompted: New current value for ZOOMFACTOR (3 to 100) <current value>:
2. Type a value between 3 and 100 in the command bar, then press Enter.

Or

1. In the Settings (1.13.11) dialog, open the **Drawing** category, then expand the **Display/Viewing** sub-category. In the **Viewing** group, select the **ZOOMFACTOR** variable.
2. Type a value between 3 and 100 in the **zoomfactor** field.

NOTE	Increase the zoom factor to speed up scroll wheel zooming in large drawings.

Setting the Middle Button Pan variable

1. Type in **mbuttonpan** the command bar, then press Enter. You are prompted: New current value for MBUTTONPAN (Off or On) <current value>:
2. Type **On** or **Off** in the command bar.

Or

1. In the Settings (1.13.11) dialog, open the **Program Options** category, then expand the **User Preferences** sub-category and select the **MBUTTONPAN** variable.
2. Choose either **Support panning** or **Support action defined in menu file** .

1.16.8 View Rotation

Commands: LOOKFROM (6.20.39), RTROT (6.26.38), RTROTCTR (6.26.47), RTROTF (6.26.48), RTROTX (6.26.49), RTROTY (6.26.50), RTROTZ (6.26.41), DDVPOINT (6.10.22), -VIEW (6.31.2), PLAN (6.24.25) and VPOINT (6.31.19) The **LookFrom** command controls the display of the Look From control (1.16.8). The **Rtrot** (Real-Time Constrained Sphere) command rotates 3D drawings in real-time. The **Rtrotctr** (Real-time Sphere Center) command freely rotates 3D drawings in real-time about a user-defined center point. The **Rtrotf** (Real-time Free Sphere) command freely rotates 3D drawings in real-time. The **Rtrotx** , **Rtroty** and **Rtrotz** commands rotates 3D drawings in real-time about the x, y or z screen axis. The **Ddvpoint** command sets 3D viewpoints or plan view, through a dialog box (short for "dynamic dialog view point"). The **Plan** command displays the plan viewpoint of drawings. The **Vpoint** command changes the 3D viewpoint. In order to view your 3D drawings from any angle, you can rotate a view. The **Real-Time Motion** tools of BricsCAD allow you to rotate a view in real-time. You can rotate the view about the X, Y or Z screen axis or in any direction (real-time sphere). If the **Continuous Motion** variable is set, the view rotation continues until you conclude the **Real-Time Motion** command. See also : View manipulation using the mouse (1.16.7).

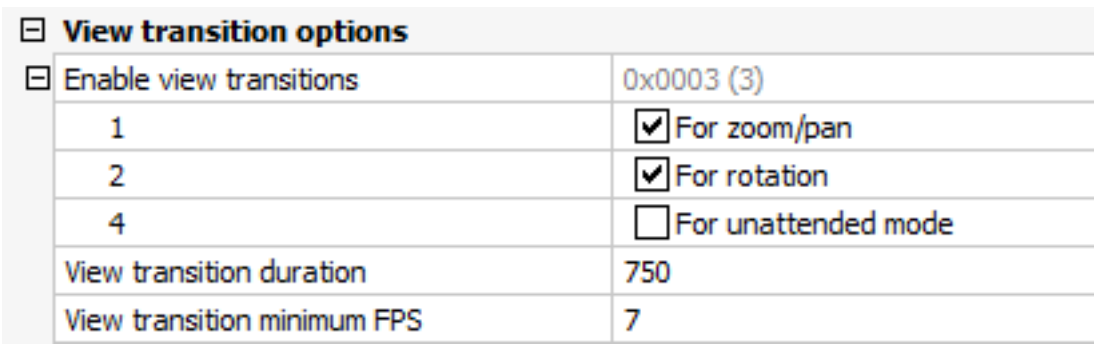
NOTE	'Real-time' commands should not be used when drawing in 2D. Use the Plan View (1.16.8) tool to restore top view.

View Transition options

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/04ViewT

- VTENABLE: determines whether animated view transitions are enabled for zoom/pan and/or for view rotation operations.
- VTDURATION: determines the duration of view transition animations in milliseconds.
- VTFPS: defines the minimum frame rate required to allow animation. That is 7 by default, which means that the redraw time should take less than 143 (=1000/7) milliseconds. If the computer is not capable to redraw the view fast enough, no animation will be available.

Figure 1.535



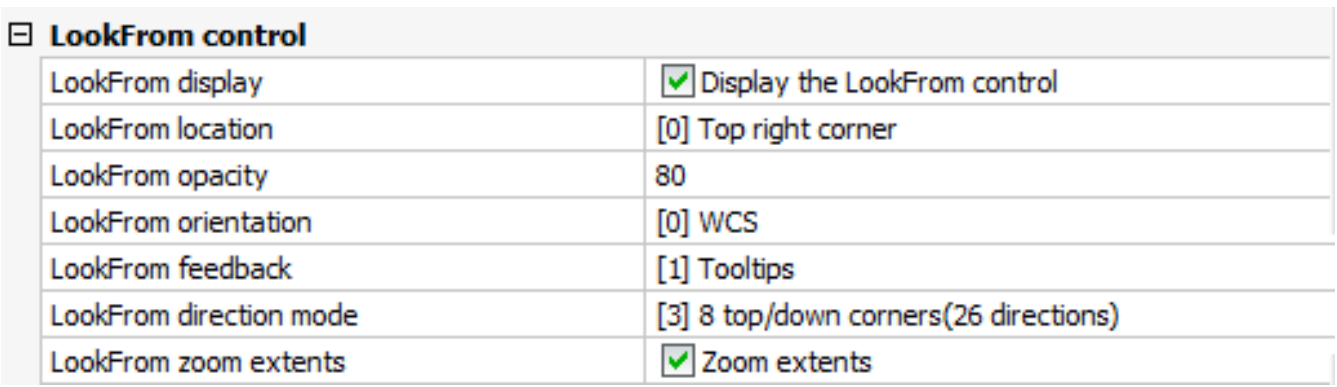
The Look From control

The **Look From** control allows to select a number of preset views such as orthographic views and isometric views. By default the **Look From** control displays in the top right corner of the graphic screen. If the UCSORTHO (1.13.11) system variable is **ON** an orthographic UCS is restored automatically when the related orthographic view is chosen on the **Look From** control.

System variables and user preferences The behavior of the **Look From** control depends on a number of system variables and user preferences (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04View

Name	Title	Description / Options
NAVVIEWCUBEDISPLAY	Look From Display	Controls whether the **Look From** control displays.
NAVVIEWCUBELOCATION	Look From Location	Controls the location of the **Look From** control in the drawing area. The options are: 0: Top right corner. 1: Top left corner 2: Bottom left corner 3: Bottom right corner
NAVVIEWCUBEOPACITY	Look From Opacity	Sets the opacity of the **Look From** control when not active. Values between 1 (invisible) and 100 (no dimming) are accepted.
NAVVIEWCUBEORIENT	Look From Orientation	Specifies the view orientation reference: either the WCS (World Coordinate System or the current UCS (User Coordinate System).
LookFromFeedback	Look From Feedback	Controls the display and location of tooltips. 0: No tooltips 1: Tooltips next to the **Look From** control. 2: Tooltips in the status bar.

Figure 1.536

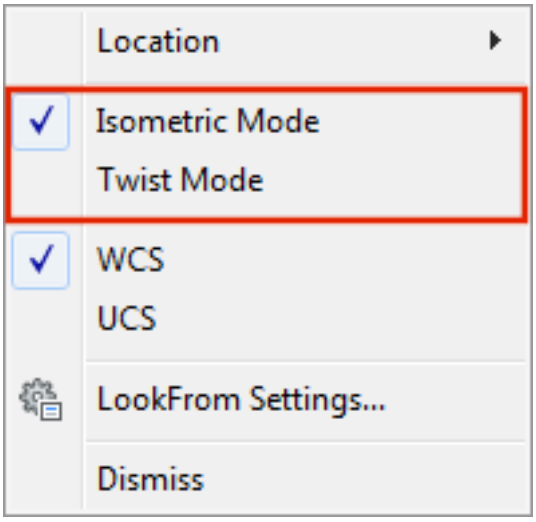


LookFromDirectionMode	Look From Direction Mode	Specifies the number of available view directions. Holding down the Ctrl-key switches between top-down and bottom-up view directions 0: Orthogonal views only (6 directions). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 1: 6 Orthogonal views and 8 Isometric views (14 directions). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 2: 4 Rotated orthogonal, 6 Orthogonal and 8 Isometric views (18 directions) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 3: 8 Rotated isometric, 4 Rotated orthogonal, 6 Orthogonal and 8 Isometric views (26 directions). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
LookFromZoomExtents	Look From Zoom Extents	Determines whether a 'zoom extents' is performed whenever a new view direction is selected.

Setting the Look From mode

1. Move the cursor to the **Look From** control.
2. Right click, then select either **Isometric Mode** or **Twist Mode** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/04LookFromModesConte

Figure 1.537

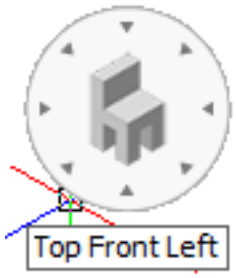


Using the Look From control in Isometric mode Isometric mode is best used in 3D models.

1. Move the cursor to the **Look From** control. The **Look From** control becomes active.
2. Position the cursor on the **Look From** control to choose a view orientation. A preview image and a tooltip indicate the currently selected view orientation. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Usr
3. Click to confirm. The view is updated.

NOTE	VTENABLE setting determines whether animated view transitions are enabled for zoom/pan and/or for view rotation operations. VTDURATION setting determines the duration of view transition animations in milliseconds. VTFPS setting defines the minimum frame rate required to allow animation. That is 7 by default, which means that the redraw time should take less than 143 (=1000/7) milliseconds. If the computer is not capable to redraw the view fast enough, no animation will be available.

Figure 1.538



Using the Look From control in Twist mode

Twist mode is best used in 2D drawings. In **Twist** mode, the **Look From** control allows to rotate the view around the screen Z-axis

1. Move the cursor to the **Look From** control. The **Look From** control becomes active.
2. Do one of the following:
3. Move the cursor on the edge of the the **Look From** control. Left half for counterclockwise, right half for clockwise rotation. Click when the arrow is at the desired rotation angle. The following angles are available: 15°, 30°,45°, 60°, 90°, 120°, 135° and 180°, both clockwise and counterclockwise. If **LookFromFeedback** = 1, the angle displays in a tooltip. Click again to rotate over the same angle again. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/04LookFromTwistAngle.p
4. Move the cursor to the center of the **Look From** control, then click to restore the view (= unrotated state).

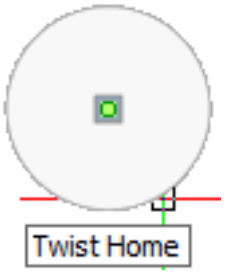


Figure 1.539



Rotating a view freely

1. Do one of the following:

   - Click the **Real-Time Constrained Sphere** tool button ( ) or the **Real-Time Sphere** tool button ( ) on the **Real-Time Motion** toolbar.
   - Choose **Real-Time Motion** > **Real-Time Constrained Sphere** or **Real-Time Motion** > **Real-Time Sphere** in the **View** menu.
   - Type **rtrot** , **rtrotf** or **rtrotctr** in the command bar, then press Enter.

   You are prompted: >> Press ENTER or Esc to complete, or right click to display context menu.

2. Press and hold the left mouse button. Move the mouse to rotate the view.

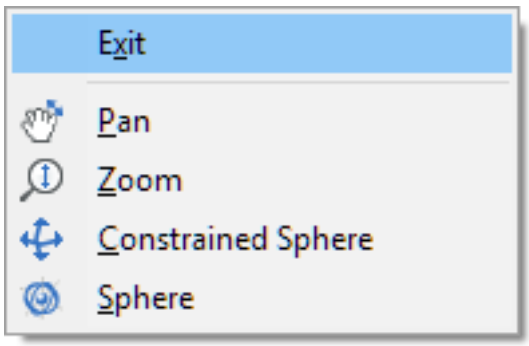
   - Using the **Real-Time Constrained Sphere** command horizontal mouse movement (parallel to the screen X-axis) rotates the 3D model about the world Z-axis.
   - Using the **Real-time Sphere Center** command you are prompted to specify a center point for the rotation first.

3. (option) Right click to display a context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us

4. To abort the **Real-Time Sphere** commands, do one of the following:

   - Right click, then choose **Exit** in the context menu..
   - Press Enter, space bar or Esc.

Figure 1.540



Rotating a view about the view X-axis

1. Do one of the following:

   - Click the **Real-Time X** tool button ( ) on the **Real-Time Motion** toolbar.
   - Choose **Real-Time Motion** > **Real-Time X** in the **View** menu.
   - Type **rtrotx** in the command bar, then press Enter.

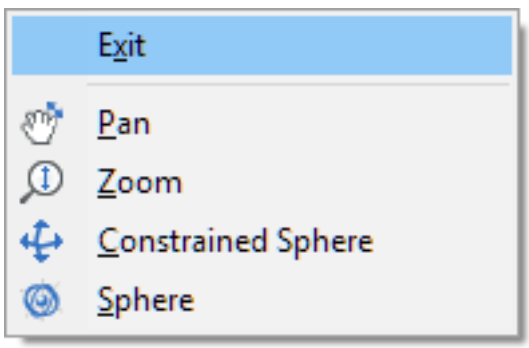
   You are prompted: >> Press ENTER or Esc to complete, or right click to display context menu.

2. Press and hold the left mouse button. Move the mouse to rotate the view.
3. (option) Right click to display a context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us

4. To abort the **Real-Time X** command, do one of the following:

   - Right click.
   - Press Enter, space bar or Esc.

Figure 1.541



Rotating a view about the view Y-axis

1. Do one of the following:

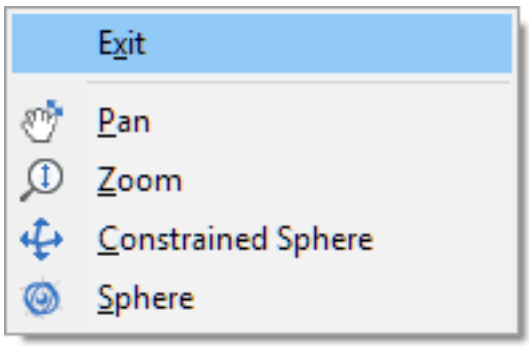
   - Click the **Real-Time Y** tool button ( ) on the **Real-Time Motion** toolbar.
   - Choose **Real-Time Motion** > **Real-Time Y** in the **View** menu.
   - Type **rtroty** in the command bar, then press Enter.

   You are prompted: >> Press ENTER or Esc to complete, or right click to display context menu.

2. Press and hold the left mouse button. Move the mouse to rotate the view.
3. (option) Right click to display a context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us
4. To abort the **Real-Time Y** command, do one of the following:

   - Right click.
   - Press Enter, space bar or Esc.

Figure 1.542



Rotating a view about the view Z-axis

1. Do one of the following:

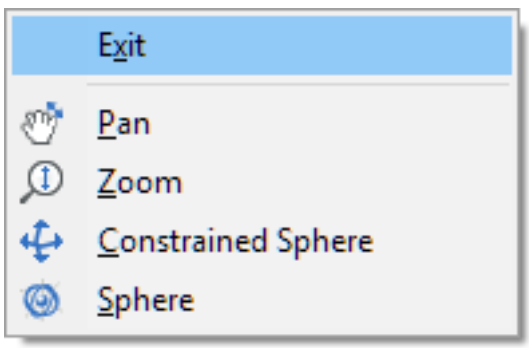
   - Click the **Real-Time Z** tool button ( ) on the **Real-Time Motion** toolbar.
   - Choose **Real-Time Motion** > **Real-Time Z** in the **View** menu.
   - Type **rtrotz** in the command bar, then press Enter.

   You are prompted: >> Press ENTER or Esc to complete, or right click to display context menu.

2. Press and hold the left mouse button. Move the mouse to rotate the view.
3. (option) Right click to display a context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us
4. To abort the **Real-Time Z** command, do one of the following:

   - Right click.
   - Press Enter, space bar or Esc.

Figure 1.543



Using Preset Viewpoints

1. Do one of the following:

   - Click the **Set Viewpoint ...** tool button ( ) on the **View** toolbar.
   - Choose **Set Viewpoint** in the **View** menu.
   - Type **ddvpoint** in the command bar, then press Enter.

   The **Preset Viewpoint** dialog opens.

2. Click the **Angle Precision** button ( ) to choose a different display mode for the **Set Viewpoint** dialog. Then choose either **45°** , **15°** or **5°** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sourc

   45° display mode In **15°** or **5° mode** click in the inner area of the **Horizontal** and **Vertical** fields to choose a**45°** display mode viewpoint. Click in the outer area of the **Horizontal** and **Vertical** fields to choose another angle.

3. To set the **Vertical** view angle, do one of the following:

   - Skip this step to accept the default vertical view directions:
   - Horizontal for East, North, West and South.
   - Downwards for the other directions

   If both the down- and left-arrow show in the **Vertical** field, the default vertical view directions are active.

   - Select a **45° mode** angle in the inner area, then click to confirm your choice. A single arrow indicates the selected vertical view direction.
   - If in **15°** or **5° mode** in, select a view direction in the outer area, then click to confirm your choice. The selected angle displays in bold.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/05
   The view is updated as you click.

4. To set the **Horizontal** view angle, do one of the following:

   - Select a **45° mode** angle in the inner area, then click to confirm your choice. An arrow indicates the selected horizontal view direction. The perspective of the arrow reflects the vertical view direction.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/05

   - If in **15°** or **5° mode** in, select a view direction in the outer area, then click to confirm your choice. The selected angle displays in bold.

   The view is updated as you click.

NOTE	Use the **Isometric Views** toolbar for standard view rotations: top, front, back, left, right and isometric views.

Figure 1.544

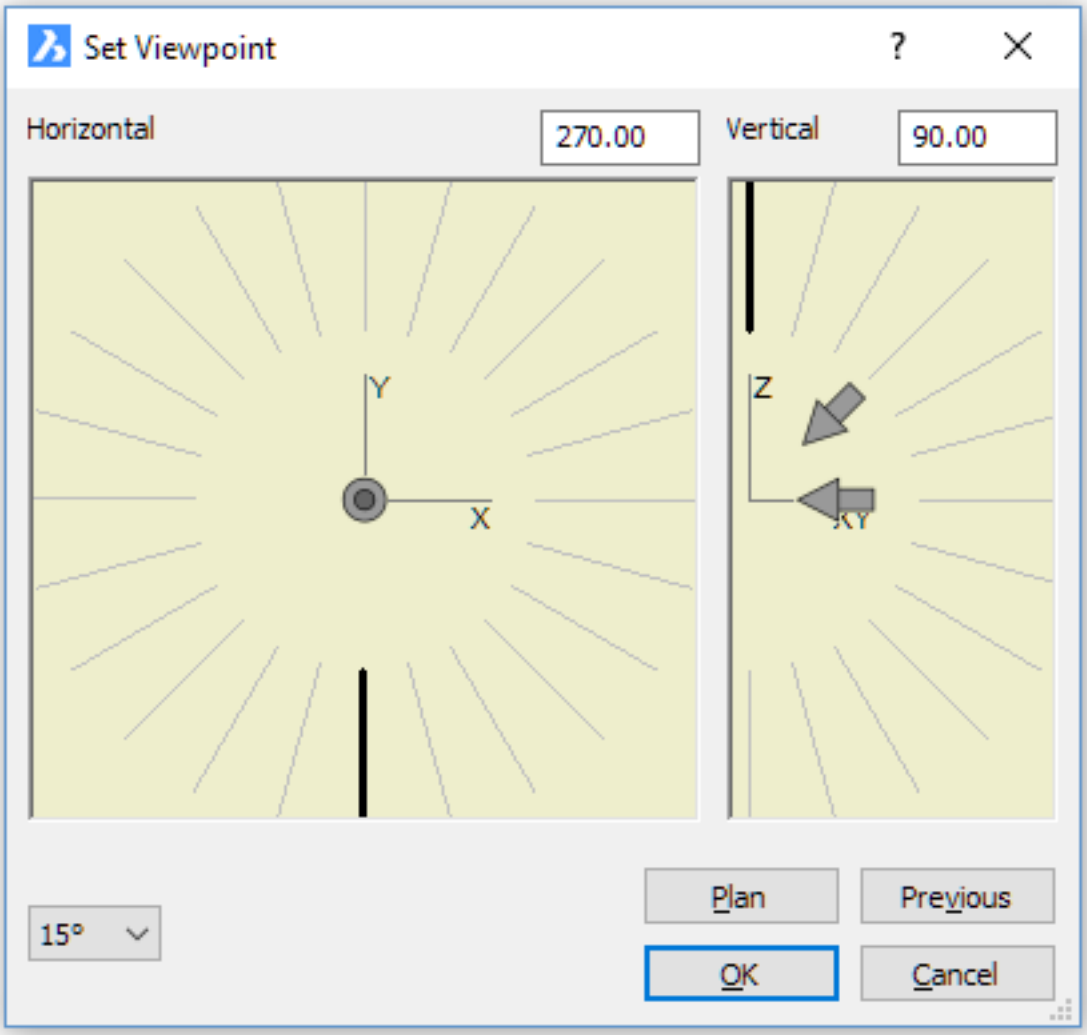


Figure 1.545

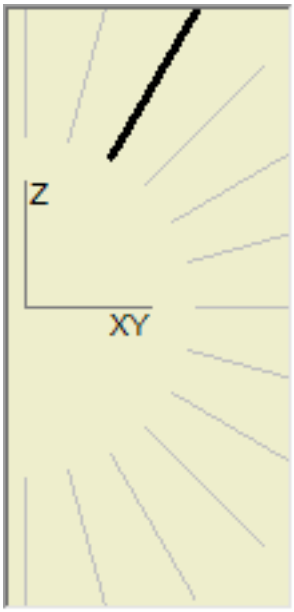
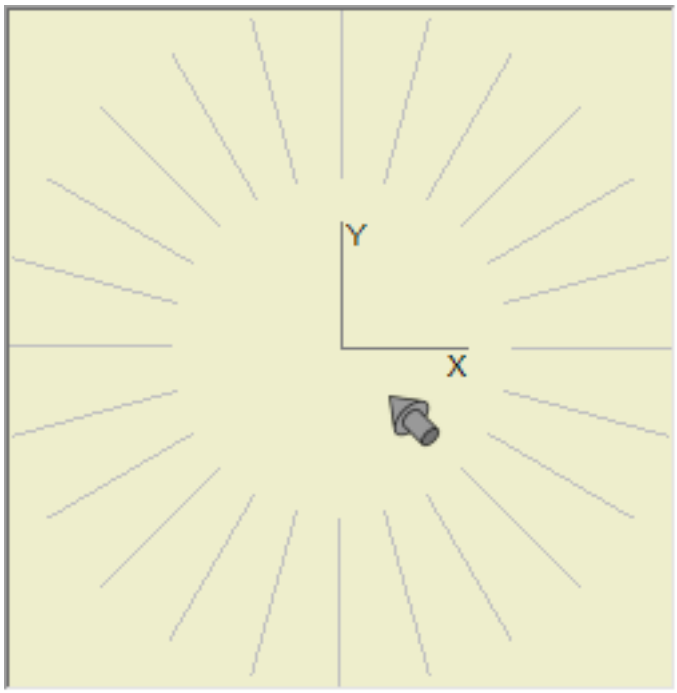
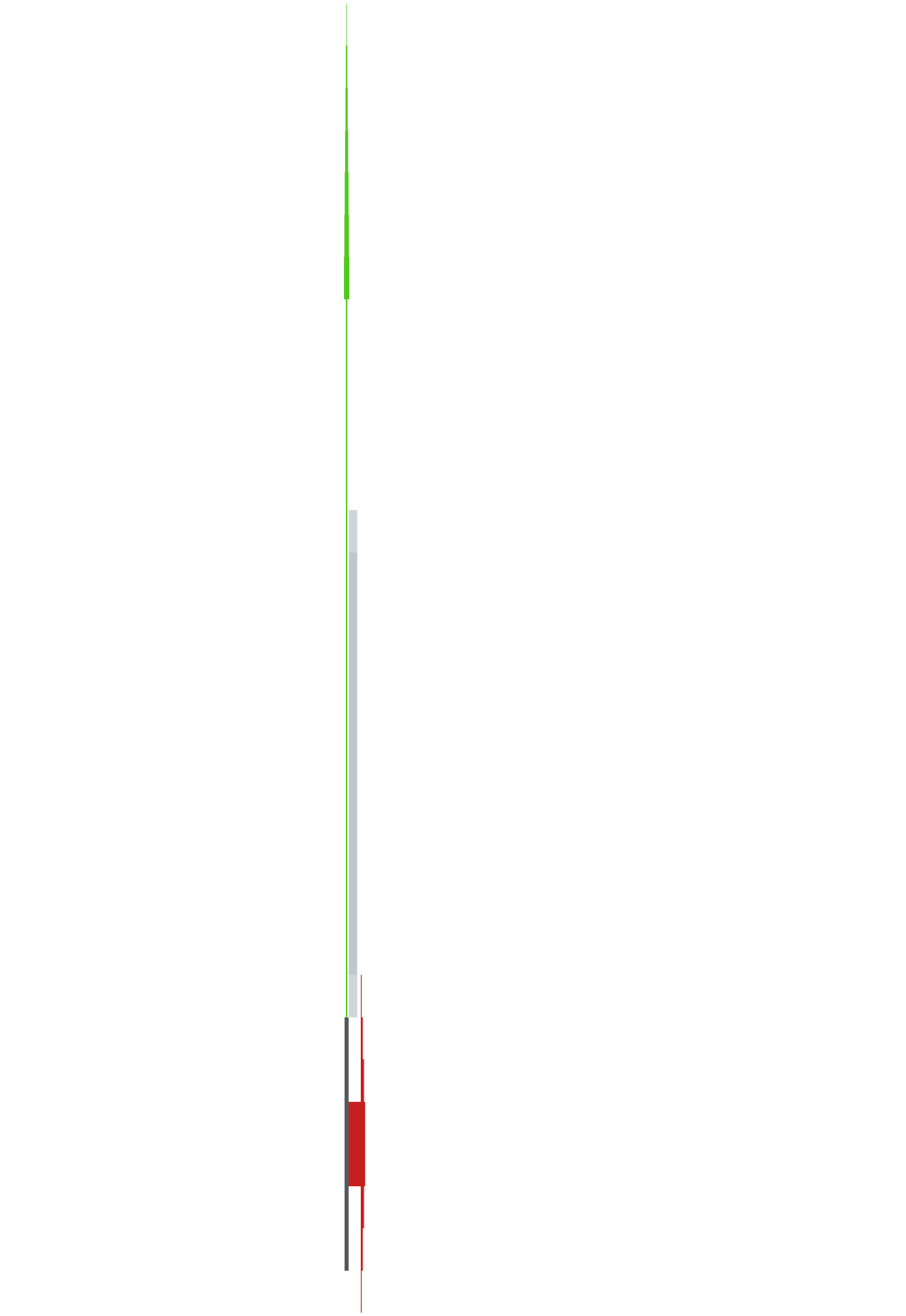


Figure 1.546



Restoring Plan View

1. Do one of the following:



- Click the **Plan View** tool button ( ) on the **View** toolbar.

Figure 1.547

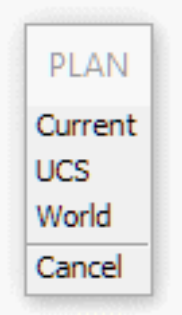
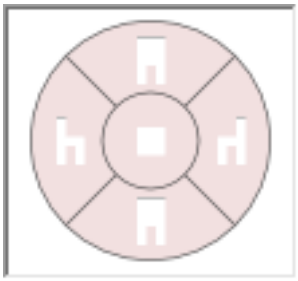


Figure 1.548



- Choose **Plan View** in the **View** menu.
- Type **plan** in the command bar, then press Enter.

You are prompted: Plan view of: UCS/World/<current UCS>: A prompt menu opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/05PlanPromptBox.png})

2. Press Enter to restore the plan view with respect to the current coordinate system.

NOTES

- If the WCS is the current coordinate system, the **Current** and **World** options have the same result.
- If the **UCSFOLLOW** variable is set to **ON** , the plan view is generated whenever the UCS changes.

Figure 1.549

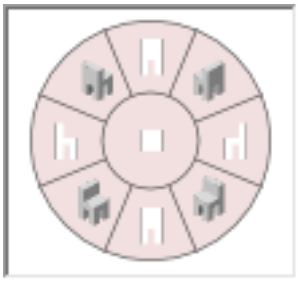


Figure 1.550

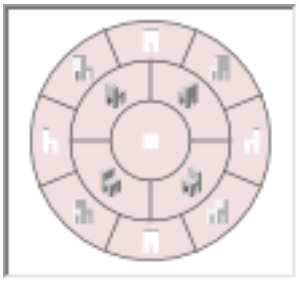


Figure 1.551

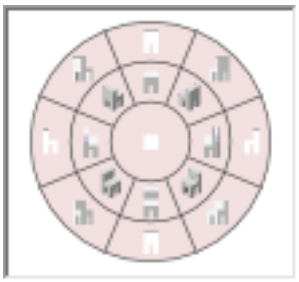
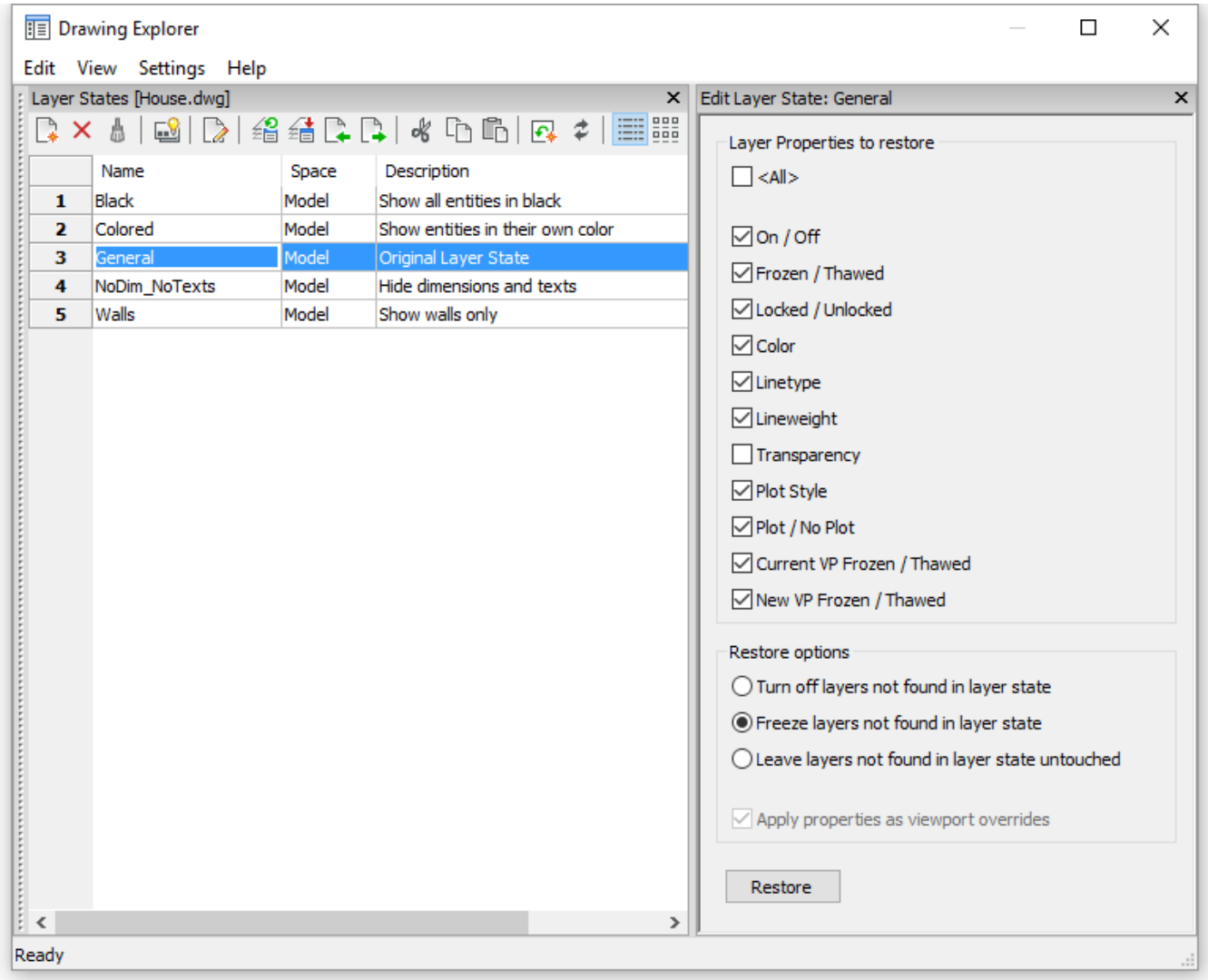


Figure 1.552



1.16.9 Layer States

Command: LAYERSTATE (6.20.7) Layer states let you save and restore configurations of layer properties and states. You might want a layer to display in blue sometimes and green at other times, or you need some layers to be off or frozen or locked when editing a specific part of a drawing. After spending a lot of time adjusting all layer settings you can use layer states to save your settings. You can save multiple layer states in a each drawing. Layer states can be copied between drawings and you can save a layer state to an external file, which can then be imported in another drawing. To easily create layer states, the Layer Utility (1.16.2) commands might be useful. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/08LayerStates
Drawing Explorer: Layer States (Detail view)

Tutorial: Use Layer States instead of Clicking Bulbs Click here (1.2.8) to watch

Opening the Layer States Explorer

1. In the **Tools > Drawing Explorer** menu choose **Layer States...**.
2. (option) Do one of the following:
   - Click the **Detail View** button ( ). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/s
   - Click the **Icon View** button ( ). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sc

Figure 1.553

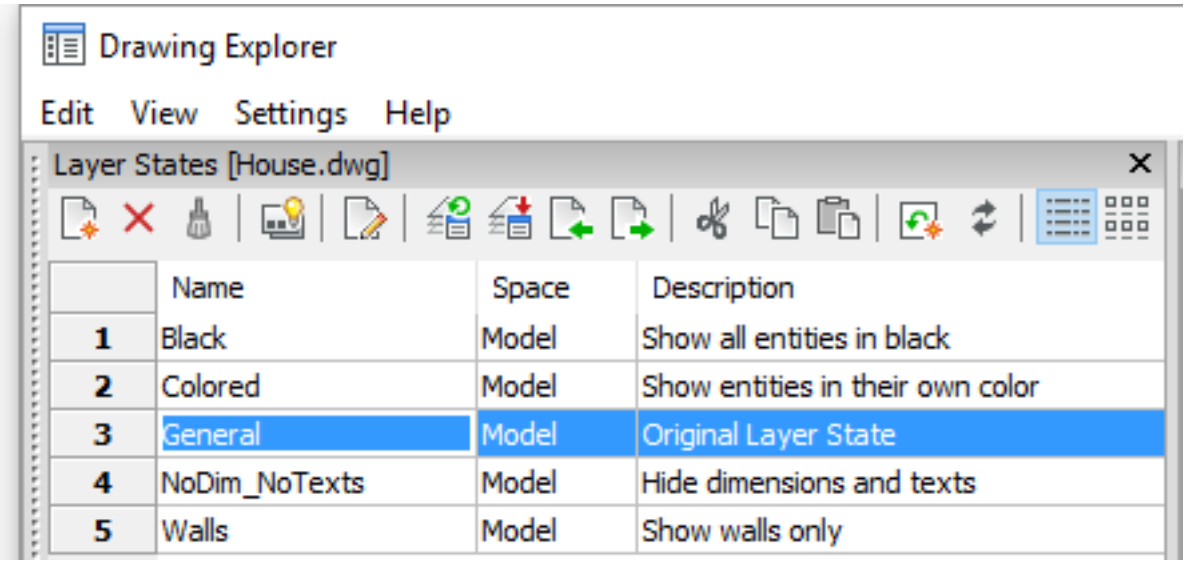
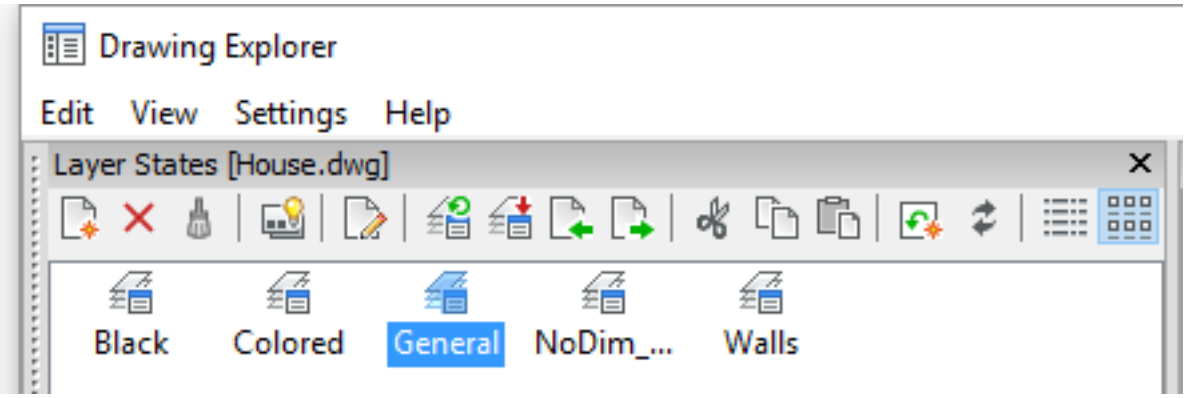


Figure 1.554

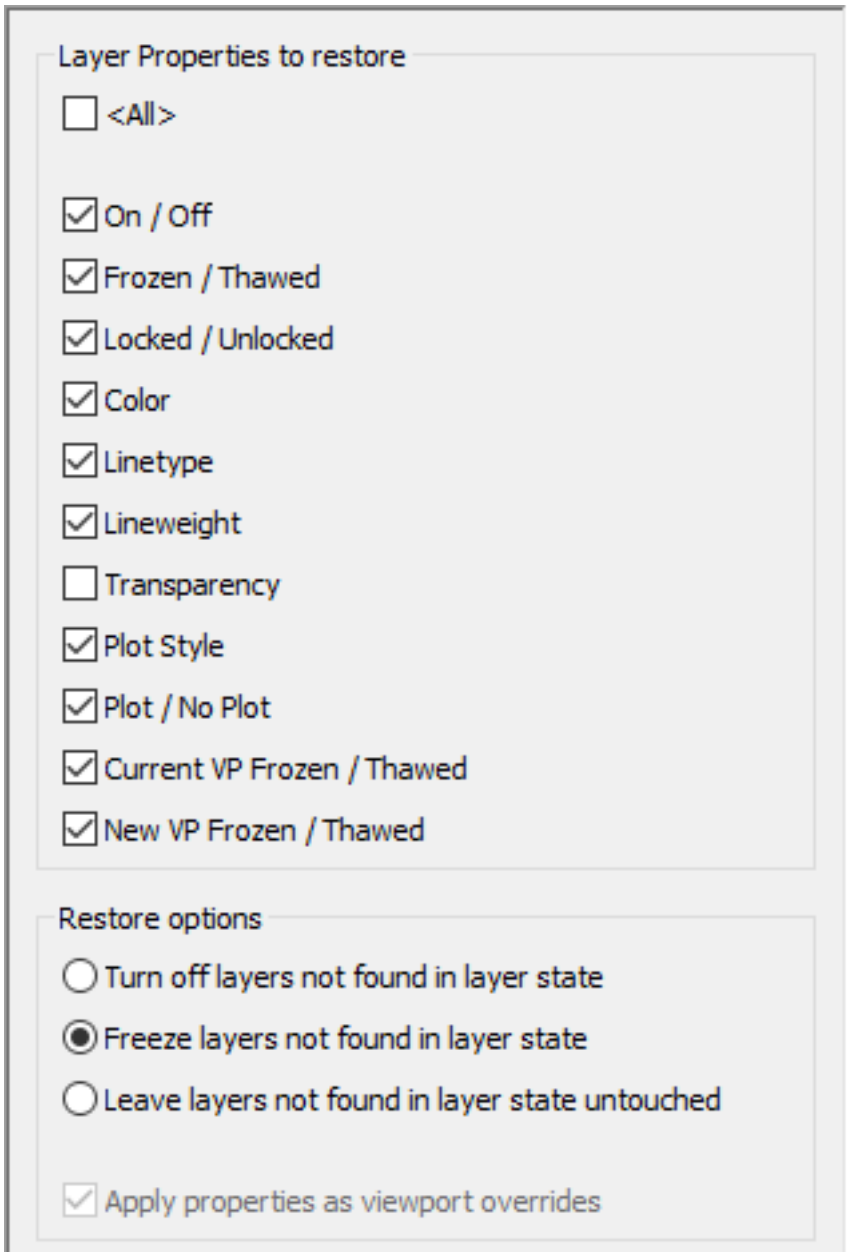


To save the current layer state

1. Open the Layer States explorer (1.16.9).
2. Click the **New** button. (). A new layer state is created.
3. Type a name for the layer state, replacing the **NewLayerState1** default name.
4. (option) When in **Detail View** type a description in the **Description** field.
5. Select the **Layer Properties to restore** . Click the check box to select a layer property, click again to remove the selection. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawir

NOTE	By default in a new layer state all layer properties are selected to be restored.

Figure 1.555


Layer Properties to restore
<All>
On / Off
Frozen / Thawed
Locked / Unlocked
Color
Linetype
Lineweight
Transparency
Plot Style
Plot / No Plot
Current VP Frozen / Thawed
New VP Frozen / Thawed
Restore options
Turn off layers not found in layer state
Freeze layers not found in layer state
Leave layers not found in layer state untouched
Apply properties as viewport overrides

To edit a layer state

1. Open the Layer States explorer (1.16.9).
2. (option) Click the **Overwrite** button () to overwrite the layer state with the current layer settings.
3. If necessary, click the **Tree View** button ().
4. Click the expand button of the layer state.
5. (option) Rename the layer state.
6. (option) Edit the layer state description.
7. (option) Expand the **Layer Properties to restore** list. Click the check box to select a layer property, click again to remove the selection.

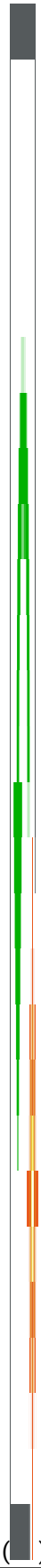
To restore a layer state

1. Open the Layer States explorer (1.16.9).
2. Select the layer state you want to restore.
3. Select a **Restore Option**. The restore option applies to layers which are added after the selected layer state was created. The options are:

   - Turn Off.
   - Freeze.
   - Do nothing.

4. Optionally check the **Apply properties as viewport overrides** option, which applies the layer state as viewport overrides (VP layer properties (6.20.2)) to the current layout viewport. This option is available only on a layout within a layout viewport.
5. Do one of the following:

   - Click the **Restore** button ().
   - Right click, then choose **Restore** in the context menu.



6. Click the **Regen** button ( )





- If **REGENAUTO** is on ( ), it is not necessary to press the **Regen** button in step 4.

To copy layer states to another drawing

1. Open both, the source and the target drawing.
2. Make the source drawing the current drawing.
3. Open the Layer States explorer (1.16.9).
4. If the layer states display in **Tree View** , choose either **Detail View** () or **Icon View** ().
5. Do one of the following
   - Select the layer state you want to copy.
   - Press and hold the Ctrl key to select multiple layer states.
6. With the cursor on a selected layer state, press and hold the left mouse key, then drag the selection to the target drawing in the **Open Drawings** section of the Drawing Explorer (1.14.2)dialog window.
7. When on the target drawing, release the left mouse button. The selected layer states are copied to the target drawing.

To export a layer state

1. Open the Layer States explorer (1.16.9).
2. Select the layer state.
3. Click the **Export** button (). The **Export Layer States** dialog opens.
4. In the **File Name** field, type a name for the layer state.
5. Choose a folder to save the layer state in.
6. Click the **Save** button.

To import a layer state

1. Open the Layer States explorer (1.16.9).
2. Click the **Import** button (). The **Import Layer States** dialog opens.
3. Browse to the folder where you have saved the layer state file.
4. Select the layer state.
5. Click the **Open** button. The layer state is imported.

NOTE	If a layer state of the same name already exists in the drawing a warning displays. It is not possible to overwrite a layer state definition.

Tutorial: Employing the Display Power of Layer States Click here (1.3.18) to watch

1.16.10 Named Views

Commands: -VIEW (6.31.2), VIEW (6.31.1), CAMERA (6.9.2) You can save the view in the current window as a named view. Saved views can be restored at any time. A Background can be assigned to a named view, which will be used when the view is rendered (1.26.2)

Exploring views

1. Do one of the following:

   - Choose **Drawing Explorer > Views...** in the **Tools** menu.
   - Type either **view** or **V** in the command bar, then press Enter.

   The **Drawing Explorer - Views** window opens.

2. (option) Click the blank tile in front of the **View Name** to restore a saved view. The current view is marked ( ). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/07

3. (option) Click the **New** button ( ) in the **Drawing Explorer** toolbar to save the current view.

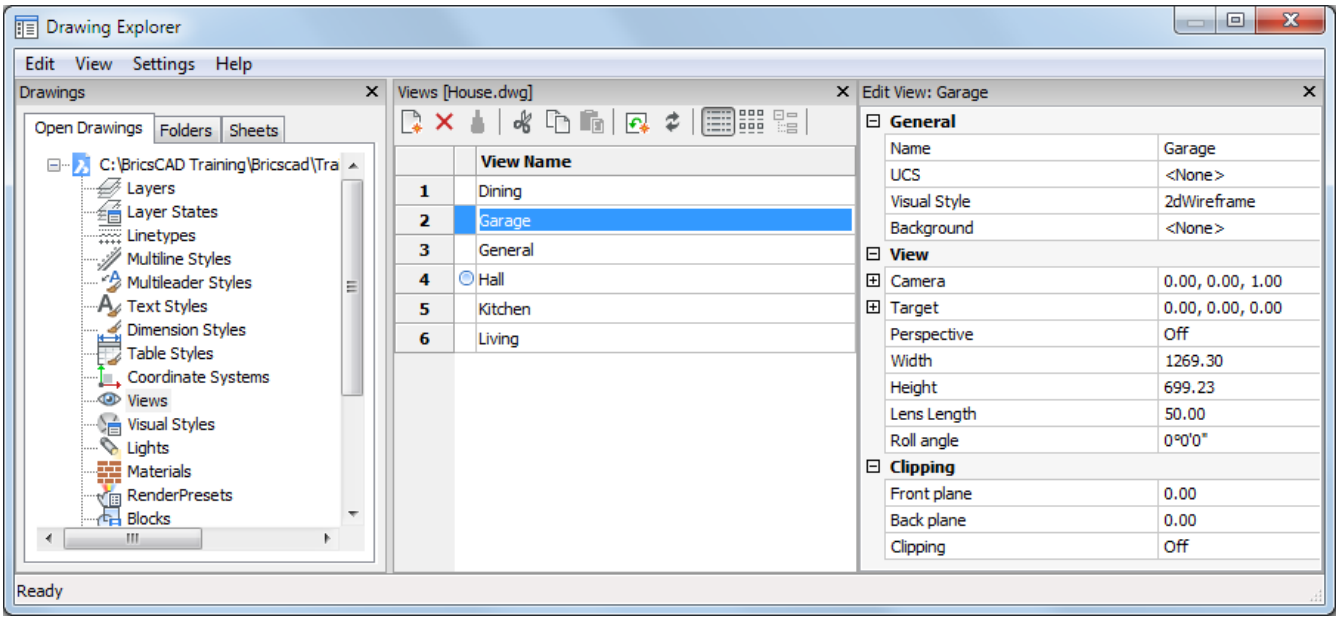
4. (option) Click the **Delete** button ( ) in the **Drawing Explorer** toolbar to delete the selected view.

5. (option) Click the **View Name** , then right click and choose Rename in the context menu. Type a new name for the saved view.

6. (option) Check the **Perspective** property to make the view a visual perspective view. If not checked the view is a parallel perspective view. See also Define a View (1.16.11).

7. (option) Click the **Background** column of a view to choose a background. The background definition of the view is applied during rendering.

Figure 1.556



Saving a view

1. Do one of the following:

   - Click the **Save/Restore View** tool button ( )
   - Choose **Save/Restore View** in the **View** menu.
   - Type **-view** in the command bar, then press Enter.

   You are prompted: View: ? to list saved views/Delete/Restore/Save/Window: (\vref {fig:C:/Dev/Task-2448/images/static

2. Do one of the following:

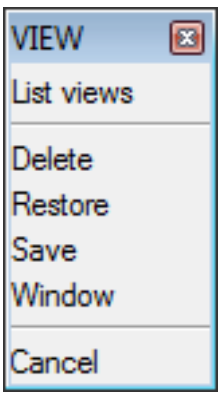
   - Choose **Save** in the **View** prompt menu.
   - Type **S** in the command bar, then press Enter

3. Type a name for the view in the command bar, then press Enter. The current view is saved.

NOTE	If you type a name that is already used, the current view is saved, replacing the previously saved view. There is no warning when overwriting a saved view.

Tutorial: How to navigate quickly around your drawing Click here (1.2.2) to watch

Figure 1.557



Restoring a view

1. Do one of the following:

   - Click the **Save/Restore View** tool button ( ) on the **View** toolbar.
   - Choose **Save/Restore View** in the **View** menu.
   - Type **-view** in the command bar, then press Enter.

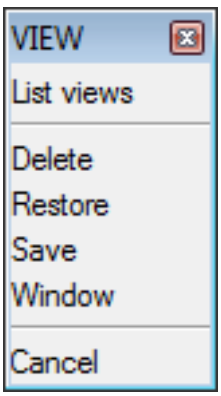
   You are prompted: View: ? to list saved views/Delete/Restore/Save/Window: (\vref {fig:C:/Dev/Task-2448/images/stati

2. Do one of the following:

   - Choose **Restore** in the **View** prompt menu.
   - Type **R** in the command bar, then press Enter

3. Type the name of the saved view you want to restore in the command bar, then press Enter. The saved view is restored.

Figure 1.558



Defining the view background

1. In the Drawing Explorer - Views dialog window, select **Background** in the **Edit** grid of the view.
2. Click the settings field to display the **Background** options. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
3. Choose an option in the list. The **Background** dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom
4. (option) Select **Solid** in the **Type** list. Click the **Color** button to define the color (1.17.4).
5. (option) Select **Gradient** in the **Type** list.
   - Set the rotation angle
   - Choose **Two** or **Three** colors.
   - Click the **Top** , **Middle** and **Bottom** color buttons todefine the color (1.17.4).
6. (option) Select **Image** in the **Type** list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/so
   - Click the browse button next to the Image settings field to select a raster image file.
   - Choose an alignment. The options are: **Center** , **Stretch** and **Tile** .
7. (option) Right click, then choose **Set Current** in the context menu if you want to use the newly defined background in a rendering right away.

Figure 1.559

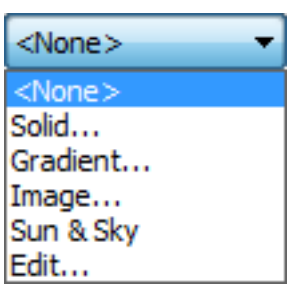


Figure 1.560

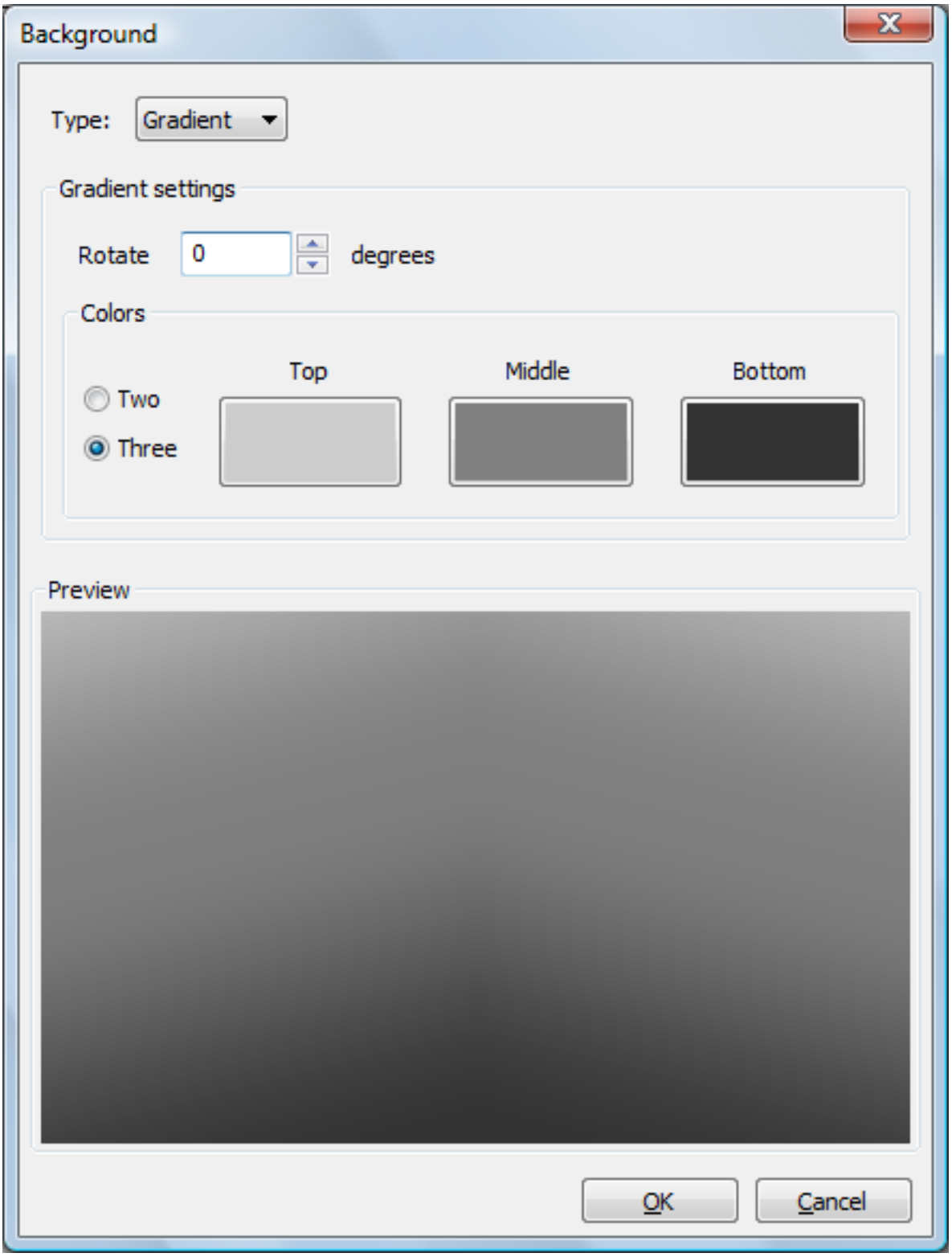
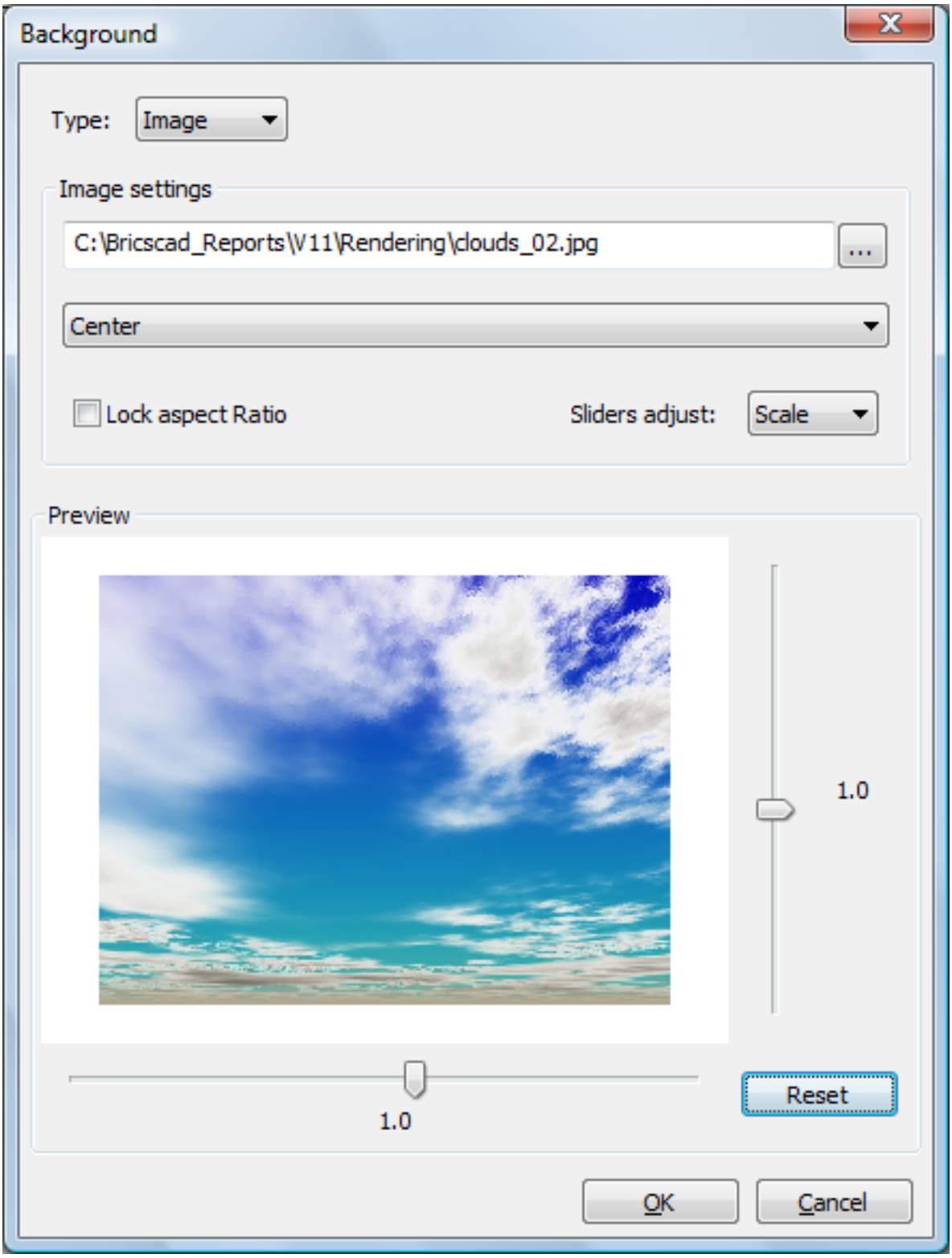

Background
Type:
Gradient
Gradient settings
Rotate
0
degrees
Colors
Top
Middle
Bottom
Two
Three
Preview
OK
Cancel

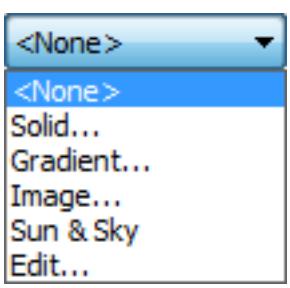
Figure 1.561


Background
Type:
Image
Image settings
C:\Bricscad_Reports\V11\Rendering\clouds_02.jpg
...
Center
Lock aspect Ratio
Sliders adjust:
Scale
Preview
1.0
1.0
Reset
OK
Cancel

To edit the view background

1. In theDrawing Explorer - Viewsdialog window, select **Background** in the **Edit** grid of the view..
2. Click the settings field to display the **Background** options. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
3. Choose **Edit...** in the list to edit the current settings. The **Background** dialog box displays.
4. Edit the settings needed, then click the **OK** button.

Figure 1.562



1.16.11 Perspective Views

Commands: DVIEW (6.10.45) and CAMERA (6.9.2) The **Define View** command dynamically defines parallel perspective or visual perspective views of your 3D models. The **Camera** command creates perspective views. The projection method of a view is controlled by the **Perspective** property of the view:

- If ON, the view is a visual perspective view.
- If OFF, the view is a parallel perspective view.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/06Parallel}	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/06Parallel}
Parallel Perspective View	Visual Perspective View

The following parameters are used in the **Define View** procedure:

- **Target Point** : Defines the center of the view (1)
- **Camera Point** : Defines the view point and view direction (2).
- **Distance to Target** : Distance between the camera position an the target point (3).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/06Came

NOTE	View manipulation tools, such as zoom, pan, real-time motion, are not available in visual perspective views. However, you can use the mouse to manipulate the view (1.16.7).

Defining a camera view

The **Camera** command creates a named view (1.16.10). Do one of the following:

- Click the **Camera** tool button ( ) on the **View** toolbar.
- Choose **Camera** in the **View** menu.
- Type **Camera** in the command bar

You are prompted: Specify camera location: Do one of the following:

- Click a point in the drawing.
- Type the coordinates of the camera location in the command bar and press Enter.

You are prompted: Specify target location: Do one of the following:

- Click a point in the drawing.
- Type the coordinates of the target location in the command bar and press Enter.

You are prompted: Enter an option: ?/Name/LOcation/Height/Target/LEns/Clipping/View/<eXit>: Do one of the following:

- Press Enter to create the camera view using a default name (e.g. Camera1).
- Enter an option:
  - Name : **Prompts you:** Enter name for new camera <Camera1>: Type a name at the command prompt, then press Enter.
- Location : **Prompts you:** Specify camera location <current location>: Click a point or enter new coordinates of the camera location, then press Enter.
- Height : **Prompts you:** Specify camera height <current height>: Enter a new height (Z-coordinate) for the camera location.
- Target : **Prompts you:** Specify target location <current location>: Click a point or enter new coordinates of the target location, then press Enter.

- Lens : **Prompts you:** Specify lens length in mm <50.00>: Enter an new lens length, then press Enter.
- Clipping : **Prompts you:** Enable front clipping plane? <Yes>/No: Choose Yes to define the front clipping plane. **Prompts you:** Specify front clipping plane offset from target plane <0.00>: Type value in the command bare or specify the distance by clicking two points. **Prompts you:** Enable back clipping plane? Yes/<No>: Choose Yes to define the front clipping plane. **Prompts you:** Specify front clipping plane offset from target plane <0.00>: Type value in the command bare or specify the distance by clicking two points.
- View : **Prompts you:** Switch to camera view? Yes/<No>: Choose Yes to display the camera view. This option concludes the **Camera** command.

NOTE	Choose **Drawing Explorer > Views...** in the **Tools** menu to edit a camera view.

To toggle the perspective property of view

Do one of the following

- Click the **Perspective** tool button ( ) on the **View** toolbar. The state of the tool button, pressed or not, indicates the current value of the PERSPECTIVE system variable.
- Choose **Perspective** in the **View** menu.
- Type **perspective** in the command bar. **You are prompted:** New current value for PERSPECTIVE (Off or On) <On>: A prompt menu displays. Do one of the following:
  - Type **off** or **on** in the command bar, then press Enter.
  - Choose **Off** or **On** in the prompt menu.

NOTE	In named views you can set the **Perspective** property in the Drawing Explorer - Views (1.16.10) dialog.

Tutorial: How to navigate quickly around your drawing Click here (1.2.2) to watch

Figure 1.563

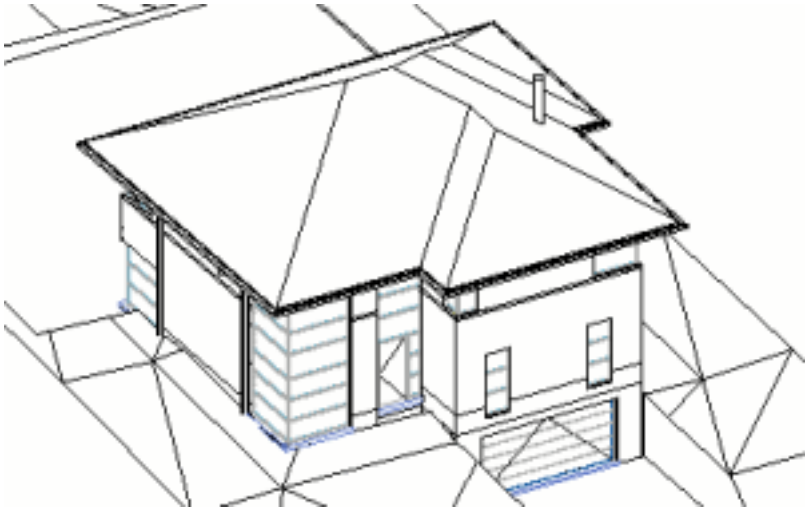


Figure 1.564

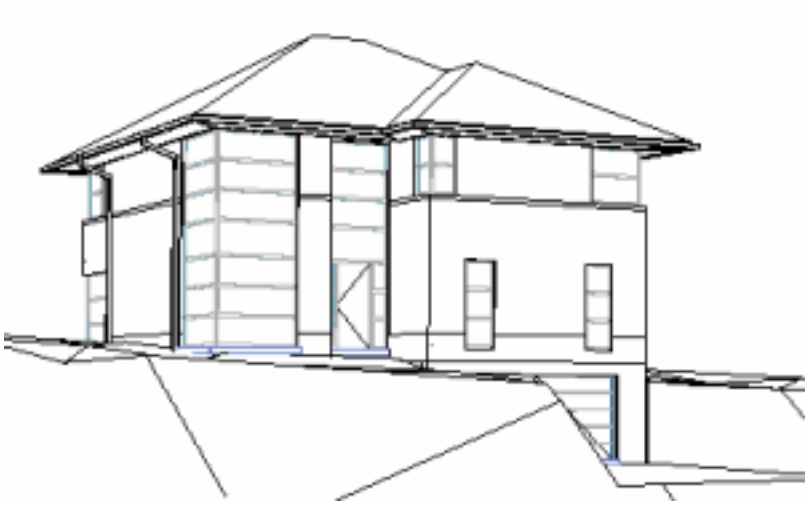
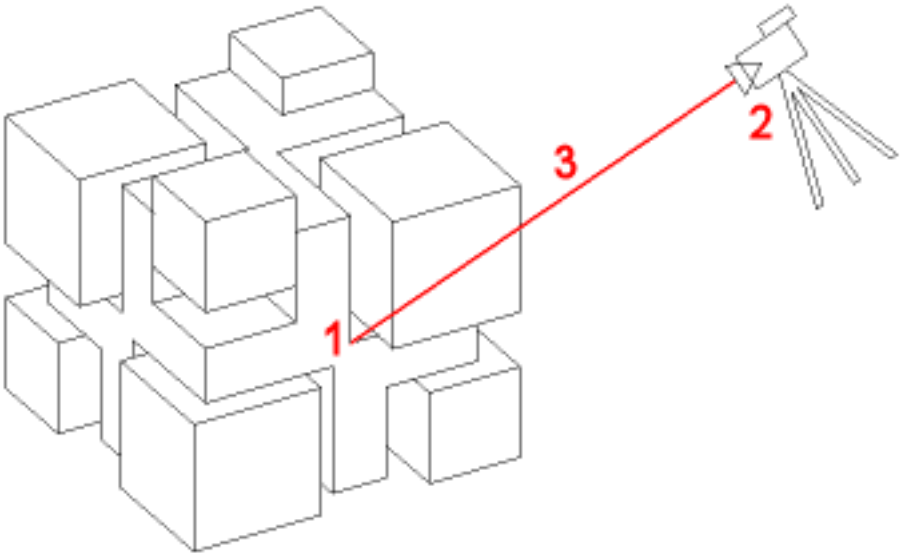


Figure 1.565



1.16.12 Model space and paper space

Understanding paper space and model space

When you start a drawing session, your initial working area is called **Model Space** . Model Space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or a user coordinate system (UCS). You view and work in model space while using the **Model** tab. In general model space consists of a single view that fills the screen. If needed, you can create additional views, called viewports, which can show different views of your drawing or 3D model. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. Click in a viewport to make it the current viewport. You can print the current viewport only. The Viewports (1.16.13) command lets you manage your viewports in model space. BricsCAD provides an additional work area, called **Paper Space** . Paper space represents a paper layout of your drawing. In this work area, you can create and arrange different views of your drawing similar to the way you arrange drawings on a sheet of paper. In paper space you can also add keynotes, annotations, borders, title blocks, and other print-related entities, which you don't want to see in model space. Each drawing has at least one **Layout** in which you can have one or more **Layout Viewports** . Such layout viewports are to be considered as a view window on your drawing in model space. Layout viewports can be placed anywere in a paperspace layout. Each viewport has its own scale and layer visibility. All viewports in the same layout can be printed simultaneously. The Mview (1.16.14) command lets you manage your viewports in paper space. Although not necessary to print your drawing, paper space offers a lot of advantages:

- Create multiple layouts to print the same drawing with different print settings, such as pen widths, printer configuration files, lineweight settings, drawing scale, and more.
- Add print-related entities that are not essential to the model itself, such as keynotes, annotations, title blocks, etc.
- For a single layout, create multiple layout viewports to print multiple views of your drawing at different scales.

Switching between model space and paper space

- To switch between Model Space and Paper Space Layouts use the Model and Layout tabs (1.16.12) at the bottom of the drawing window.
- When working in a layout, use the MSPACE (6.21.34) and PSPACE (6.24.54) commands to toggle between paper space and model space (1.16.12).

Tutorial: Paper Space Layouts in BricsCAD Click here (1.8.2) to watch

Using the model and layout tabs

1. To open model space, click the **Model** tab (2) at the bottom of the drawing window.
2. To open a paper space layout, click the corresponding **Layout** tab (3) at the bottom of the drawing window.
3. Use Ctrl-PgUp or Ctrl-PgDn to switch to the previous or next layout tab.
4. Click the navigation buttons to shift the tab display (1):
5.  : Display the model tab.
6.  : Shift one tab to the left.
7.  : Shift one tab to the right.
8.  : Shift to the last tab.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/08Layou

NOTE A drawing contains at least one layout, which is named **Layout1** by default.

Toggle between model space and paper space in a layout

Commands:MSPACE (6.21.34)andPSPACE (6.24.54)

1. (option) When in **model space** (Model Space, with floating viewports) do one of the following to switch to **paper space** :

- Type either **pspace** or **ps** , then press Enter
- Double click outside a viewport.

2. (option) When in **paper space** do one of the following to switch to **model space** (Model Space, with floating viewports):

- Type either **mspace** or **ms** , then press Enter.
- Double click inside a viewport.

NOTE	**MSPACE** and **PSPACE** commands are available when working in a layout only.

Tutorial: Copying Layouts to Model Tab Click here (1.5.3) to watch

Figure 1.566

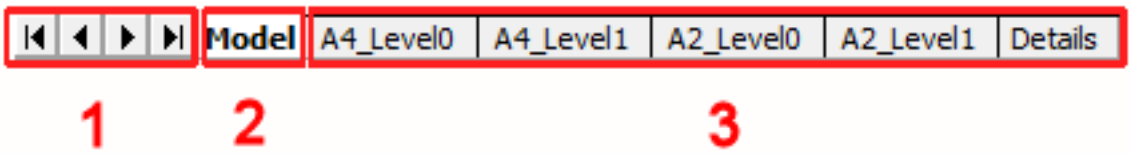
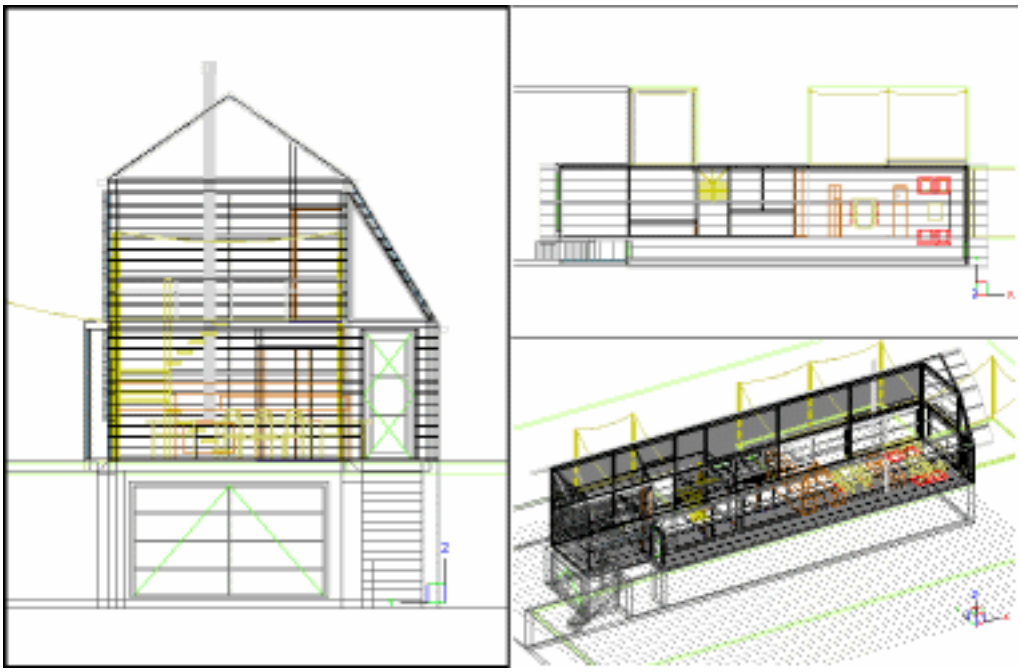


Figure 1.567



1.16.13 Model Space Viewports

Command: VPORTS (6.31.20) The display in model space (Model Space, with tiled viewports) can be divided into multiple viewports, each of which can contain a different view of the current drawing. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. When no command is active click in a viewport to make it the current viewport. You can print the current viewport only. The following settings can be defined differently for each viewport:

- grid display and snap
- coordinate system: WCS or UCS

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/09Multip
Model Space divided in 3 viewports

Creating viewports in model space

1. Do one of the following:

   - Click the **Viewports** tool button ( ) on the **Views** toolbar.
   - Choose **Viewports** in the **Views** menu.
   - Type **vports** in the command bar.

   You are prompted: Viewports: ? to list/Save/Restore/Delete/SIngle/Join/2/3/4/<3>: A prompt menu opens.
   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/09

2. (option) To create 2 viewports, do one of the following:

   - Choose **Create 2 viewports** in the prompt menu.
   - Type **2** in the command bar, then press Enter.

   You are prompted: Two viewports: Horizontal/<Vertical>:

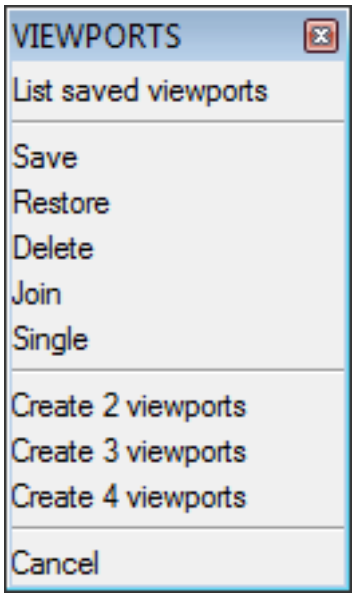
Option	Result
Horizontal	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
Vertical	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

3. (option) To create 3 viewports, do one of the following:

   - Choose **Create 3 viewports** in the prompt menu.
   - Type **3** in the command bar, then press Enter.

   You are prompted: Three viewports: Horizontal/Vertical/Above/Below/Left/<Right>:

Figure 1.568



Drawing in multiple viewports

1. Click in a viewport to make it current. The border of the current viewport highlights.
2. Draw the entities in the current viewport.
3. (option) Repeat step 1 to make a different viewport current.

NOTES

- The crosshairs show in the current viewport only.
- You can start a command in the current viewport, and finish the operation in a different viewport.

Joining adjacent viewports

1. Launch the **Viewports** command.
2. Select the **Join** command option. You are prompted: Select inside dominant viewport <Current>:
3. Click in the dominant viewport or press Enter to choose the current viewport. The border of the selected viewport highlights. You are prompted. Select inside viewport to join:
4. Click the viewport to join again to confirm. The two selected viewports are joined.

NOTE You can join viewports which share an edge of equal length only.

Tutorial: Working with Tiled Viewports in Model Space Click here (1.3.7) to watch

Figure 1.569

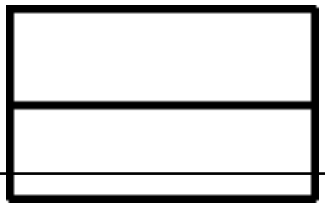


Figure 1.570

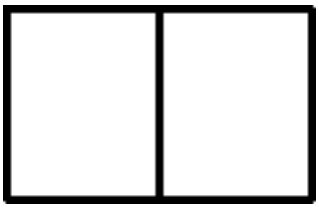


Figure 1.571

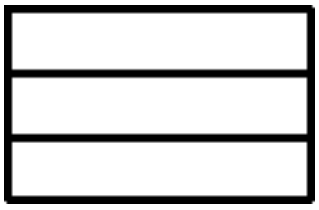


Figure 1.572



Figure 1.573

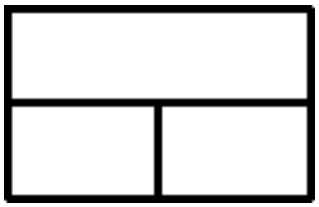


Figure 1.574

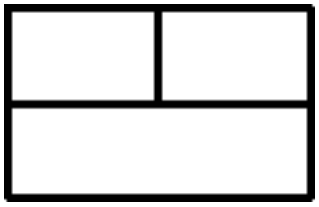


Figure 1.575

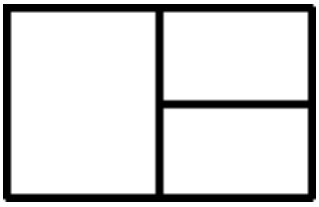


Figure 1.576

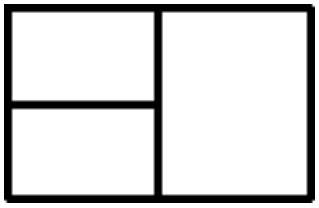
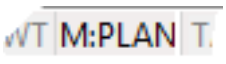
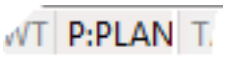


Figure 1.577



1.16.14 Paper space viewports

Commands: MVIEW (6.21.37), MVSETUP (6.21.39), ALIGNSPACE (6.4.7) and VPCLIP (6.31.17) The **Mview** command creates one or more viewports in layout tabs (short for "make viewports"). The **MvSetup** command creates multiple viewports; in model space the command creates a rectangle, representing a paper sheet with respect to a specified scale. The **AlignSpace** command adjusts viewport angle, zoom factor and pan position based on alignment points specified in model space and paper space. The **Vpclip** command clips viewports in layouts (short for "view port clipping"). In a layout you can create multiple viewports each of which displays a unique view of the entities created in model space. Each layout viewport functions as a window into your model space drawing. You can control the view, scale, and content of each layout viewport separately. A layout viewport is created as a separate entity that you can copy, delete, move, scale, and stretch as you would any other drawing entity. You can snap to the viewport borders using entity snap. When you are working in model space with floating viewports (see Toggle between model space and paper space (1.16.12)), click any layout viewport to make it the current viewport, and then add or modify model space entities in that viewport. Any changes you make in one layout viewport are immediately visible in the other viewports (if the other layout viewports are displaying that portion of the drawing). Zooming or panning in the current viewport affects only that viewport. Each viewport has its own layer visibility settings. You can also turn off the display of the content of a viewport. To preserve the scaling of a viewport you can lock the display. It is no longer possible to zoom or pan in a locked viewport. Apart from viewports, you can add print-related entities in a paper space layout that are not essential to the model itself, such as keynotes, annotations, title blocks, etc. Such entities are part of a specific paper space layout and do not appear in other layouts or in model space. When you are working in a layout, either **Model Space** or **Paper Space** is your current workspace. The **Workspace** field in the Status Bar (1.12.2) indicates which workspace is current: **M:<Layout Name>** indicates you are working in **Model Space** , while **P:<Layout Name>** means **Paper Space** is the current workspace.

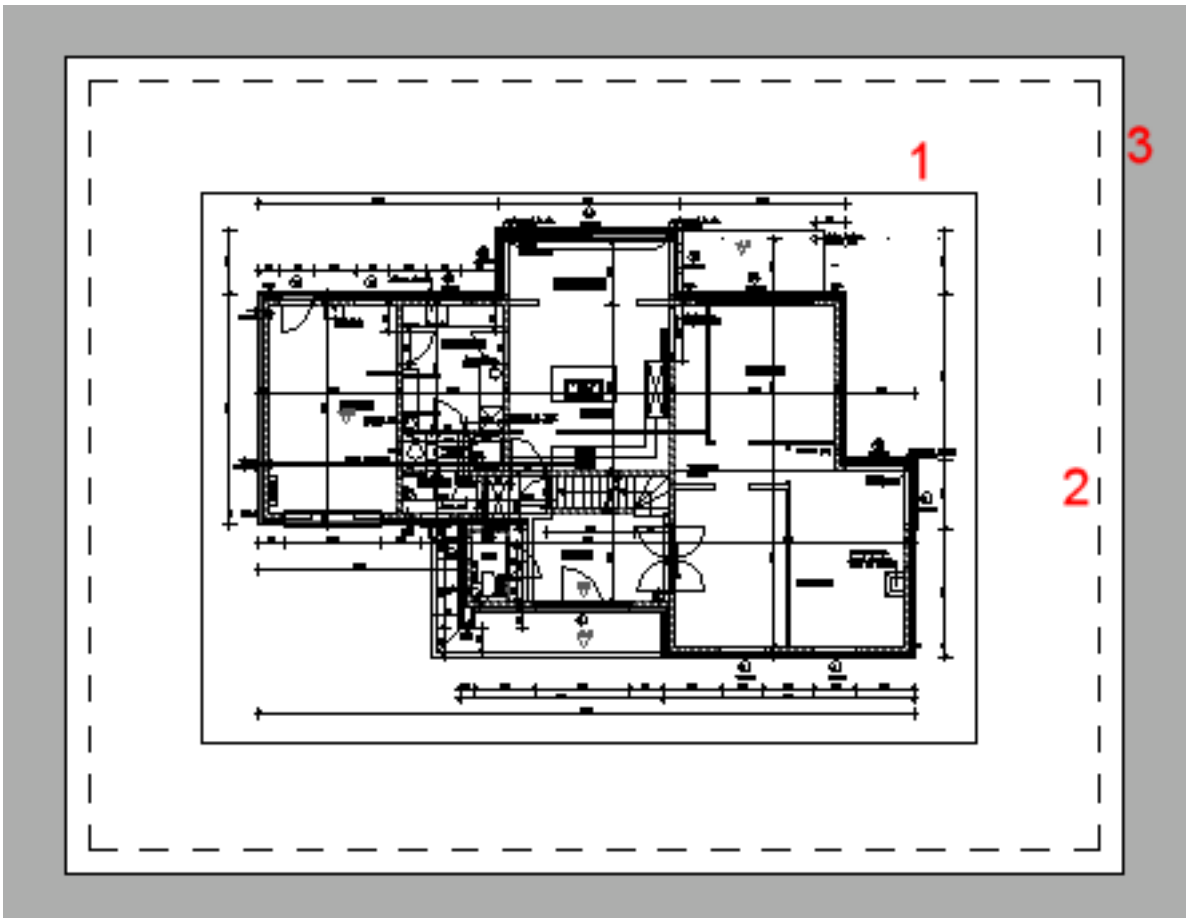
	Model Space, with floating viewports of layout PLAN
	Paper Space of layout PLAN

Displaying the paper sheet and the printable area

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART/04PaperMargins.png

1. Viewport
2. Printable area. The display of the printable area is controlled through the DISPPAPERMARGINS (1.13.11) system variable.
3. Paper sheet. The display of the paper sheet is controlled through the DISPPAPERBKG (1.13.11) system variable.

Figure 1.578



Creating viewports in a layout

Command: MVIEW (6.21.37)

1. Click the appropriate layout tab (1.16.12) at the bottom of the drawing window.
2. Do one of the following:
   - Click the **Paper Space Views** tool button ( ) on the **Views** toolbar.
   - Choose **Paper Space Views** in the **Views** menu.
   - Type **mview** at the command prompt, then press Enter.
3. (option) To add 1 viewport. do one of the following:
   - Click to specify the first corner of the viewport, then specify the opposite corner. A single viewport which fits in the specified rectangle is created.
   - Choose **Fit to view** in the prompt menu or type **F** , then press Enter. A single viewport which fits in the current drawing display window is created.
4. (option) To create 2 viewports, do one of the following:
   - Choose **Create 2 viewports** in the prompt menu.
   - Type **2** in the command bar, then press Enter.

   You are prompted: Two viewports: Horizontal/<Vertical>:

Option	Result
Horizontal	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
Vertical	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

   You are prompted: Fit to screen/<First corner of bounding rectangle>:

   - Click to specify the first corner of the bounding rectangle, then specify the opposite corner. Two viewports which fit in the specified bounding rectangle are created.
   - Choose **Fit to view** in the prompt menu or type **F** , then press Enter. Two viewports which fit in the current drawing display window are created.

Create 4 viewports (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

You are prompted: Fit to screen/<First corner of bounding rectangle>:

- Click to specify the first corner of the bounding rectangle, then specify the opposite corner. Four viewports which fit in the specified bounding rectangle are created.
- Choose **Fit to view** in the prompt menu or type **F** , then press Enter. Four viewports which fit in the current drawing display window are created.

7. (option) To create a non-rectangular (clipped) viewport using an existing polyline or circle, do one of the following:

   - Choose **Object** in the prompt menu.
   - Type **O** in the command bar, then press Enter.

   You are prompted: Select Object to clip viewport. Select a closed polyline or a circle in the layout.

8. (option) To create a non-rectangular (clipped) viewport, do one of the following:

   - Choose **Polygonal** in the prompt menu.
   - Type **P** in the command bar, then press Enter.

   You are prompted: Specify start point: Specify the vertices of the non-rectangular viewport. Right click or press Enter to create the viewport.

NOTES

- Make sure no other viewports display when choosing the **Fit to view** option because they will be hidden behind the newly created viewport.
- Each newly created viewport shows all entities on the layers which are currently visible in model space.
- Since a new viewport is created on the current layer it is recommended to use a dedicated layer for viewport entities. Make this layer current when you want to add a new viewport. If you don't want the viewport outlines to be printed, set the **Do Not Plot** property of the viewport layer.
- Only circles and closed polylines can be used to create a clipped viewport. If you want to create an elliptical viewport, use the Polyline Ellipse (1.17.17) variable to create a polyline representation of an ellipse.
- To select a clipped viewport do the following: Click the viewport border. Both the viewport and the clipping polygon are selected then, as indicated in the Properties Bar (1.12.4): the list button caption reads: **All (2)** . Press the list button and select **Viewport** to see the properties of the viewport.

Tutorial: Printing in BricsCAD: Paper Space Layouts Click here (1.8.3) to watch

Tutorial: Paper Space Layouts in BricsCAD Click here (1.8.2) to watch

Creating an array of layout viewports

Command: MVSETUP (6.21.39)

1. Type **mvsetup** in the command bar. You are prompted: Enter an option [Align/Create/Scale viewports/Undo]:
2. Do one of the following:
   - Type **C** in the command bar, then press Enter.
   - Choose **Create** in the context menu.

   You are prompted: Enter an option [Delete objects/Create viewports/Undo]<Create>:
3. Press Enter to accept the default option. You are prompted: Enter choice for layout options <0-3> [none(0)/single(1)/std.e of viewports(3)] <0>:
4. Do one of the following:
   - Type **3** in the command bar, then press Enter.
   - Choose **Array of Viewports** in the prompt menu.

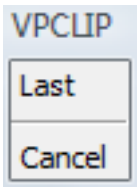
   You are prompted: Specify first corner of bounding area for viewport(s):
5. Pick a point in the layout. **You are prompted:** Other corner of rectangle:
6. Pick a second point in the layout. **You are prompted:** Enter number of viewports in X direction <1>:
7. Specify the number columns in the array. **You are prompted:** Enter number of viewports in Y direction <1>:
8. Specify the number of rows in the array. **You are prompted:** Specify distance between viewports in X direction <0.000>:
9. Press Enter, to create contiguous viewport columns. **You are prompted:** Specify distance between viewports in Y direction <horizontal distance>:
10. Do one of the following:
    - Press Enter to set the vertical spacing equal to the horizontal spacing.
    - Type **0** (zero) to create contiguous viewport rows.
    - Specify the vertical spacing between the viewports.

    The viewports are created.
11. Press Enter to conclude the command.

To clip a viewport

Command: VPCLIP (6.31.17)

1. Type **vpclip** in the command bar, then press Enter. **You are prompted:** Select viewport to clip:
2. Select the viewport. **You are prompted:** Polygonal <Select clipping object>: A prompt menu displays.



3. Do one of the following:
   - Select the clipping object.
   - Choose **Polygonal** in the prompt menu or type **P** , then press Enter. **You are prompted** to draw a polyline (1.17.12).

   The viewport is clipped.

NOTE

- Only closed polylines and circles are accepted as clipping boundaries.
- If you want elliptical viewports, use the polyline representation (1.17.17) to create the ellipses.

To remove the clipping boundary of a viewport

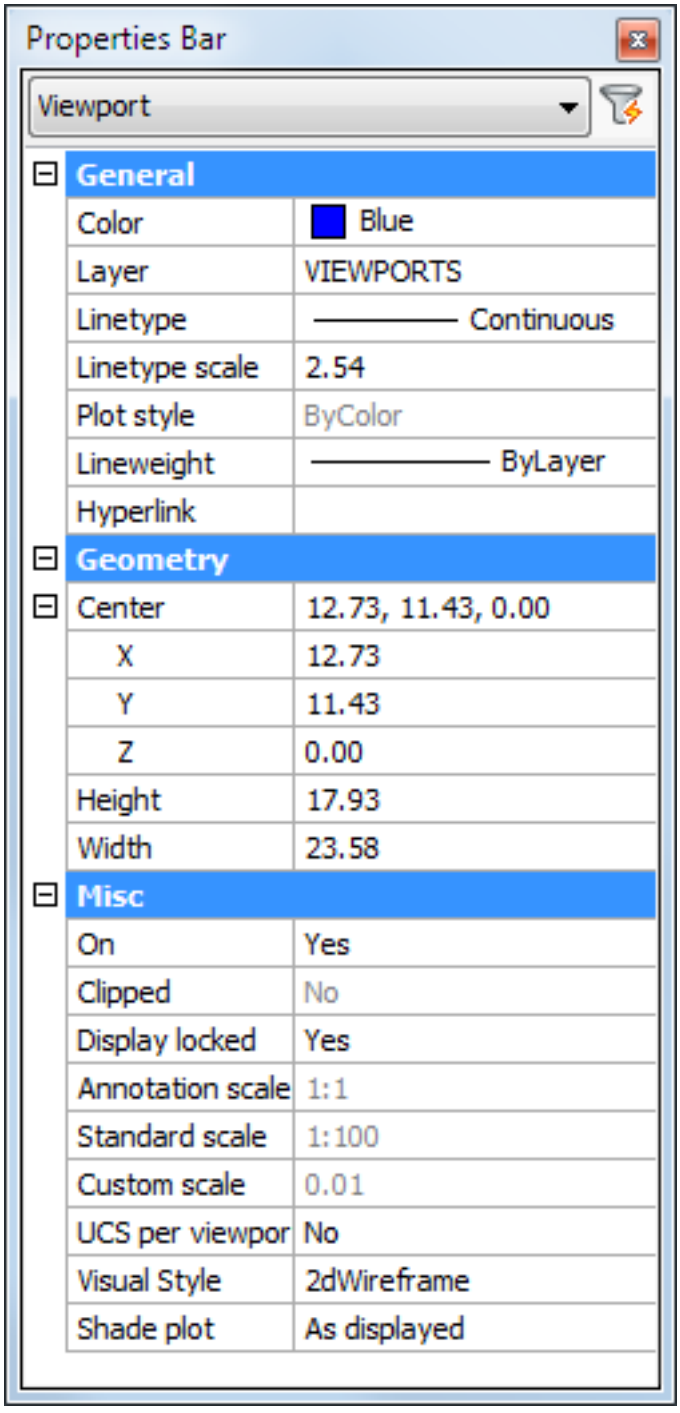
Command:VPCLIP (6.31.17)

1. Type **vpclip** in the command bar, then press Enter. **You are prompted:** Select viewport to clip:
2. Select a clipped viewport. **You are prompted:** Polygonal/Delete<Select clipping entity>:
3. Do one of the following:
   - Type **D** , then press Enter.
   - Choose **Delete** in the prompt menu.

Defining viewport properties

1. Select the viewport. The viewport properties display in the **Properties Bar**. (\vref {fig:C:/Dev/Task-2448/images/staticbri
2. Click the settings field of a property to modify.

Figure 1.579


Properties Bar
Viewport
General
Color
Blue
Layer
VIEWPORTS
Linetype
Continuous
Linetype scale
2.54
Plot style
ByColor
Lineweight
ByLayer
Hyperlink
Geometry
Center
12.73, 11.43, 0.00
X
12.73
Y
11.43
Z
0.00
Height
17.93
Width
23.58
Misc
On
Yes
Clipped
No
Display locked
Yes
Annotation scale
1:1
Standard scale
1:100
Custom scale
0.01
UCS per viewpor
No
Visual Style
2dWireframe
Shade plot
As displayed

Properties

Name	Description
Center	XYZ coordinates of the center point of the viewport. To specify the center of the viewport graphically, drag the center handle of the viewport.
Height (*)	Height of the viewport in **drawing units** . To specify the height of the viewport graphically, drag the top or bottom handle of the viewport.
Width (*)	Width of the viewport in **drawing units** . To specify the width of the viewport graphically, drag the left or right handle of the viewport.
On	Controls the display of the content of the viewport.
Clipped	Allows to create non-rectangular viewports.
Display locked	Locks the scaling of the viewport content to preserve the scaling factor.
Scale	The SCALELISTEDIT (6.27.7) command allows to edit the available scales in the current drawing. Annotation scale Sets the annotation scale for the viewport. If the ANNOALLVISIBLE (1.13.11) system variable is OFF, annotative entities which do not supoort the annotation scale do not display. Standard scale The **Standard scale** field automatically matches the **Annotation scale** field. Although you can set the **Standard scale** field different from the **Annotation scale** field manually, it is not recommended if annotative entities (1.21.27) exist in the drawing. The **Standard scale** field reads 'Custom' if the value in the **Custom scale** field is not available in the drawings scale list. Custom scale Displays the viewport scale factor. Allows to set the viewport scale to a scale which is not available in the drawings scale list.
Standard scale (*)	Lets you choose a standard scaling factor.
Custom scale (*)	Lets you define the scaling factor in a decimal format. Displays the current scaling factor.
UCS per viewport	If Yes, lets you define a UCS for this viewport.
Shade Plot	Defines how the viewport will be plotted. The options are: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04

(*) You must choose the width and height of a paperspace viewport with respect to the viewport scale (1.16.14).

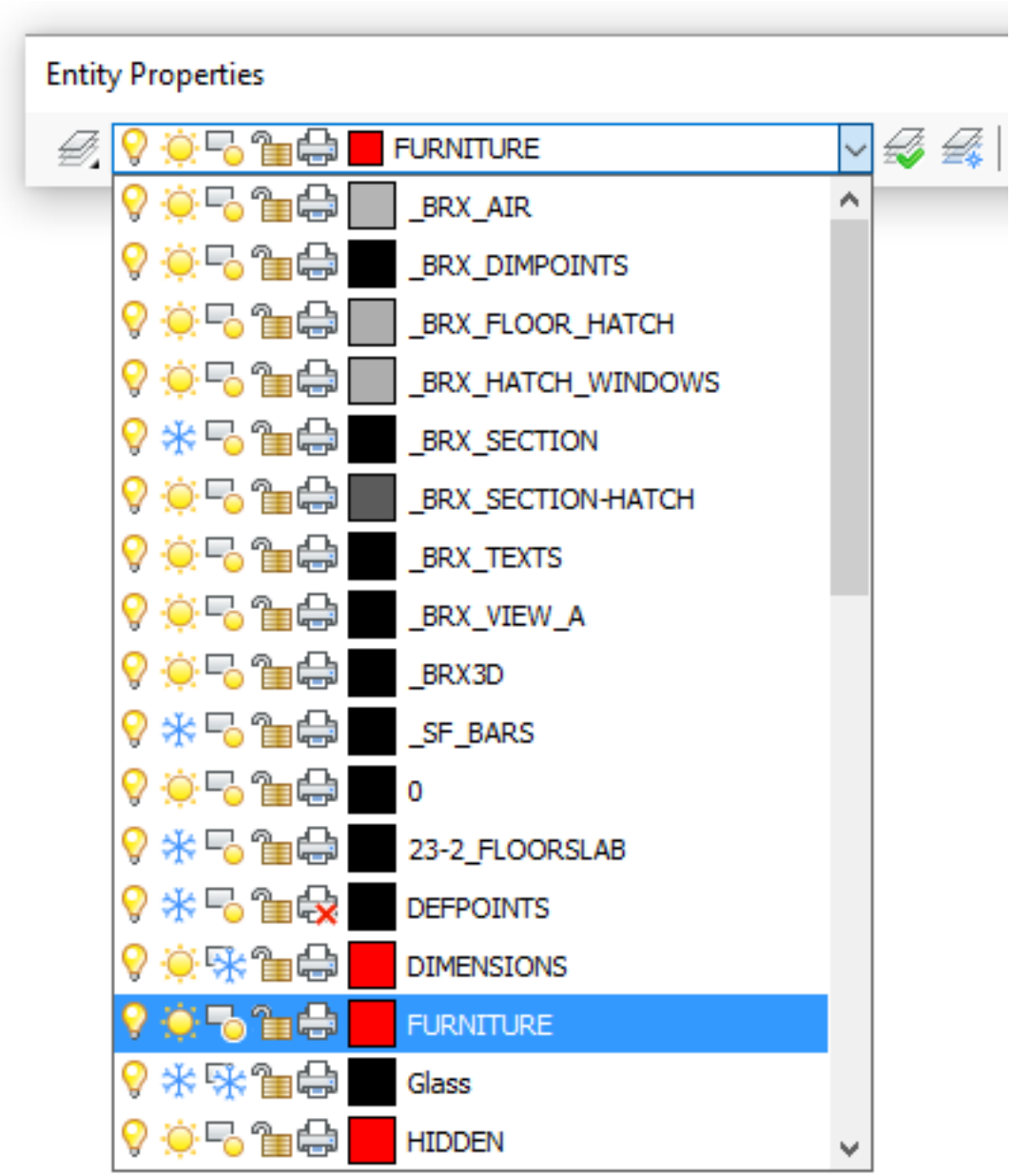
Navigating in a paper space layout

View manipulation commands, such as zoom, pan and view manipulation using the mouse, act slightly different in a paper space layout compared to model space (with tiled viewports). If you are working in paper space (no viewport selected), view manipulation commands act on the complete paper space layout. If you are working in a viewport (model space with floating viewports), view manipulation commands act on the active viewport only. Except if the display of the current viewport is locked, then the view manipulation commands act on the complete paper space layout.

NOTE	It is not possible to rotate the display of a paper space layout. View Rotate (1.16.8) commands are available in viewports of which the display is not locked only.

Tutorial: How to navigate quickly around your drawing Click here (1.2.2) to watch

Figure 1.580



Setting the layer visibility in a viewport

Using the Entity Properties toolbar

1. Do one of the following:

   - If **Paper Space** is the current workspace: double click inside the viewport to make it the current viewport.
   - If **Model Space with floating viewports** is the current workspace: click inside the viewport to make it the current viewport.

2. Select the layer in the layer list of the **Entity Properties** toolbar. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he

3. Do one of the following:

   - Click the **Thawed** icon ( ) of the layer(s) you want to freeze in the current viewport.
   - Click the **Frozen** icon ( ) of the layer(s) you want to thaw in the current viewport.

4. (option) Repeat the previous steps to set the layer visibility in another viewport.

Using the Drawing Explorer / Layers dialog

1. Do one of the following:
   - If **Paper Space** is the current workspace: double click inside the viewport to make it the current viewport.
   - If **Model Space with floating viewports** is the current workspace: click inside the viewport to make it the current viewport.
2. Do one of the following.
   - Click the **Layers...** tool button on the **Settings** toolbar.
   - Choose **Layers...** in the **Settings** menu.
   - Type **layer** in the command bar, then press Enter.

   The **Drawing Explorer - Layers** window opens.
3. In the **VP Freeze** column:
   - Click the **Thawed** icon ( ) of the layer(s) you want to freeze.
   - Click the **Frozen** icon ( ) of the layer(s) you want to thaw.
4. Close the **Drawing Explorer - Layers** window.
5. (option) Repeat the previous steps to set the layer visibility in another viewport.

NOTE	Use the **Viewports / Vpfreeze** option of the LAYOFF (6.20.12) and LAYFRZ (6.20.8) commands to freeze layers in a viewport by clicking entities.

Setting the scale of a viewport

1. Make sure the Plot Scale (1.25.1) property of the layout is set correctly.
2. Switch to **paperspace** . (See Toggle between model space and paper space (1.16.12))
3. Click the viewport border. The viewport properties display in the BricsCAD Properties Bar (1.12.4).
4. Do one of the following:
   - Select the **Annotation scale** field, then click the **Browse** button ( ) and select a scale in the scale list. The **Standard scale** and **Custom scale** fields are updated automatically.
   - Select the **Standard scale** field, then click the **Browse** button ( ) and select a scale in the scale list. The **Custom scale** field is updated automatically.
   - Select the **Custom scale** field and type the scale factor. If the scale factor is not available in the drawings scale list, the **Standard scale** field reads 'Custom'.
5. (option) If necessary, adjust the size of the viewport to the new scale.
6. (option) Set the **Display Locked** property to **Yes** . The scale fields are greyed out and cannot be edited anymore.

NOTE	In a **locked viewport** it is impossible to: • zoom, pan or rotate the view • modify the scale

Setting the scale of a selection of viewports

Command:MVSETUP (6.21.39)

1. Make sure thePlot Scale (1.25.1)property of the layout is set correctly.
2. Type **mvsetup** in the command bar. You are prompted: Enter an option [Align/Create/Scale viewports/Undo]:

3. Do one of the following:
   - Type **S** in the command bar, then press Enter.
   - Choose **Scale viewports** in the prompt menu.

   You are prompted: Select entities
4. Select the viewports.
5. Press Enter or right click to stop selecting viewports. **You are prompted:** Set the scaling mode of viewports: Interactively/<Uniform>:
6. Do one of the following:
   - Press Enter to set the scale of the selected viewports uniformly. You are prompted: Enter the number of paper space units <1.0>:
     - Press Enter to accept the number of paper space units. You are prompted: Enter the number of model space units <1.0>:
     - Type the desired scaling factor: e.g. 50 to apply a 1/50 scale.
     - Press Enter to apply the scale to all selected viewports.
     - Press Enter to conclude the command.
   - Choose **Interactively** in the prompt menu or type **I** , then press Enter. The first viewport in the selection set highlights. You are prompted: Enter the number of paper space units <1.0>:
     - Press Enter to accept the number of paper space units. You are prompted: Enter the number of model space units <1.0>:
     - Type the desired scaling factor: e.g. 50 to apply a 1/50 scale.
     - Press Enter to apply the scale to the viewport. The next viewport highlights. You are prompted: Enter the number of paper space units <1.0>:
     - Repeat the previous steps for each of the selected viewports.
7. Press Enter to conclude the command.

To rotate the display in a viewport

The display of a layout viewport can be rotated only if the display of the viewport is not locked. If the VPROTATE-ASSOC (1.13.11) system variable is ON, the ROTATE (6.26.34) command rotates the content of a viewport over a specified angle. The **Align > Rotate** option of theMVSETUP (6.21.39)command rotates the content of a viewport relative to the X-axis of the world coordinate system (WCS).

Rotating the viewport display

1. Select the viewport, then right click and choose **2D Rotate** in the context menu. **You are prompted:** Rotation point.
2. Use the **Center** () entity snap to select the center point of the viewport as the rotation point. **You are prompted:** Copy/Base angle/<Rotation angle> <0>:
3. Type the rotation angle, then press Enter.
4. The content of the viewport is rotated over the specified angle.

Defining the rotation angle of the viewport display

1. Type **mvsetup** in the command bar. **You are prompted:** Enter an option [Align/Create/Scale viewports/Undo]:
2. Do on of the following:
   - Type **A** in the command bar, then press Enter.
   - Choose **Align** in the prompt menu. **You are prompted:** Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]:
3. Do one of the following:
   - Type **R** in the command bar, then press Enter.

- Choose **Rotate view** in the prompt menu. **You are prompted:** Specify basepoint in the viewport to be rotate:

4. Specify the rotation point in the viewport. **You are prompted:** Specify rotation angle:
5. Type the rotation angle in the command bar, then press Enter. The viewport display is rotated over the specified angle with respect to the X-axis of the WCS.

NOTE	Type 0 (zero) in the final step of the above procedure to undo the rotation of the viewport display.

To align viewports

Command:MVSETUP (6.21.39)

1. Type **mvsetup** in the command bar. **You are prompted:** Enter an option [Align/Create/Scale viewports/Undo]:
2. Do on of the following:
   - Type **A** in the command bar, then press Enter.
   - Choose **Align** in the prompt menu. **You are prompted:** Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]:
3. Do one of the following:
   - To align two viewports horizontally:
     - Choose **Horizontal alignment** in the context menu or type **H** in the command bar, then press Enter. **You are prompted:** Specify basepoint:
     - Pick a point in the reference viewport. **You are prompted:** Specify point in viewport to be panned:
     - Pick a point in the viewport to be aligned. The display in the viewport is moved vertically to align the two points horizontally.
   - To align two viewports vertically:
     - Choose **Vertical alignment** in the context menu or type **V** in the command bar, then press Enter. **You are prompted:** Specify basepoint:
     - Pick a point in the reference viewport. **You are prompted:** Specify point in viewport to be panned:
     - Pick a point in the viewport to be aligned. The display in the viewport is moved horizontally to align the two points vertically.

Tutorial: Printing in BricsCAD: Viewports Click here (1.8.4) to watch

To align model space and paper space points

Command: ALIGNSPACE (6.4.7)

Aligning 1 point

1. Type **alignspace** in the command bar, then press Enter. **You are prompted:** First alignment point in Model space:
2. Click inside a layout viewport, then pick a point. **You are prompted:** Second alignment point in Model space or <Return> for none:
3. Right click or press Enter. **You are prompted:** Alignment point in Paper space:
4. Pick a point in the layout. **You are prompted:** Activate the viewport which will be used for alignment.
5. Click inside the viewport you want to align.
6. Right click or press Enter to confirm. The display in the selected viewport is moved to make the two points coincident.

Figure 1.581

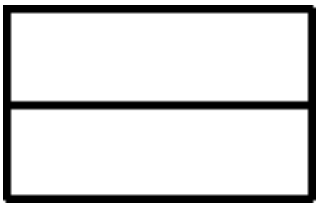
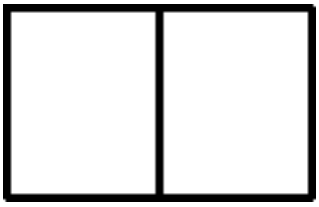


Figure 1.582



Aligning 2 points

1. Type **alignspace** in the command bar, then press Enter. **You are prompted:** First alignment point in Model space:
2. Click inside a layout viewport, then pick a point. **You are prompted:** Second alignment point in Model space or <Return> for none:
3. Pick a second point in the layout viewport. **You are prompted:** First alignment point in Paper space:
4. Pick a point in the layout. **You are prompted:** Second alignment point in Paper space:
5. Pick a second point in the layout. **You are prompted:** Activate the viewport which will be used for alignment.
6. Click inside the viewport you want to align.
7. Right click or press Enter to confirm. The display in the selected viewport is moved, rotated and/or scaled to make the points two by two coincident.

Figure 1.583

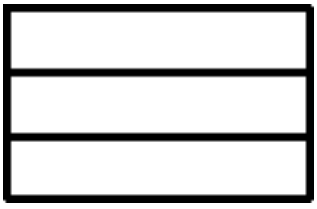


Figure 1.584



Figure 1.585

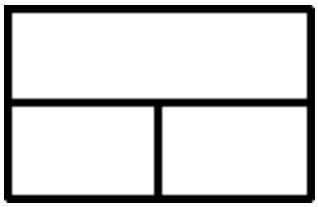


Figure 1.586

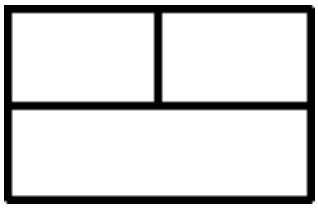


Figure 1.587

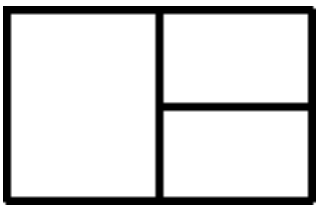


Figure 1.588

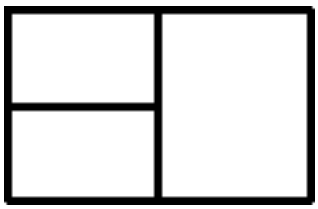


Figure 1.589

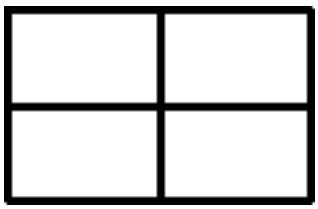
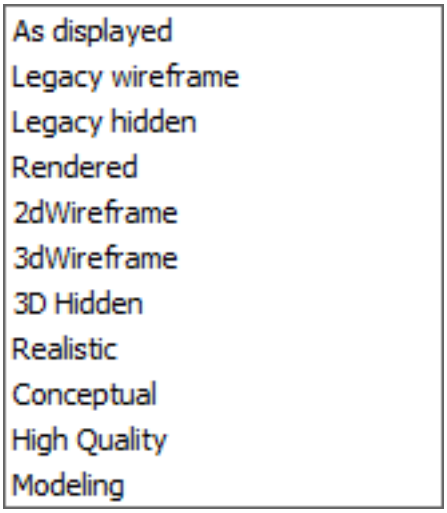


Figure 1.590


As displayed
Legacy wireframe
Legacy hidden
Rendered
2dWireframe
3dWireframe
3D Hidden
Realistic
Conceptual
High Quality
Modeling

1.16.15 Layouts

Commands: LAYOUT (6.20.14), LAYOUTMANAGER (6.20.15) In BricsCAD, you can create up to 255 layouts in a single drawing. Each layout represents a sheet of paper. For each layout you can specify the print area, print scale, lineweight scale, pen mappings, and add viewports, dimensions, a title block, and other geometry specific to the layout. The entities you add to a layout in paper space do not appear in model space. Each layout requires at least one layout viewport. If the CREATEVIEWPORTS (1.13.11) system variable is ON a viewport is added automatically in a new layout. If this variable is OFF, newly created viewports are empty. To add new layouts you can either define them yourself or your can import layouts from another drawing: template file (.dwt), drawing file (.dwg) or drawing interchange file (.dxf). Viewports can display all or part of the drawing's model space entities. The CACHELAYOUT (1.13.11) system variable controls whether the content of layouts is cached. The extra memory required for caching is made up for by improvements to the memory requirements for displaying entities, the net result being that the memory needed to display drawings has been reduced. As a result of this switching between layouts is up to 10 times faster now. The Ctrl-PgUp or Ctrl-PgDn keyboard shortcuts allow to open the next or previous layout.

Creating a new layout

1. Do one of the following:

   - Click the **New Layout** tool button () on the **Layout** or the **Insert** toolbar.
   - Choose **New Layout** in the **Insert** menu.
   - Type **layout** in the command bar, press Enter, then type **N** and press Enter.

   You are prompted: Enter a layout name. <Layout1>:

2. Do one of the following:

   - Type a unique name for the layout, then press Enter. The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign ($), hyphen (-), and underscore (_), or any combination.
   - Right click or press Enter to accept the default name (e.g. Layout1).

   The new layout tab is added.

3. Create at least one viewport (see Creating viewports in a layout (1.16.14)). If the CREATEVIEWPORTS (1.13.11) system variable is ON a viewport is added automatically in a new layout.

Copying layouts

1. Right click the layout tab you want to copy. A context menu displays.
2. Choose **Copy** in the context menu. **You are prompted:** Enter new layout name:
3. Type a new unique name in the command bar, then press Enter. A copy of the selected layout is added.

Arranging the layout tabs

Drag the layout tab to the desired position **Or**

1. Right click the layout tab you want to move. A context menu displays.
2. Do one of the following:

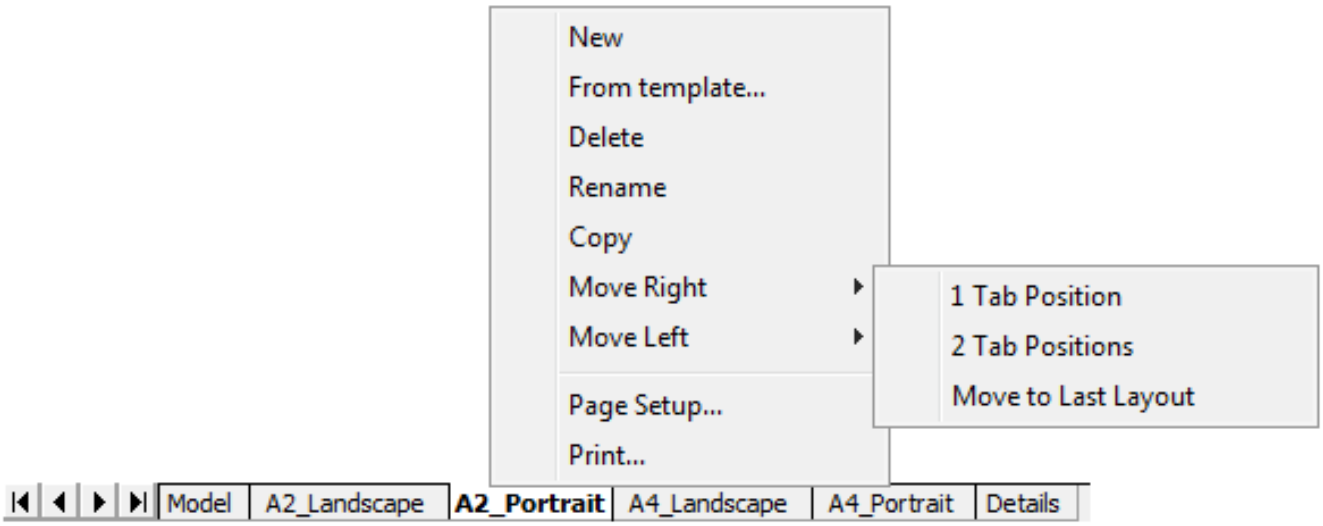
   - Choose **Move Right** .
   - Choose **Move Left** .

3. To move the selected layout tab do one of the following:

   - Select the number of tab positions.
   - Choose **Move to Last Layout** or **Move to First Layout** .

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/11

Figure 1.591



Importing layouts

1. Right click the **Model** tab or one of the layout tabs. A context menu displays.
2. Choose **From Template...** in the context menu. The **Select Template From File** window opens.
3. Browse to the drawing that you want to import layouts from.
4. Click the **Open** button on the **Select Template From File** window. The **Insert Layout(s)** window opens.
5. Select the layout(s) you want to import. Press and hold the Ctrl key to select multiple layouts.
6. Click the OK button on the **Insert Layout(s)** window. The selected layouts are imported.

Renaming layouts

1. Double click the layout tab you want to rename. The **Rename Layout** dialog opens.
2. Type a new unique name in the **Name** field of the **Rename Layout** dialog.
3. Click the **OK** button on the **Rename Layout** dialog.

Deleting a layout

1. Right click the layout tab you want to move. A context menu displays.
2. Choose **Delete** in the context menu. The selected layout is deleted.

Deleting multiple layouts

1. Do one of the following:
2. Choose **Drawing Explorer > Page Setups...** in the **Tools** menu.
3. Choose **Page Setup...** in the **File** menu. The **Drawing Explorer - Page Setups** dialog displays.
4. Select the layouts you want to delete. To select a series of layouts: select the first layout, then press and hold the Shift key and select the last layout. Press and hold the Ctrl-key to add/remove layouts.
5. Click the **Delete** tool button ( ) in the toolbar on the **Drawing Explorer - Page Setups**toolbar. An alert box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/A
6. Press the **Yes** button on the **Delete Layouts** alert box.

Tutorial: Copying Layouts to Model Tab Click here (1.5.3) to watch

Figure 1.592

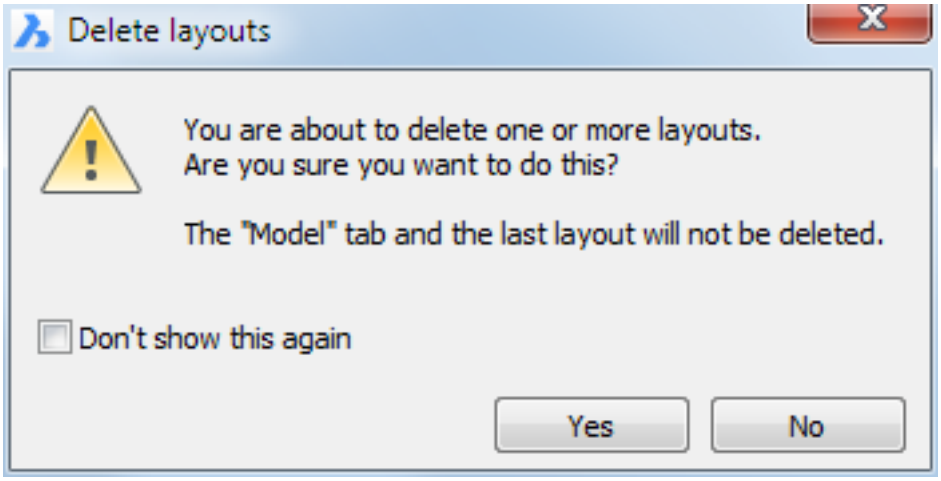
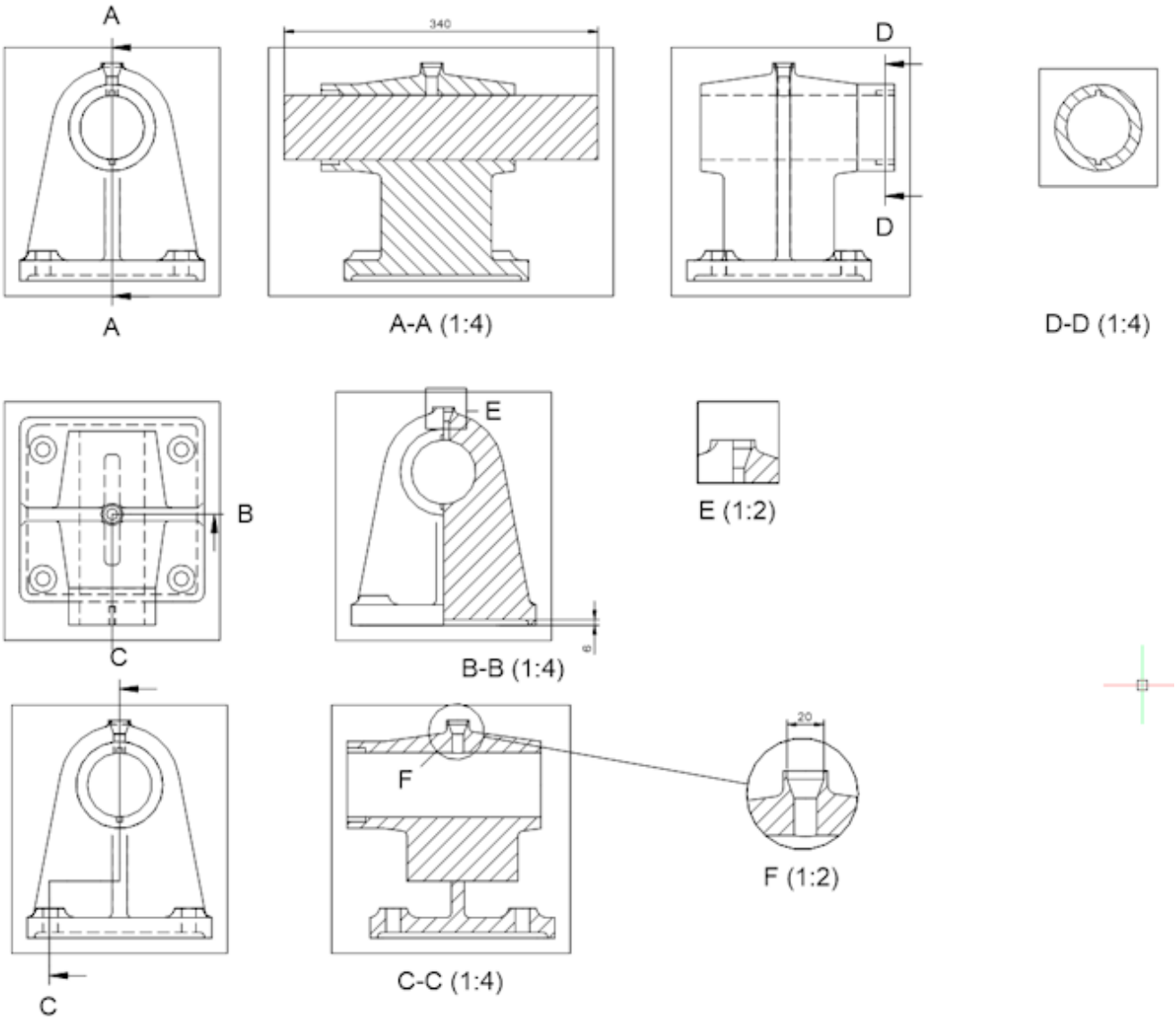

Delete layouts
You are about to delete one or more layouts.
Are you sure you want to do this?
The "Model" tab and the last layout will not be deleted.
Don't show this again
Yes
No

Figure 1.593



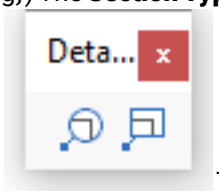
1.16.16 Generated Drawing Views

Commands: VIEWBASE (6.31.3), VIEWSECTION (6.31.10), VIEWSECTIONSTYLE (6.31.11), VIEWPROJ (6.31.8), VIEWDETAIL (6.31.4), VIEWDETAILSTYLE (6.31.5), VIEWUPDATE (6.31.12), VIEWEXPORT (6.31.7) and VIEWEDIT (6.31.6) The Generated Drawing Views functionality allows to automatically generate associative orthographic and standard isometric views of a 3D solid model. All drawing views are placed in a paper space layout (1.16.15), they are not visible in model space (1.16.12). Analytical hidden line removal (HLR) procedures are used to create the drawing views using standard 2D entities, mostly lines and arcs. Much like the result of the FlatShot (6.14.9) command. Drawing views created in V14.1 will be automatically converted into HLR geometry by the ViewUpdate (6.31.12) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntitie
The **ViewBase** command generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout (1.16.15). The **ViewSection** command creates a cross section view based on a standard drawing view generated by the **ViewBase** command in a paper space layout (1.16.15). The **ViewSectionStyle** command allows to modify the appearance of the section views created by the **ViewSection** command. The **ViewProj** command generates additional projected views from an existing generated drawing view. The **ViewDetail** command creates a detail view of a portion of a standard generated drawing, at a larger scale. The **ViewdetailStyle** command allows to modify the appearance of the detail views created by the **ViewDetail** command. The **ViewUpdate** command Updates a selection of drawing views obtained by **ViewBase** and **ViewSection** when VIEWUPDATEAUTO (1.13.11) = 0. The **ViewExport** command exports the content of drawing views obtained by **ViewBase** and **ViewSection** to the Model Space (1.16.12) of the drawing. This command can be used in Paper Space (1.16.12) only. The **ViewEdit** command allows changing the scale and the hidden line visibility of drawing views.

NOTE	Drawing views generated in V14.2 or higher will show correctly in BricsCAD V13. The most recent V13.2 version is recommended. Drawing views generated in V14.2 or higher do not display in V12 and older BricsCAD versions.

Commands and Toolbars

Tools to generate drawing views of your 3D solid model are available on: The **Drawing Views** toolbar, which is a fly-out of the **View** toolbar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterfa 4GeneratedViews.png}) The **Section Types** and **Detail Section Types** toolbars: (\vref {fig:C:/Dev/Task-2448/images/staticbrics



6SectionTypes.png}) The **View | Drawing Views** menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco The **Drawing Views** panel of the **Annotate** Ribbon tab. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/

Figure 1.594

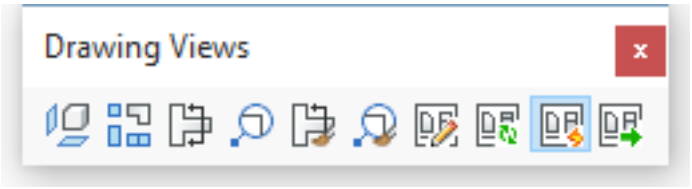


Figure 1.595

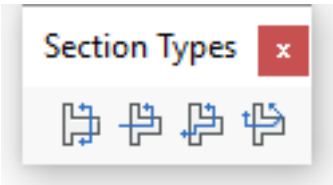


Figure 1.596

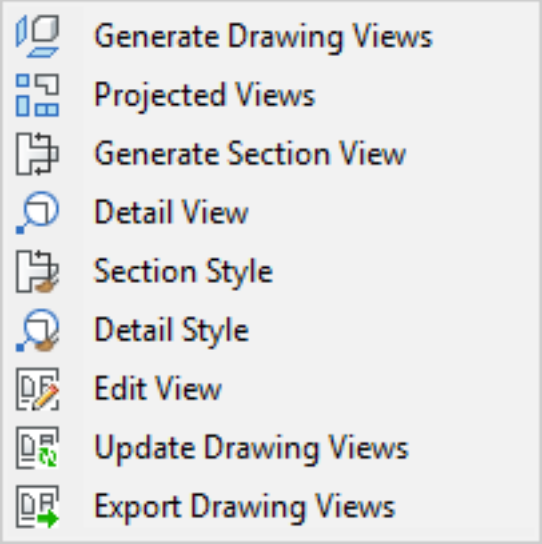
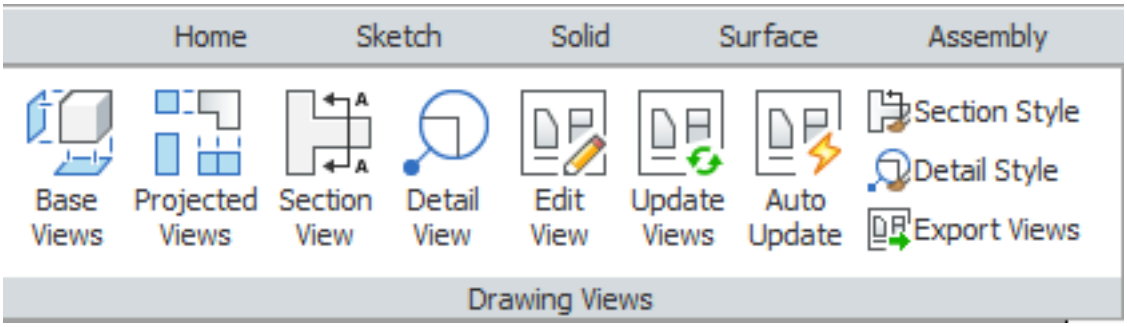


Figure 1.597



Standard Drawing Views

Standard views include multiview orthographic and isometric projections. In a technical drawing, a multiview orthographic projection is an illustration technique in which up to six images of an object are generated, with each projection plane parallel to one of the coordinate axes of the object. Isometric projection is a method to represent three-dimensional objects in two dimensions in technical and engineering drawings. It is an axonometric projection in which the three coordinate axes appear equally foreshortened and the angles between any two of them are 120 degrees. To generate the standard views of your 3D solid model, run the ViewBase (6.31.3) command from Model Space (1.16.12). You can choose between first angle (European) projection or third angle (American) projection. Orthographic projection (or orthogonal projection) is a means of representing a three-dimensional object in two dimensions. It is a parallel projection, where all the projection lines are orthogonal to the projection plane, resulting in every plane of the scene appearing in affine transformation on the viewing surface. It is further divided into multiview orthographic projections and axonometric projections. Two projection planes, one horizontal (H.P.) and one vertical (V.P.) divide the space in four quarters.

- In the first angle projection system the model is placed in the first quarter space (first angle).
- In the third angle projection system the model is placed in the third quarter (third angle).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/GdProje
With multiview orthographic projections, up to six pictures of a 3D model are produced, with each projection plane parallel to one of the coordinate axes of the model. The views are positioned relative to each other according to either of two schemes: first-angle or third-angle projection. In each, the appearances of views may be thought of as being projected onto planes that form a 6-sided box around the model. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/GdProjectionMethods.png})
The projection type is set by the **Projection Type** option of the ViewBase (6.31.3) command or by editing the PROJECTIONTYPE (1.13.11) system variable:

- 0 = First angle projection
- 1 = Third angle projection

The value of the PROJECTIONTYPE (1.13.11) system variable is saved in the drawing. Generated drawing view lines are created on default layers:

- BM_Isometric_Hidden;
- BM_Isometric_Visible;
- BM_Ortho_Hidden;
- BM_Ortho_Visible.

The display of the **BM_Isometric_Hidden** layer is turned off by default. Additional projected views can be added using the ViewProj (6.31.8) command. A projected view inherits the scale, display settings and alignment from the parent view.

Figure 1.598

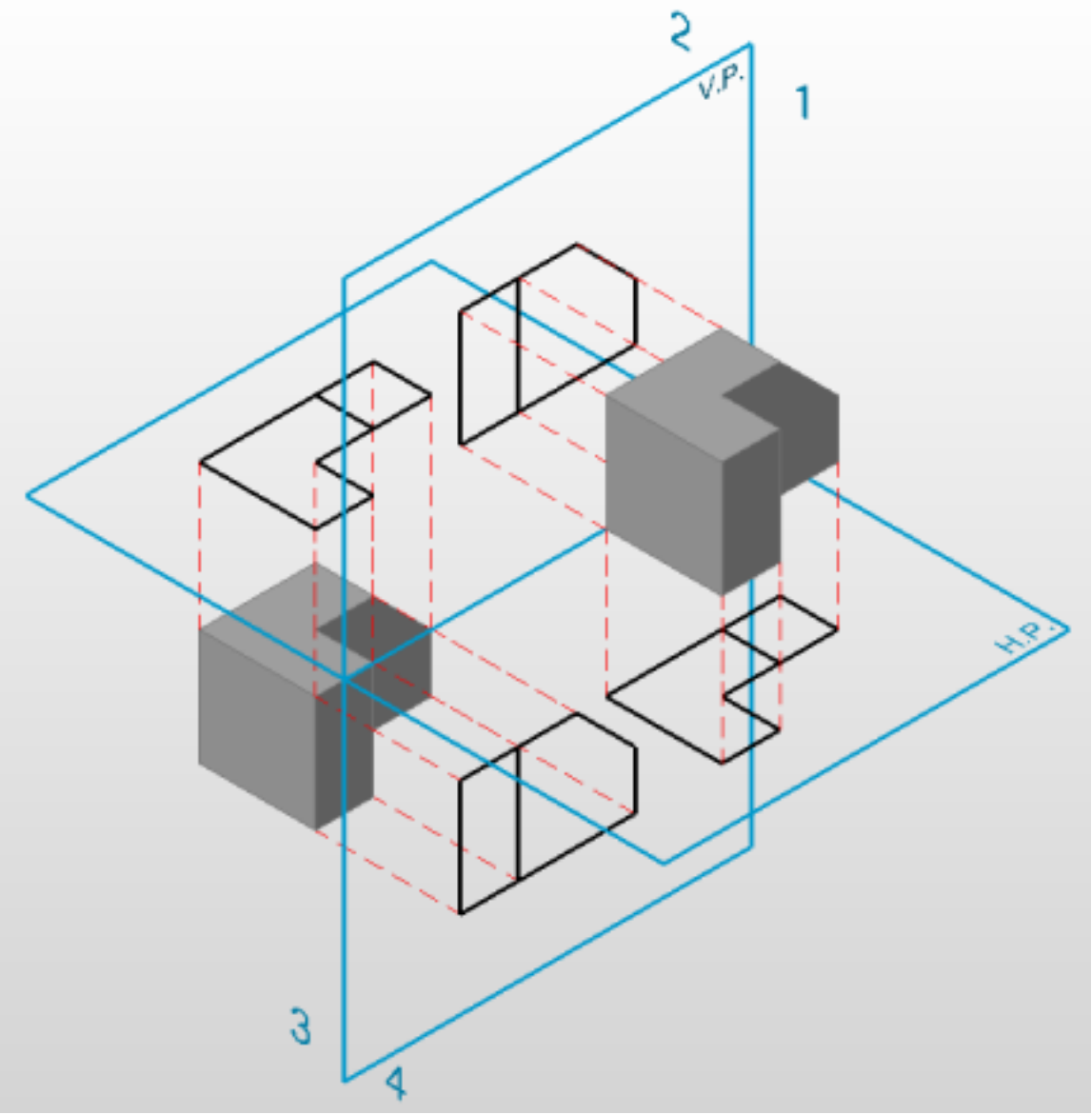
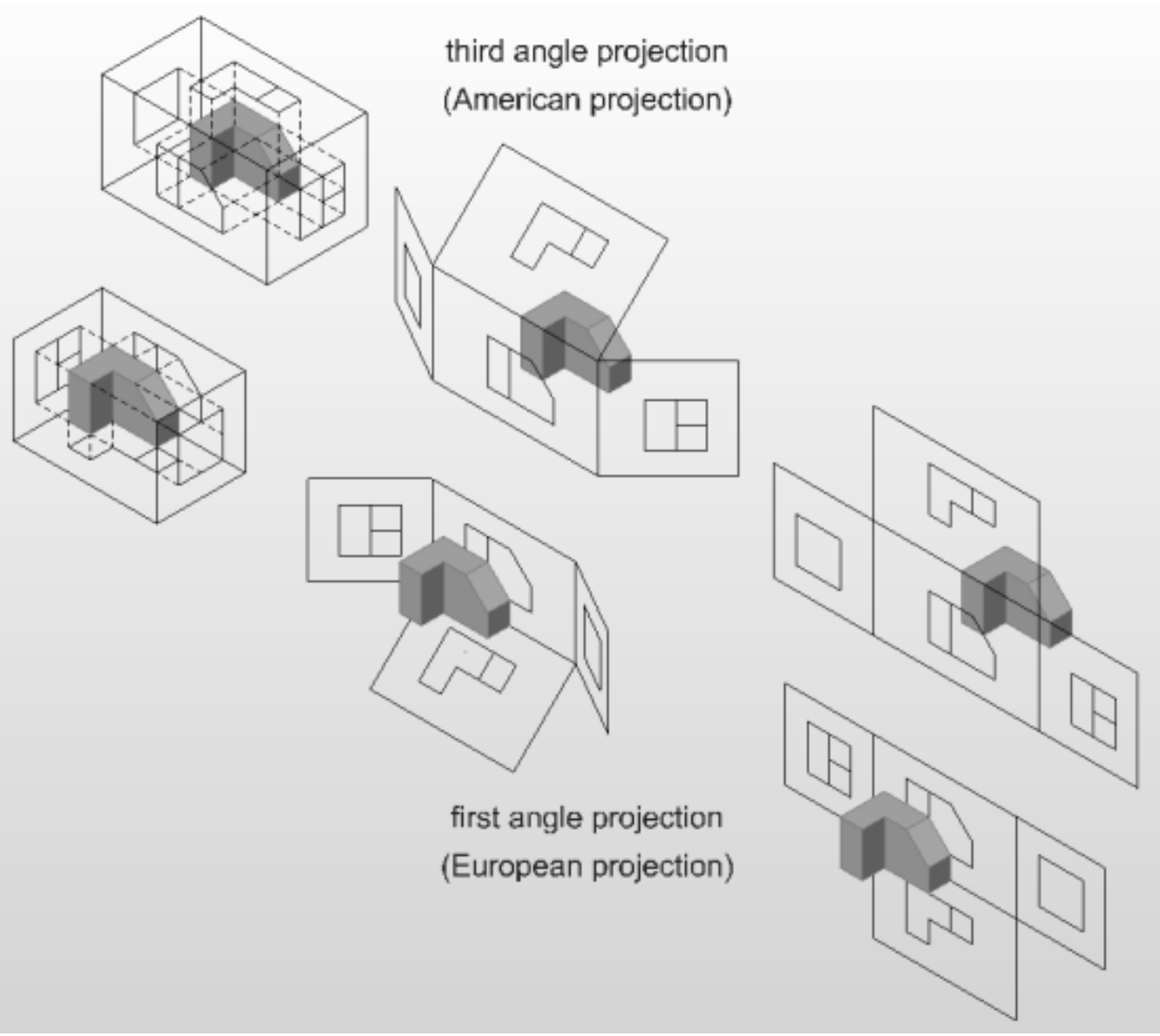


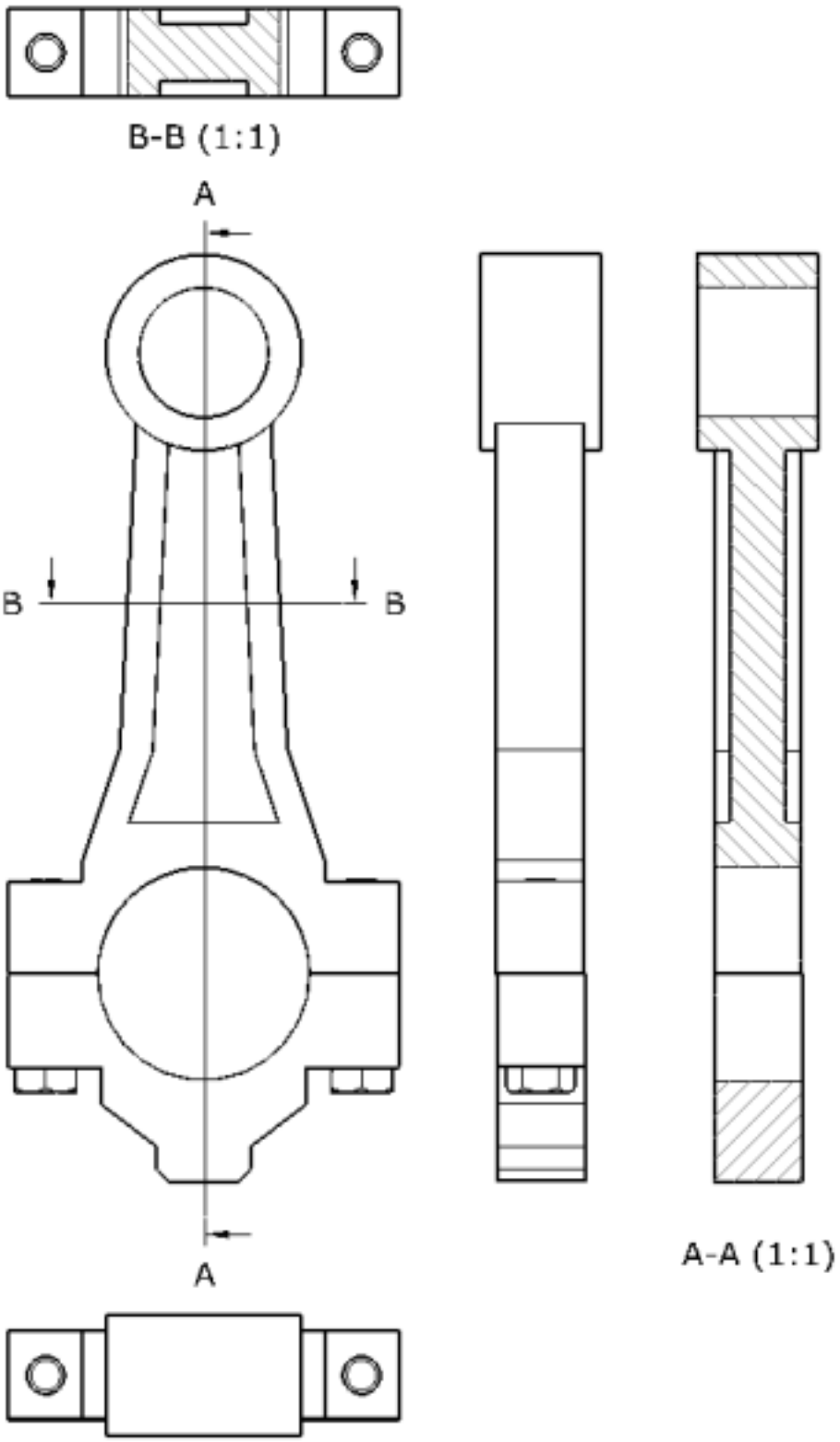
Figure 1.599


third angle projection
(American projection)
first angle projection
(European projection)

Generating Cross Section Views

A cross section is the intersection of a 3D solid model with a section plane. In technical drawings, the internal parts of the 3D model are hatched in cross sections. To generate section views of a 3D solid model, run the View-Section (6.31.10) command from the paper space layout (1.16.15) where the standard drawing views exist. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/GdSections.pn

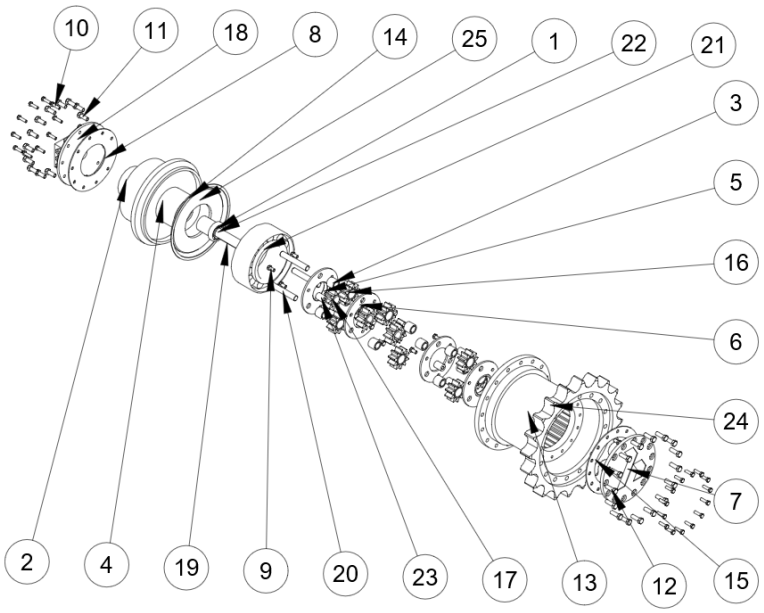
Figure 1.600



Special views

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/driveass
It is possible to generate drawing views for a selected exploded representation (3.6.7). Normally, an orthogonal projection is used for that; however, BrcisCAD allows to create any standard projection for exploded representations. Sections and detailed views are also supported. To create a view for an exploded representation,run the ViewBase (6.31.3) command with the Special views option, select the Exploded view option and then select the required exploded representation from the dialog box. Generated views for exploded representation can be associative with the original model, soannotations and balloons (3.10.2)can be placed. To do that, the views should be created or updated withGENERATEASSOCVIEWS (1.13.11) = ON.

Figure 1.601



Bill of Materials Drive_Assembly		
No.	Component	Quantity
1	BBRG-03	2
2	BBRG-03	1
3	BBRG-60	9
4	Body	1
5	Carrier_1	2
6	Carrier_2	2
7	Cover	1
8	Cover_Interal_Seal	1
9	DIN 933	12
10	DIN 933	24
11	DIN 933	16
12	DIN 933	16
13	Drum	1
14	External_Medium_Seal	1
15	External_Seal	1
16	Gear_Planet	9
17	Gear_Sun	3
18	Internal_Cover	1
19	Key	1
20	Pin	3
21	RLR BRG-32	1
22	Shaft	1
23	Sleeve	6
24	Sprocket	1
25	Washer	1

Updating Drawing Views

By default BricsCAD checks whether the source 3D solid model or the associated exploded representation was modified and automatically recalculates the drawing when:

- opening a paper space layout (1.16.15) containing out-of-date views if the VIEWUPDATEAUTO (1.13.11) system variable is ON.
- running the ViewUpdate (6.31.12) command.
- running the BmUpdate (6.8.25) command (BricsCAD Platinum only).

NOTES

- Hidden line removal calculation might take some time for complex 3D models. In such cases it is recommended to set VIEWUPDATEAUTO (1.13.11) = OFF. The viewport border of out-of-date drawing views turns red to indicate an update is needed. When executing the ViewUpdate (6.31.12) command, BricsCAD prompts you to either select the drawing views to be updated or update all drawing views.
- If GENERATEASSOCVIEWS (1.13.11) = ON, associative dimensions (1.21.3) will be updated according to the modified 3D geometry.

Exporting Views to Model Space

The associativity mechanism of drawing views in a paper space layout (1.16.15) allows limited control by the user. Such views can be moved or scaled, but you cannot edit the geometry. In order to get the full control to the drawing views geometry, the ViewExport (6.31.7) command allows to move or copy drawing views to model space (1.16.12) or to a separate drawing. Exported drawing views lose their associativity with the 3D model and become standard blocks, which can be exploded, edited, erased, etc.

Tutorial: Copying Layouts to Model Tab Click here (1.5.3) to watch

- symbol, and its color and size
- color, type, and weight of detail view boundary

Tutorial: Generate drawing views in BricsCAD Click here (1.6.33) to watch

1.17 2D Drafting

1.17.1 Working with external references

You can link entire drawings to the current drawing as external references. Unlike inserting a drawing as a block, in which you add all the entities from the separate drawing into the current drawing, external references attach a pointer to the external file. The entities in the external reference appear in the current drawing, but the entities themselves are not added to the drawing. Thus, attaching an external reference does not significantly increase the size of the current drawing file. External references provide additional capabilities not available when you insert a drawing as a block. When you insert a drawing as a block, the entities are stored in the drawing. Any changes you make to the original drawing are not reflected in the drawing in which you inserted it. When you attach an external reference, however, any changes you make to the original drawing file are reflected in the drawings that reference it. These changes appear automatically each time you open the drawing containing the external reference. If you know that the original drawing was modified, you can reload the external reference anytime you're working on the drawing. External references are useful for assembling master drawings from component drawings. Use external references to coordinate your work with others in a group. External references help reduce drawing file size and ensure that you are always working with the most recent version of a drawing. Commands:XREF (6.34.8),-XREF (6.34.9),XATTACH (6.34.1)andXCLIP (6.34.2) The**Xref**command attaches DWG files to the current drawing through theDrawing Explorer (1.14.2)(short for "external reference"). The**-Xref**command attaches DWG files to the current drawing through the command bar. The**Xattach**command attaches externally-referenced drawings through a dialog box (short for "external attach"). The**Xclip**command clips externally-referenced drawings, and adjust front and rear clipping planes (short for "xref clip"). The**XDWGFADECTL**system variable controls the fading of exnternal references. Values between 0 (no fading) and 90 are accepted. The fading factor affects the screen display only, not the printing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalRefer
External References (elevations) display faded: XDWGFADECTL + 50 The**XNOTIFYTIME**system variable defines the frequency to check for modified externally referenced resources: xrefs, images and pdf attachments if**XREFNOTIFY, IMAGENOTIFY**and/or**PDFNOTIFY** is ON. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sourc
The**XREFNOTIFY**system variable controls whether warnings for missing or modified xrefs are issued.

Figure 1.602

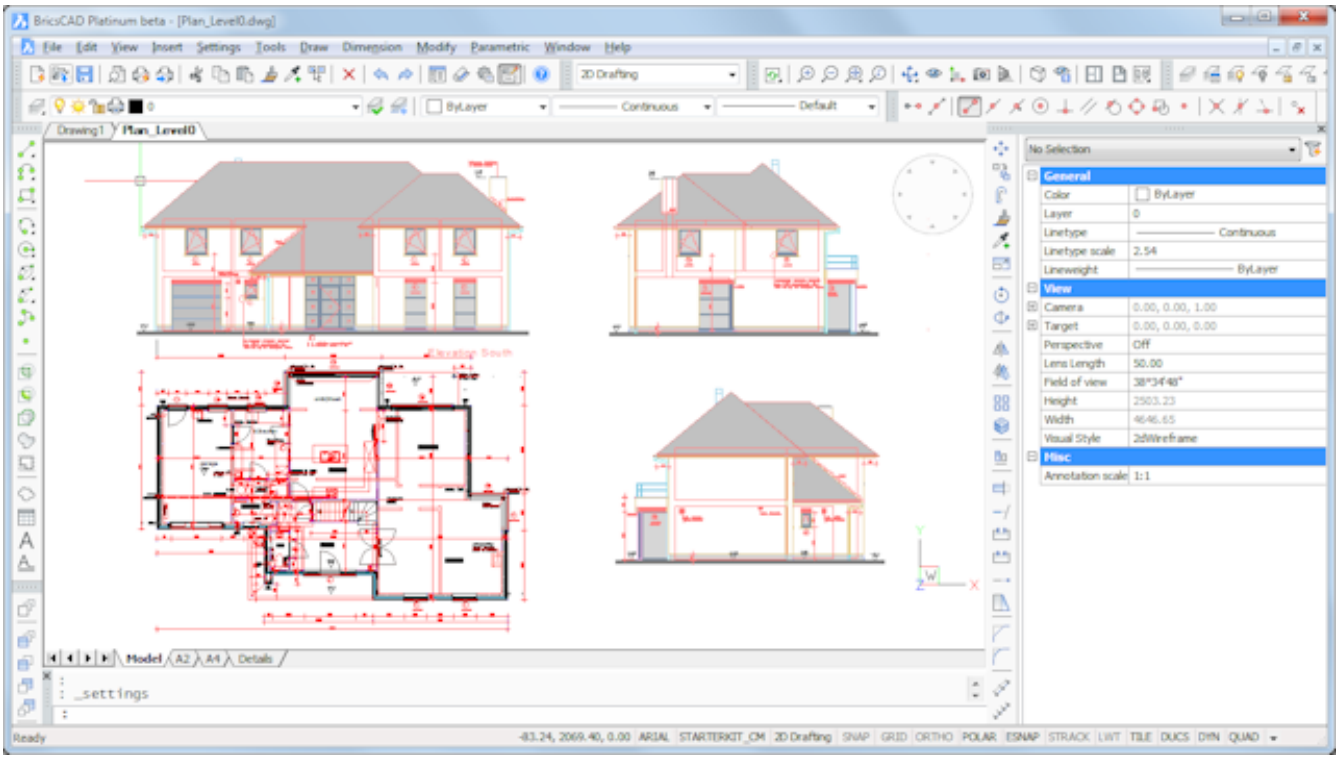
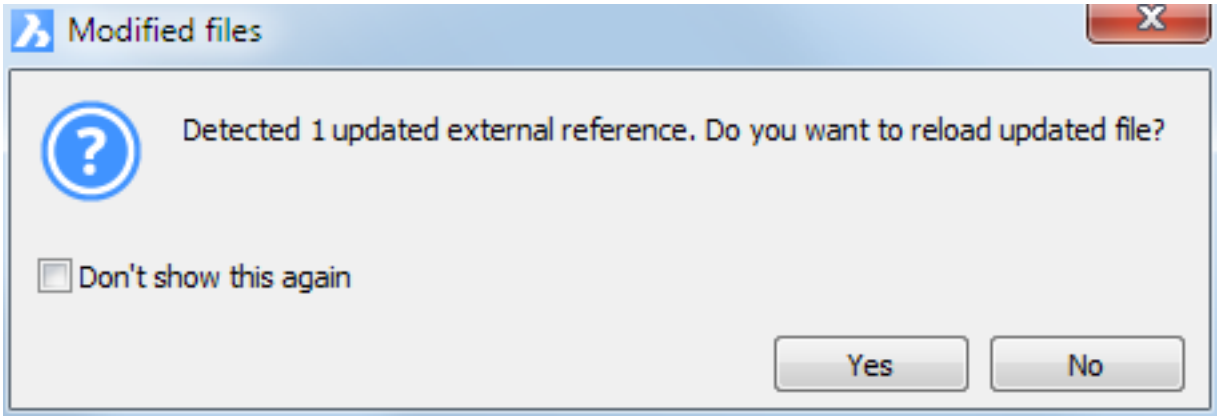


Figure 1.603



1.17.2 Drawing Entities Overview

In a drawing you can create a variety of different entity types. Drawing entities can be very simple such as lines, circles, arcs, points and rays or complex such as polylines, splines and planes. Apart from 2D entities you can also create 3D surface entities or 3D solids in BricsCAD. To further complete your drawing BricsCAD provides a series of tools to add text, hatching and dimensions. To create an entity you can choose between selecting the command in the Draw menu, using the tools on the Draw, Draw 3D and Solids toolbars or typing the command in the command bar. When you use a tool or a drawing command, the program prompts you to enter coordinate points, such as endpoints or insertion points. You can enter the points or distances either using a mouse or by typing coordinate values in the command bar. As you draw, BricsCAD also displays a context-sensitive prompt menu with appropriate additional options for the type of entity you are drawing. New entities are created on the current layer (1.17.7), using the current color (1.17.4), linetype (6.20.33) and lineweight (1.17.5).

1.17.3 Fill Mode

Command: FILL (6.14.6) The **Fill** command toggles the display of filled and hatched areas. Specifies whether multilines, traces, solids, hatches (including solid-fill) and wide polylines are filled in. If **FILLMODE** is OFF, all filled entities display and print as outlines. You can reduce the time it takes to display or print a drawing by turning FILLMODE off.

	Polyline	**Donut**	**Trace**	**Plane (Solid)**
Fill On	(\vref {fig:C:/Dev/Task-2448/images/staticbricsys	(\vref {fig:C:/Dev/Task-2448/images/staticbricsys	(\vref {fig:C:/Dev/Task-2448/images/staticbricsys	(\vref {fig:C:/Dev/Task-2448/images/staticbricsys
Fill Off	(\vref {fig:C:/Dev/Task-2448/images/staticbricsys	(\vref {fig:C:/Dev/Task-2448/images/staticbricsys	(\vref {fig:C:/Dev/Task-2448/images/staticbricsys	(\vref {fig:C:/Dev/Task-2448/images/staticbricsys

Setting Fill Mode

1. To toggle **Fill Mode** on / off do one of the following:
   - Choose **Fill** in the **Settings** Menu.
   - Type **fill** or **fillmode** in the command bar, then type **T** + Enter.
2. Regenerate (1.16.4) the viewport.

Note: If multiple viewports are open, click the Regen All (1.16.4) tool button ( ) on the **View** toolbar (**Redraw/Regen** flyout) to regenerate all open viewports simultaneously.

Figure 1.604



Figure 1.605



Figure 1.606



Figure 1.607



Figure 1.608

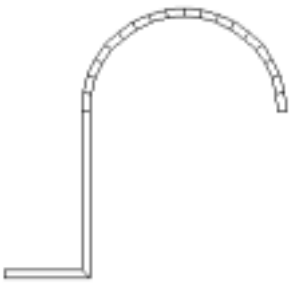


Figure 1.609

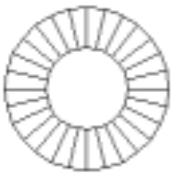


Figure 1.610

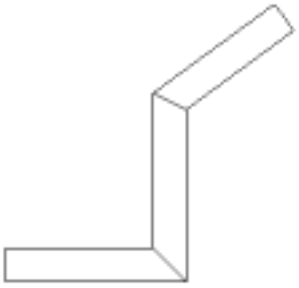
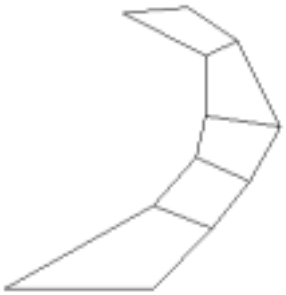


Figure 1.611



1.17.4 Entity Color

Commands: COLOR (6.9.33) and -COLOR (6.9.35) The **Color** command specifies the current color, through a dialog box. The **-Color** command sets the current color, through the command bar. An entity's color determines how it is displayed and how it prints. Entities are created in the current color. You can choose between 255 **index colors** or define a **true color.**

Index Colors

Index color is the specification of the color of a pixel on a display screen using a an 8-bit color value, allowing up to 256 possible colors. Each of the Index Colors has a unique number from 1 to 255. Seven of the index colors can also be referred to by name: red (1), yellow (2), green (3), cyan (4), blue (5), magenta (6) and white/black (7). Index color 7 displays white on a black screen background and black on a white screen background. Index color 7 always prints in black. The two additional color properties are BYLAYER and BYBLOCK. These color properties cause an entity to adopt the color either of the layer or of the block in which it is a member. BYLAYER is color number 256, and BYBLOCK is color number 0. In all commands where you would use a color, you can indicate BYLAYER and BYBLOCK as well as by numbers 256 and 0, respectively.

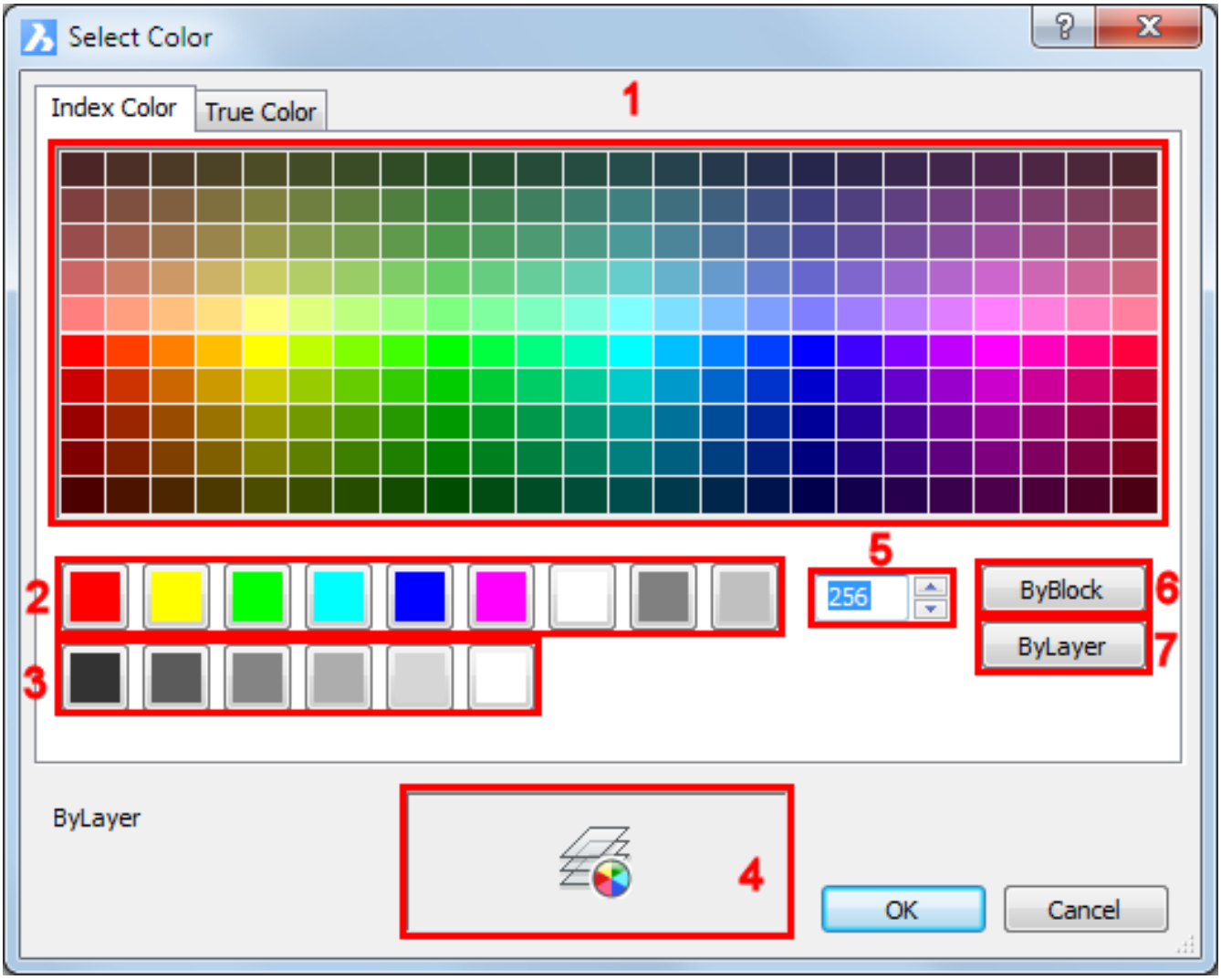
Color BYLAYER: Entities which have a color **BYLAYER** adopt the color of their layer. This allows you to change the color of all such entities by adjusting the color of the layer.

Color BYBLOCK: Entities which have a color **BYBLOCK** are drawn in index color 7 (black or white, depending on the screen background color). When included in a block definition, such entities adopt the color of the block. **Note:** Whether entities using an index color will print in this color, depends on the **Color Table** (CTB) or **Style Table** (STB) that is used for printing. Only if the **Color** setting in the CTB or STB definition file is set to **Use Object Color** , the printed color matches the entity color.

Index color dialog (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntitie

1. Colors 10 - 249
2. Colors 1 to 9
3. Colors 250 - 255
4. Current Color
5. Color Number field
6. 'Color By Block' button
7. 'Color By Layer' button

Figure 1.612


Select Color
Index Color
True Color
1
2
3
4
5
256
ByBlock
6
ByLayer
7
ByLayer
OK
Cancel

True Colors

True color is the specification of the color of a pixel on a display screen using a 24-bit value, which allows up to 16 777 216 possible colors. The number of bits used to define a pixel's color shade is its **bit-depth**. True color is sometimes known as **24-bit color**. Some new color display systems offer a 32-bit color mode. The extra byte, called the **alpha channel**, is used for control and special effects information. True colors use a **RGB** color definition (**R**ed, **G**reen, **Bl**ue). Each of these parameters has a range from 0 to 255. The RGB definition for black is (0,0,0), the RGB definition for white is (255,255,255).

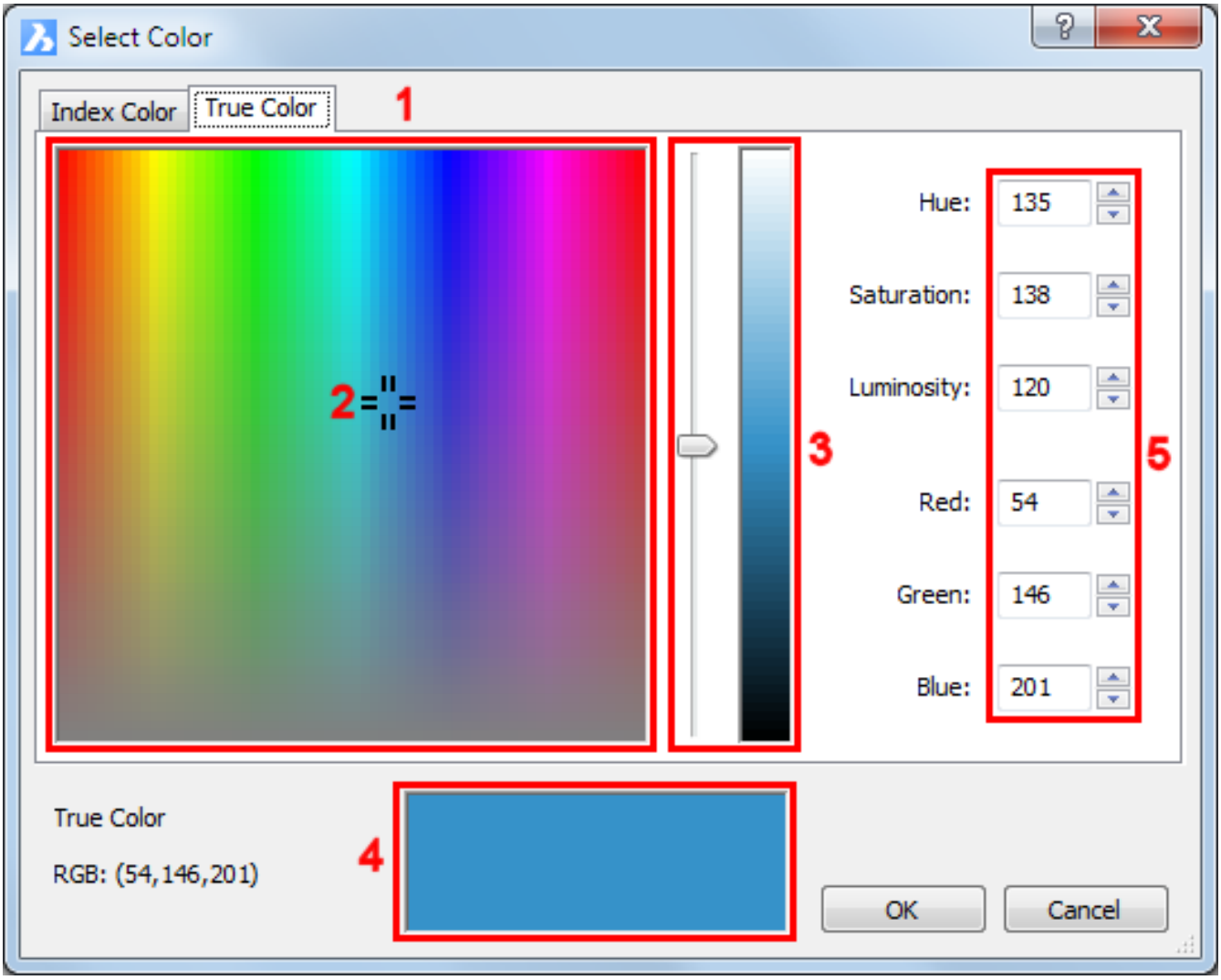
NOTES

- In drawings that use **Style Tables** (STB) to set up plotting configurations, entities in a true color will only be printed in this color if the **Color** setting in the STB definition file is set to **Use Object Color.**
- In drawings that use **Color Tables** (CTB) to set up plotting configurations, entities in a true color always print in this color.

True color dialog (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities

1. Color Selection pane
2. Color Picker
3. Luminosity slider
4. Current Color
5. Color Parameter fields

Figure 1.613


Select Color
Index Color
True Color
1
2
3
4
5
Hue: 135
Saturation: 138
Luminosity: 120
Red: 54
Green: 146
Blue: 201
True Color
RGB: (54,146,201)
OK
Cancel

Setting the current entity color

1. To open the Select Color dialog, do one of the following:
   - In the **Color** field of the Entity Properties toolbar, choose **Select Color** .
   - Click the **Select Color** tool button ( ) on the Settings toolbar.
   - In the BricsCAD Properties (1.12.4) Bar click **Color** , then choose **Select Color** in the drop down list.
   - Double click the **Color Field** in the Status Bar (1.12.2).
   - Type **color** in the command bar, then press Enter.
2. (option) To select an index color, click the **Index Color** tab on the Select Color dialog, then do one of the following:
   - Click one of the colored tiles.
   - Type the color number in the **Color Number** field.
3. (option) To define a true color, click the **True Color** tab on the Select Color dialog, then do one of the following:
   - Click the in **Color Selection** pane.
   - Set the color parameters in the **Color Parameter** fields.
3. Click the **OK** button to confirm.

1.17.5 Lineweight

Lineweights determine how thick or thin entities appear on the screen and / or when printed. The following lineweights are available: BYLAYER, BYBLOCK, DEFAULT and many additional lineweights in millimeters or inches, depending on the **Lineweight Units** (LWUNITS) system variable. You cannot assign lineweights to planes, points, TrueType fonts, and raster images. New entities are drawn using the current lineweight (1.17.5).

Lineweight BYLAYER: When you create an entity, it is created using the current lineweight. By default, the current lineweight for a new entity is BYLAYER. This means that the entity lineweight is determined by the current layer. When you assign BYLAYER, changing a layer's lineweight changes the lineweight of all such entities on that layer.

Lineweight BYBLOCK: Entities created using lineweight BYBLOCK, are drawn using the DEFAULT lineweight until include them into a block. The entities then inherit the block's lineweight setting when you insert the block into a drawing.

DEFAULT lineweight: The default lineweight is saved in the **Default Lineweight** (LWDEFAULT) system variable. Entities created using the default lineweight will be adjusted if the **Default Lineweight** setting is redefined.

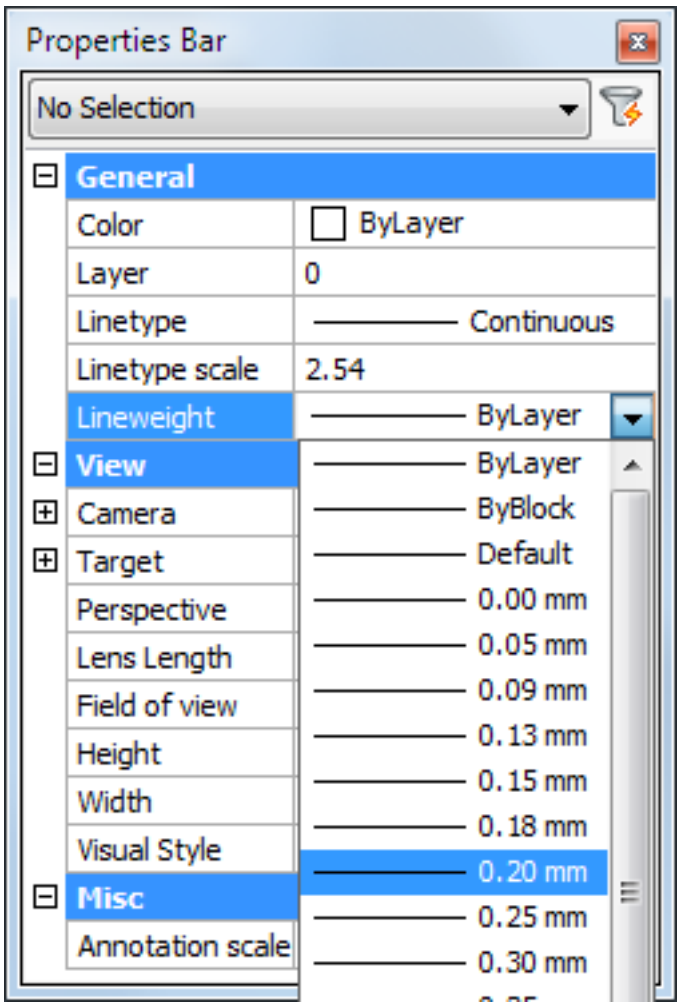
NOTE	For entities using an index color (1.17.4), lineweights will be used for printing only if the **Color Table** (CTB) or **Plot Style** (STB) is defined as 'Use object lineweight'. Otherwise the line weight is defined by the objects color or plot style. Entities which are created in a true color (1.17.4) always use the object line weight when printed.

Setting the current lineweight

1. (option) On the BricsCAD Properties Bar (1.12.4), select **Lineweight**. Make sure, no entity is selected. Choose a lineweight in the drop down list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source

2. (option) On the **Entity Properties** toolbar, click the down arrow next to the **Lineweight** field, then choose a lineweight in the drop down list.

Figure 1.614


Properties Bar
No Selection
General
Color
ByLayer
Layer
0
Linetype
Continuous
Linetype scale
2.54
Lineweight
ByLayer
View
Camera
Target
Perspective
Lens Length
Field of view
Height
Width
Visual Style
Misc
Annotation scale
ByLayer
ByBlock
Default
0.00 mm
0.05 mm
0.09 mm
0.13 mm
0.15 mm
0.18 mm
0.20 mm
0.25 mm
0.30 mm

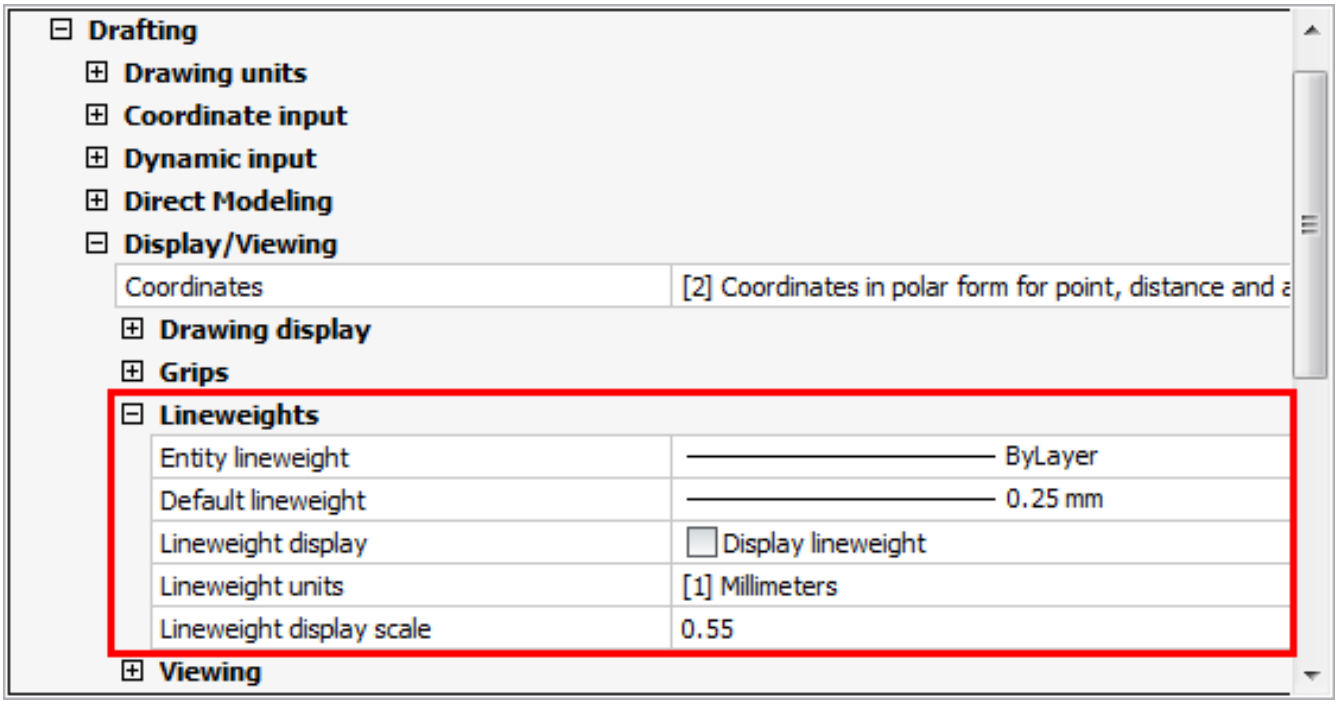
Defining the lineweight settings

1. Choose **Settings** in the **Settings** menu. The Settings (1.13.11) dialog opens.
2. Select the **Drawing settings** category.
3. Expand the **Display/Viewing** settings sub-category.
4. Expand the **Lineweights** settings group. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/
5. (option) Select the **Default Lineweight** (LWDEFAULT) system variable, then select a lineweight in the drop down list.
6. (option) Select the **Lineweight Display** (LWDISPLAY) system variable, then click the checkbox to toggle the display of lineweights.
7. (option) Select the **Lineweight Units** (LWUNITS) system variable, then choose **Millimeters** or **Inches**.
8. Close the Settings dialog.

Note: Click the lineweight field (LWT) in the Status Bar (1.12.2) to toggle the display of lineweights on/off.

Tutorial: Printing in BricsCAD: Plot Style Tables Click here (1.8.5) to watch

Figure 1.615


Drafting
Drawing units
Coordinate input
Dynamic input
Direct Modeling
Display/Viewing
Coordinates
[2] Coordinates in polar form for point, distance and a
Drawing display
Grips
Lineweights
Entity lineweight
ByLayer
Default lineweight
0.25 mm
Lineweight display
Display lineweight
Lineweight units
[1] Millimeters
Lineweight display scale
0.55
Viewing

1.17.6 Entity Linetype

You use different linetypes to differentiate the purpose of one line from another. A linetype consists of a repeating pattern of dots, dashes, or blank spaces. Linetypes determine the appearance of entities both on the screen and when printed. By default, every drawing has at least three linetypes: CONTINUOUS, BYLAYER and BYBLOCK. Your drawing may also contain an unlimited number of additional linetypes. Linetype definitions are saved in the drawing. New linetypes are either imported from other drawings or loaded from a linetype file (*.lin). Which linetype file is used is controlled by the MEASUREMENT system variable:

- MEASUREMENT = ON uses **Iso.lin**: Linetype definitions are expressed in mm.
- MEASUREMENT = OFF: **Default.lin**: Linetype definitions are expressed in inch.

New entities are drawn using the current linetype.

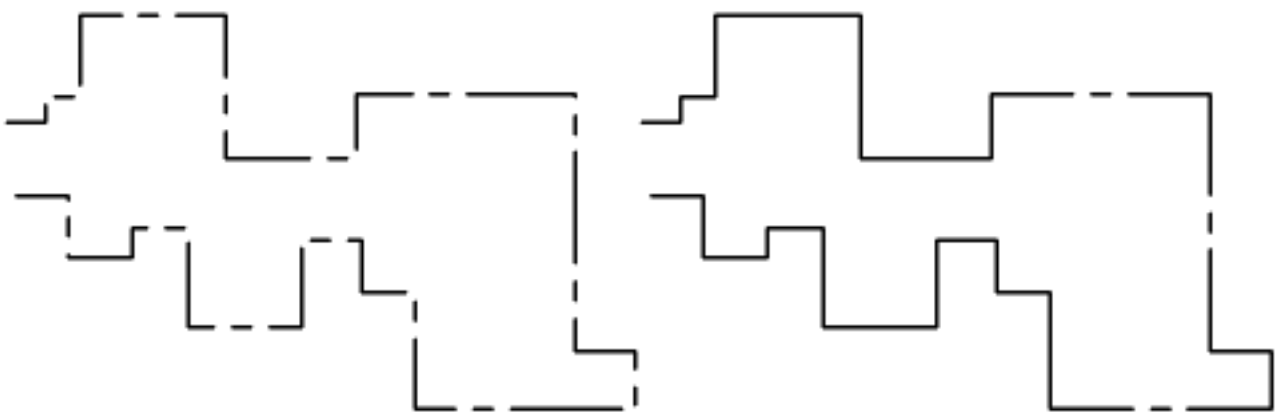
Linetype BYLAYER: Entities which have a linetype BYLAYER adopt the linetype of their layer. This allows you to change the linetype of all such entities by adjusting the linetype of the layer.

Linetype BYBLOCK: Entities created using linetype BYBLOCK, are drawn as continuous lines until you include them into a block. The entities then inherit the block's linetype setting when you insert the block into a drawing.

Linetype scale (LTSCALE) Non-continuous linetypes consist of lines, gaps and dots put together in a variety of patterns. In complex linetypes also text or shapes can be included. In the linetype definition the length of the lines and gaps are either defined in inch (default.lin) or mm (iso.lin). If the drawing units are different from inch or mm, linetypes need to be scaled. The linetype scaling is controlled by the **Linetype Scale** (LTSCALE) system variable, which must be set with respect to the drawing units (DU): **Drawing Unit LTSCALE** Imperial (MEASUREMENT = OFF) Inch 1 Foot 1/12 Metric(MEASUREMENT = ON) mm 1 cm 0.1 m 0.001 BricsCAD will always put a dash at the start and end of a line, polyline, arc or spline. The linetype pattern is also nicely centered, so that both ends of the entity look the same.

Polyline Generation (PLINEGEN) Linetypes are normally generated from vertex to vertex. Polylines of which the vertices are very close together might be rendered as a continuous line, if the linetype pattern does not fit between two subsequent vertices. The **Polyline Generation** (PLINEGEN) system variable addresses this problem: when set to 1 or ON, the linetype is drawn from one end of the polyline to the other end, instead of from vertex to vertex. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/00EntityCreationSetting
The same polyline drawn with PLINEGEN on (left) and off (right).

Figure 1.616



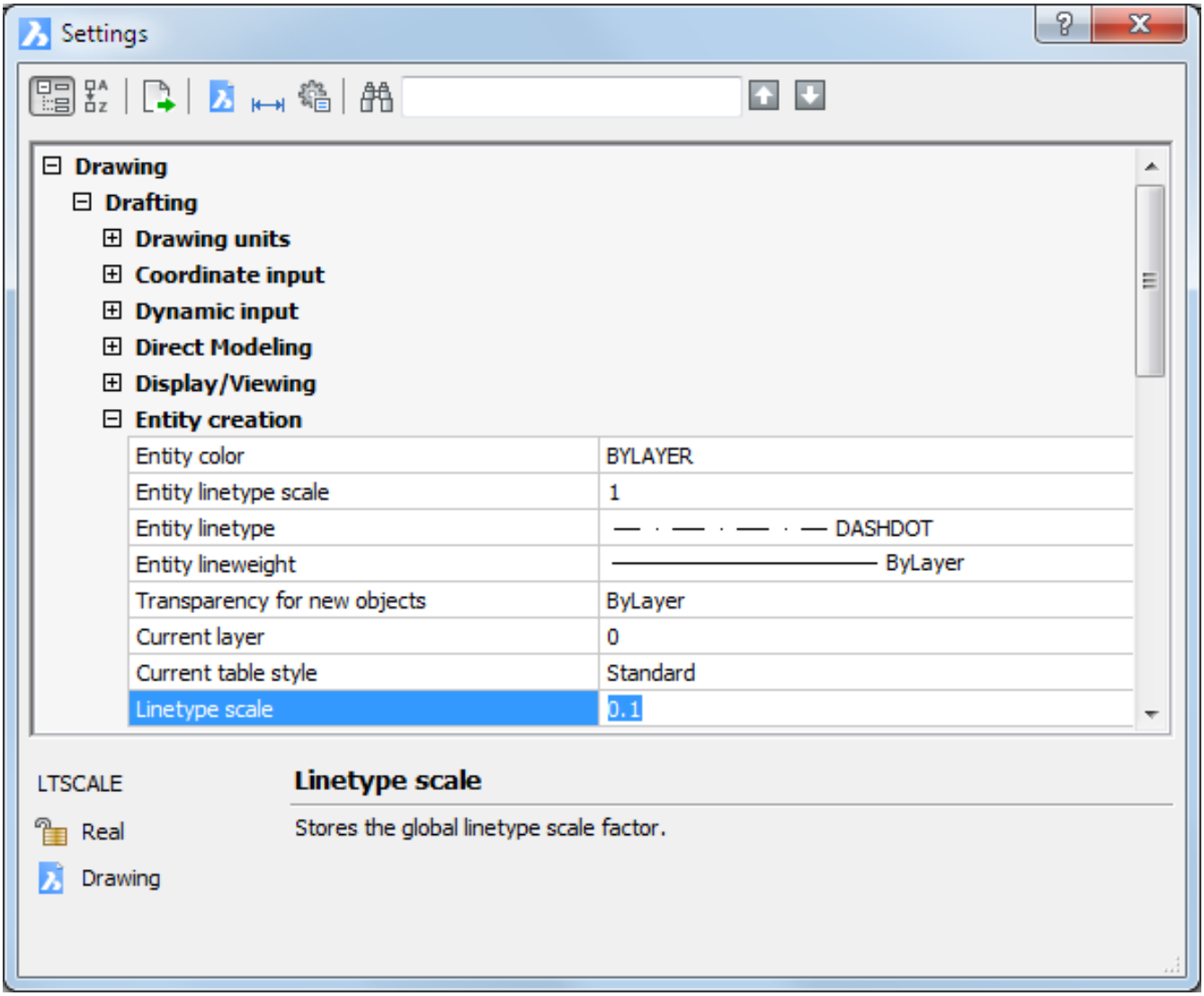
Paper space linetype scale (PSLTSCALE) PSLTSCALE controls linetype scaling in paperspace layouts. If ON, linetype scaling is independent from the viewport scale. In general, PSLTSCALE is best set ON. Remember that this variable can be set per layout.

Model space linetype scale (MSLTSCALE) If MSLTSCALE is ON, the current annotation scale (1.21.27) (CANNOSCALE) is applied to linetypes in model space. MSLTSCALE is best set OFF. It might be usefull to switch it ON when printing from model space.

Setting the Global Linetype Scale

1. Choose **Settings** in the **Settings** menu. The Settings (1.13.11) dialog opens.
2. Select the **Drawing settings** category.
3. Expand the **Entity Creation** settings sub-category.
4. Select the **Linetype Scale** setting (LTSCALE). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us
5. Type a new value in the **Global Linetype Scale** field, then press Enter. The drawing is regenerated.
6. Close the **Settings** dialog.

Figure 1.617

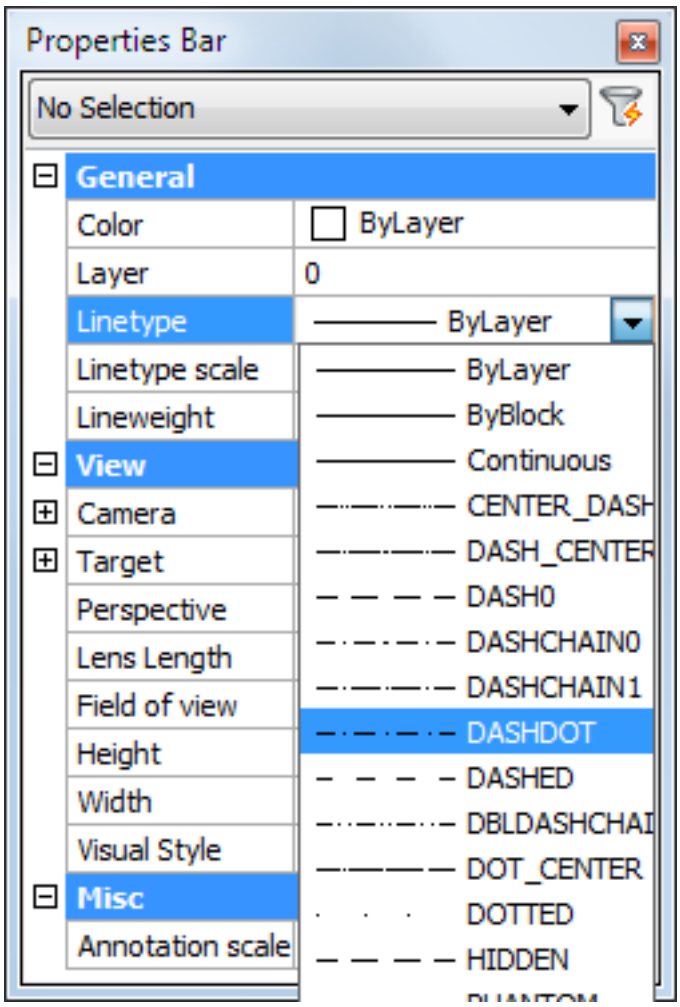

Settings
Drawing
Drafting
Drawing units
Coordinate input
Dynamic input
Direct Modeling
Display/Viewing
Entity creation
Entity color
BYLAYER
Entity linetype scale
1
Entity linetype
DASHDOT
Entity lineweight
ByLayer
Transparency for new objects
ByLayer
Current layer
0
Current table style
Standard
Linetype scale
0.1
LTSCALE
Real
Drawing
Linetype scale
Stores the global linetype scale factor.

Setting the current linetype

Do one of the following:

- On the BricsCAD Properties Bar (1.12.4), select **Linetype** . Make sure, no entity is selected. Choose a linetype in the drop down list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Drawing
- On the **Entity Properties** toolbar, click the down arrow next to the **Linetype** field, then choose a linetype in the drop down list.
- Right click on the **Linetype** field in the Status Bar (1.12.2), then choose a linetype in the context menu.

Figure 1.618


Properties Bar
No Selection
General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
Lineweight
View
Camera
Target
Perspective
Lens Length
Field of view
Height
Width
Visual Style
Misc
Annotation scale
ByLayer
ByBlock
Continuous
CENTER_DASH
DASH_CENTER
DASH0
DASHCHAIN0
DASHCHAIN1
DASHDOT
DASHED
DBLDASHCHAI
DOT_CENTER
DOTTED
HIDDEN

Adding a new linetype

1. In the **Tools > Drawing Explorer** choose **Linetype...** The **Drawing Explorer - Linetypes** dialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/00EntityCreationS
2. Click the **New** tool button ( ) in the **Details** toolbar. The **Linetypes** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/00EntityCreationSettings/ART/04L
3. (option) Click the **File** button on the **Linetypes** dialog to load a different linetype file (*.lin).
4. Select a linetype, then click the **OK** button. The linetype is added.
5. (option) Click the linetype name to rename the linetype.
6. (option) click the linetype description to edit the description of the linetype.

Figure 1.619

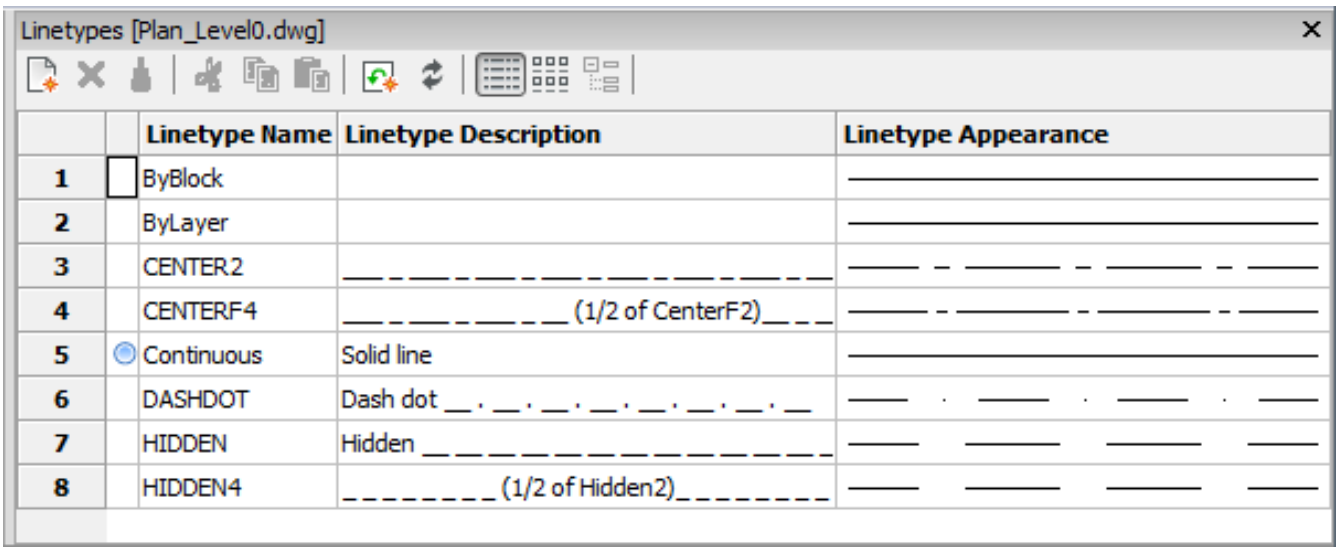
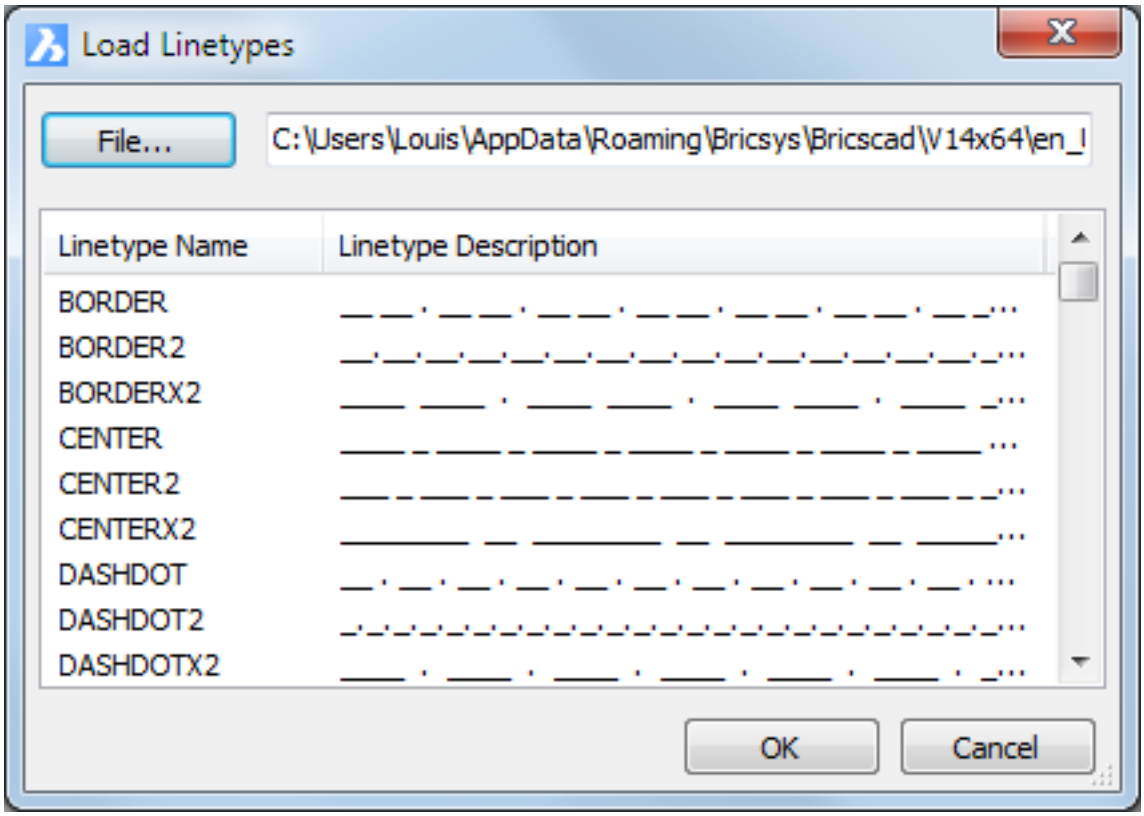


Figure 1.620

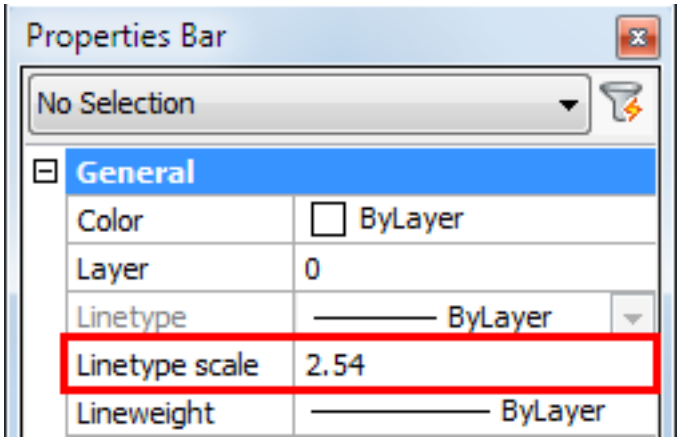


Setting the Entity Linetype Scale

The CELTSCALE system variable sets the value of the Linetype Scale property for new entities. This variable is best set to 1, except for special linetypes, such as BATTING or GAS_LINE, which most likely need to be scaled.

1. On the BricsCAD Properties Bar (1.12.4), select **Linetype scale** . Make sure, no entity is selected.
2. Type a new value in the Linetype scale field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrC

Figure 1.621



1.17.7 Current Layer

When you create new entities, they are drawn on the current layer. To draw new entities on a different layer, you must first make that layer the current layer.

Setting the current layer

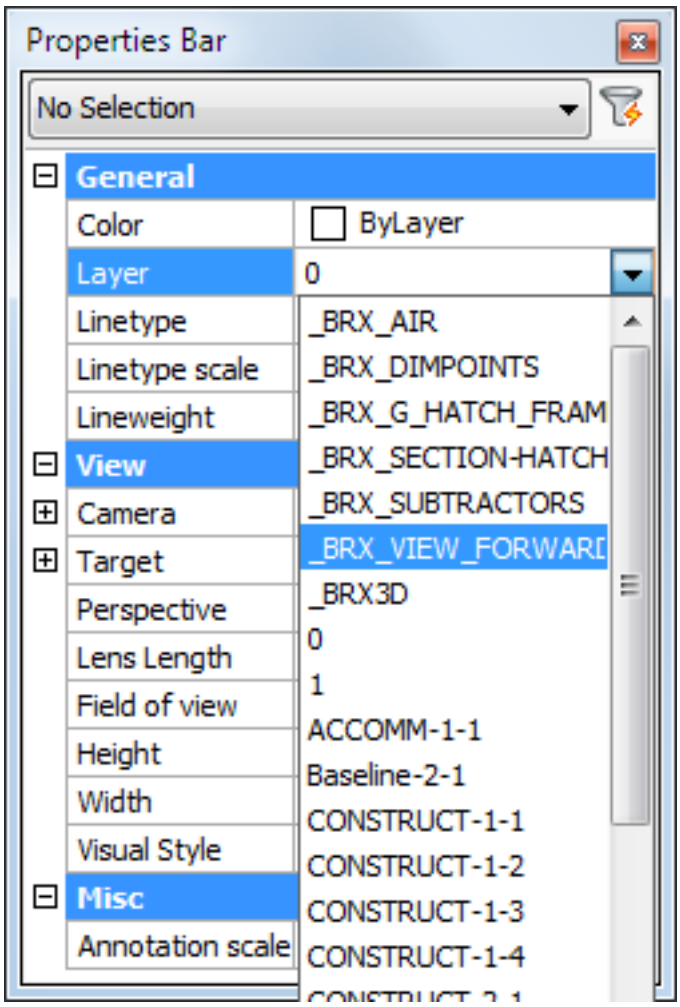
Do one of the following:

- On the BricsCAD Properties Bar (1.12.4), select **Layer** . Make sure, no entity is selected. Choose a layer in the drop down list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntitie
- On the Entity Properties toolbar, click the down arrow next to the **Layer** field, then choose a layer in the drop down list.
- Right click on the **Layer** field in the Status Bar (1.12.2), then choose a layer in the context menu.

NOTES

- It is possible to turn the display of the current layer off (). As a result, everything you draw will be invisible until the display of the current layer is turned on () again. A warning displays if you try to turn off the current layer.
- The LAYISO (6.20.9) command allows to isolate layers by either lock or turn off other layers. If no entity on the current layer is selected, the layer of the entity that is selected first temporarily becomes the current layer until the LAYUNISO (6.20.18) command is executed.
- It is not possible the freeze () the current layer.

Figure 1.622


Properties Bar
No Selection
General
Color
ByLayer
Layer
0
Linetype
Linetype scale
Lineweight
View
Camera
Target
Perspective
Lens Length
Field of view
Height
Width
Visual Style
Misc
Annotation scale
_BRX_AIR
_BRX_DIMPOINTS
_BRX_G_HATCH_FRAM
_BRX_SECTION-HATCH
_BRX_SUBTRACTORS
_BRX_VIEW_FORWARI
_BRX3D
0
1
ACCOMM-1-1
Baseline-2-1
CONSTRUCT-1-1
CONSTRUCT-1-2
CONSTRUCT-1-3
CONSTRUCT-1-4

1.17.8 Transparency

The Transparency property allows to control the transparency of drawing entities. You can set the transparency for a selection of entities or for all entities on the same layer. The default transparency for new entities is controlled by the CETRANSPARENCY system variable. The default transparency for Hatches and Gradient Fills is controlled by the HPTRANSPARENCY system variable. Transparency can be used to enhance the display of your drawings, e.g. to de-emphasize less important details or entities that are used for reference only. Transparencies will only be printed if the Plot Transparencies option is checked on the Print or Page Setup dialog boxes. The options to set the Transparency are:

- **ByLayer** - apply the value of the transparency property of the layer.
- **ByBlock** - the transparency value is controlled by block.
- **Value** - values between 0 (fully opaque) and 90 are accepted. The transparency level is limited to 90 % to avoid confusion with layers that are frozen or turned off.

Tutorial: Making Entities Transparent in BricsCAD Click here (1.5.2) to watch

1.17.9 Lines

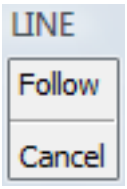
Command: LINE (6.20.31) A line consists of two points: a start point and an endpoint. Using the **Line** command you can draw a series of connected lines, but each line is considered a separate line entity.

Drawing lines

1. Do one of the following

   - Click the **Line** tool button ( ) on the **Draw** toolbar.
   - Choose **Line** in the **Draw** menu.
   - Type **line** in the command bar, then press Enter.
   - Type **L** in the command bar, then press Enter.

   You are prompted: ENTER to use last point/Follow/<Start of line>: A prompt menu displays:



2. Specify the start point. The line displays dynamically.
3. Specify the end point. The line is drawn and the following line displays dynamically starting in the endpoint of the first line.

   - Using Ortho Mode (1.15.2) or Polar Tracking (1.15.16) you can type the length of the line in the command bar.
   - Using Snap Tracking (1.15.17) you can position the end point with respect to the start point and one or two snap points on entities in the drawing.

   The prompt menu changes: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10D

4. (option) Repeat step 3 to draw a series of connected lines. When the second line is drawn, the options in the prompt menu are: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEn
   After the third line, the options in the prompt menu are: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
5. Right click to stop drawing lines.
6. (option) Right click to restart.

Figure 1.623



Figure 1.624

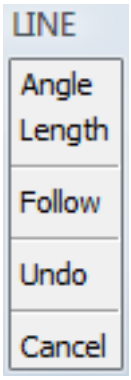


Figure 1.625



1.17.10 Rays

Command : RAY (6.26.1) A **Ray** is a line that starts at a point and extends to infinity. Because rays extend to infinity, they are not calculated as part of the drawing extents. The default method for drawing a ray is to select the start point of the ray, and then specify its direction. Rays and Infinite Lines (1.17.11) or sometimes referred to as construction lines.

Drawing rays

1. Do one of the following:

   - Click the **Ray** tool button ( ) on the **Lines** flyout of the **Draw** toolbar.
   - Choose **Ray** in the **Draw** menu.
   - Type **ray** in the command bar, then press Enter.

   You are prompted: Infinite ray: Bisect/Horizontal/Vertical/Angle/Parallel/<Start of Ray>: A prompt menu opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Dr

2. Specify the start point of the ray. The ray displays dynamically.
3. Specify the direction of the ray. The next ray displays dynamically.
4. (option) Keep specifying directions to draw a series of rays, starting from the same point.
5. Right click to stop drawing rays.

Figure 1.626



1.17.11 Infinite Lines

Command : XLINE (6.34.5) The **Xline** command creates infinite lines. An **Infinite Line** is a line through a given point, oriented at a specified angle and extending to infinity in both directions. Because infinite lines extend to infinity, they are not calculated as part of the drawing extents. The default method for drawing an infinite line is to select a point and then specify its direction. Rays (1.17.10) and Infinite Lines or sometimes referred to as construction lines.

Drawing infinite lines

1. Do one of the following:

   - Click the **Infinite Line** tool button ( ) on the **Lines** flyout of the Draw toolbar.
   - Choose **Infinite Line** in the Draw menu.
   - Type **xline** in the command bar, then press Enter.

   You are prompted: Xline: Bisect/Horizontal/Vertical/Angle/Parallel/<Point along line>: A prompt box displays: (\vref{fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Dra

2. Specify the start point of the infinite line. The infinite line is created. A new infinite line through the same start point displays dynamically.

3. (option) Specify the direction of the infinite line. The infinite line is created. A new infinite line through the same start point displays dynamically.

4. (option) Repeat step 3 to draw a series of infinite lines, through the same start point.

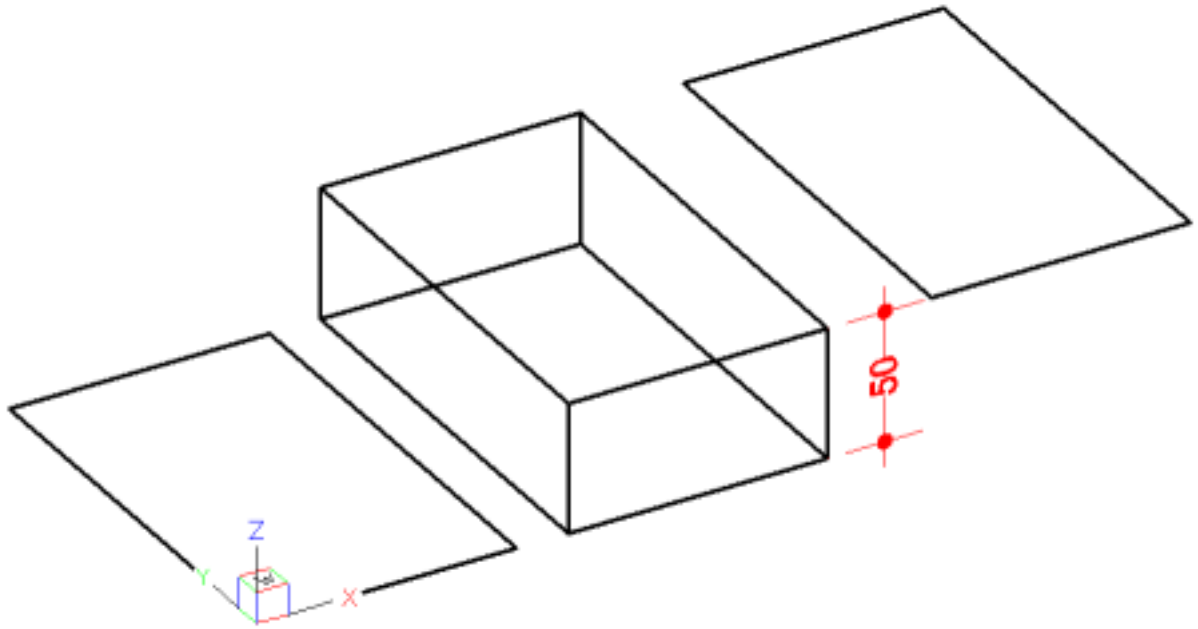
5. Right click to stop drawing infinite lines.

6. (option) Right click to restart.

Figure 1.627



Figure 1.628



1.17.12 Polylines

Commands: PLINE (6.24.27), BOUNDARY (6.6.15) and -BOUNDARY (6.6.16) A **polyline** is an open or closed sequence of connected line and/or arc segments, which are treated as a single entity. Each segment of a polyline can have a width that is either constant or tapers over the length of the segment. The **Pline** command interactively creates open and closed polylines entering points. The **Boundary** and **- Boundary** commands create closed polylines from an enclosed area, defined by other entities. The **THICKNESS** system variable specifies the height of a polyline. If **THICKNESS** = 0 (zero) there is no visual difference between polylines and a series of lines and/or arcs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntities/AP
THICKNESS = 0 (left), **THICKNESS** = 50 (middle), **ELEVATION** = 50 (right) When a polyline is edited, you can modify the entire polyline or change individual segments and you can add or remove vertices. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntities/ART/04PolylinesZe
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntit
zero width polyline constant width polyline (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sou
tapered width polyline **Note:**When Fill Mode (1.13.11) is turned OFF, all filled entities, such as wide polylines and planes, display and print as outlines. Depending on the value of the GRIPS (1.13.11) system variable have grips on their endpoints and vertices and/or at the midpoint of each segment. The value of GRIPS can be 0, 1 or 2.

Value	Grips display
0	No grips
1	Endpoint grips only
2	Endpoint and midpoints grips

Drawing polylines

1. Do one of the following:

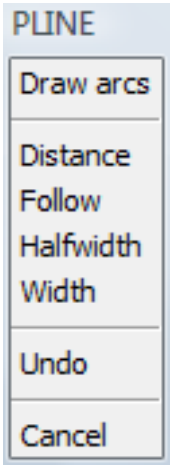


- Click the **Polyline** tool button ( ) on the Draw toolbar.

Figure 1.629

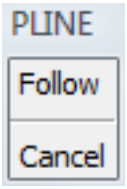


Figure 1.630



- Choose **Polyline** in the Draw menu.
- Type **pline** in the command bar, then press Enter.
- Type **PL** in the command bar, then press Enter.

You are prompted: ENTER to use last point/Follow/<Start of polyline>: A prompt menu displays:



2. Specify the start point of the polyline. The prompt menu changes: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

3. Specify the second point of the polyline. The prompt menu changes: (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc

4. (option) Repeat step 3 to add more straight segments.

5. (option) Do one of the following to start drawing arc segments:

   - Type **A** , then press Enter.
   - Choose **Draw arcs** in the prompt menu.

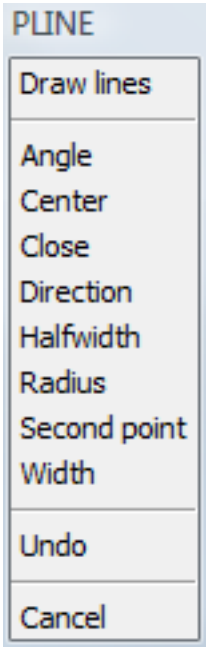
   The prompt menu changes: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10D
   An arc segment displays dynamically. The arc is tangent to the previous line segment.

6. Specify the endpoint of the arc.

7. (option) Repeat step 6 to add more arc segments.

8. (option) Do one of the following to start drawing line segments:

Figure 1.631



- Type **L** , then press Enter.
- Choose **Draw lines** in the prompt menu.

9. Specify the endpoint of the line segment.

10. Right click or choose **Done** in the prompt menu to stop.

11. (option) Right click to restart.

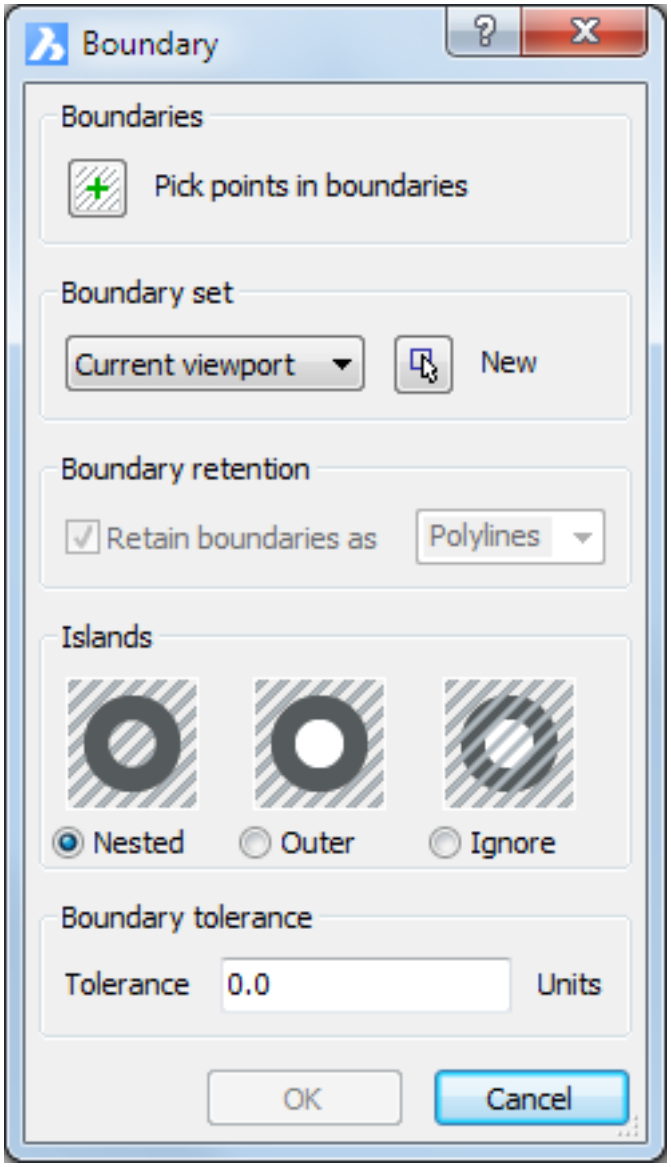
Creating boundary polylines

1. Do one of the following:



- Click the **Boundary Polyline** tool button ( ) on the Draw toolbar.

Figure 1.632



- Choose **Boundary Polyline** in the Draw menu.
- Type **boundary** in the command bar, then press Enter.

The Boundary dialog box opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/



2. (option) Click the **Select** button ( ) The Boundary dialog closes. Select the boundary entities in the drawing, then Press Enter. The Boundary dialog opens.

Figure 1.633

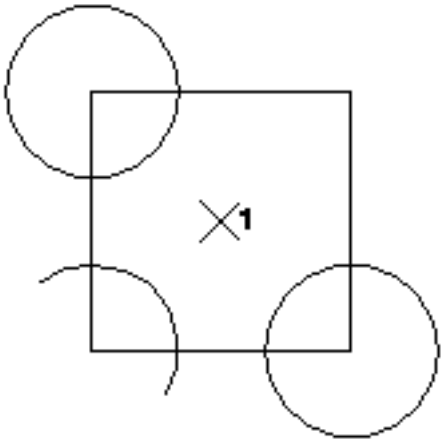
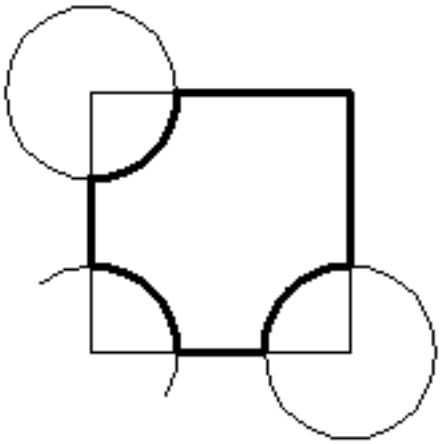


Figure 1.634



3. Click the **Pick points in boundaries** button on the Boundary dialog box. The dialog box closes.
4. Click in the area where you want to create the boundary polyline: point 1 in the image below. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntities/ART/04Bound
5. The boundary polyline displays in dashed lines.
6. (option) Keep picking points to create more boundary polylines.
7. Right click to accept the polyline(s). The Boundary dialog box opens again.
8. Click the **OK** button on the Boundary dialog box. The polyline(s) is (are) created. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntities/ART/04Bound

NOTES

- If you select the boundary entities before launching the **Boundary Polyline** tool, you can make the selection set active by pressing the



Figure 1.635

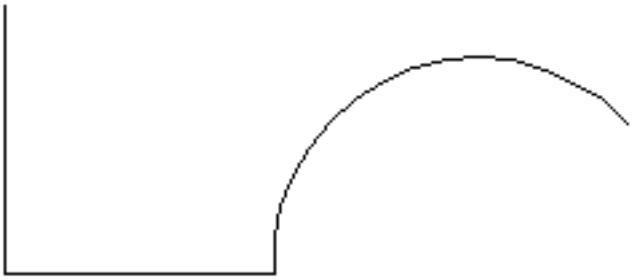


Figure 1.636

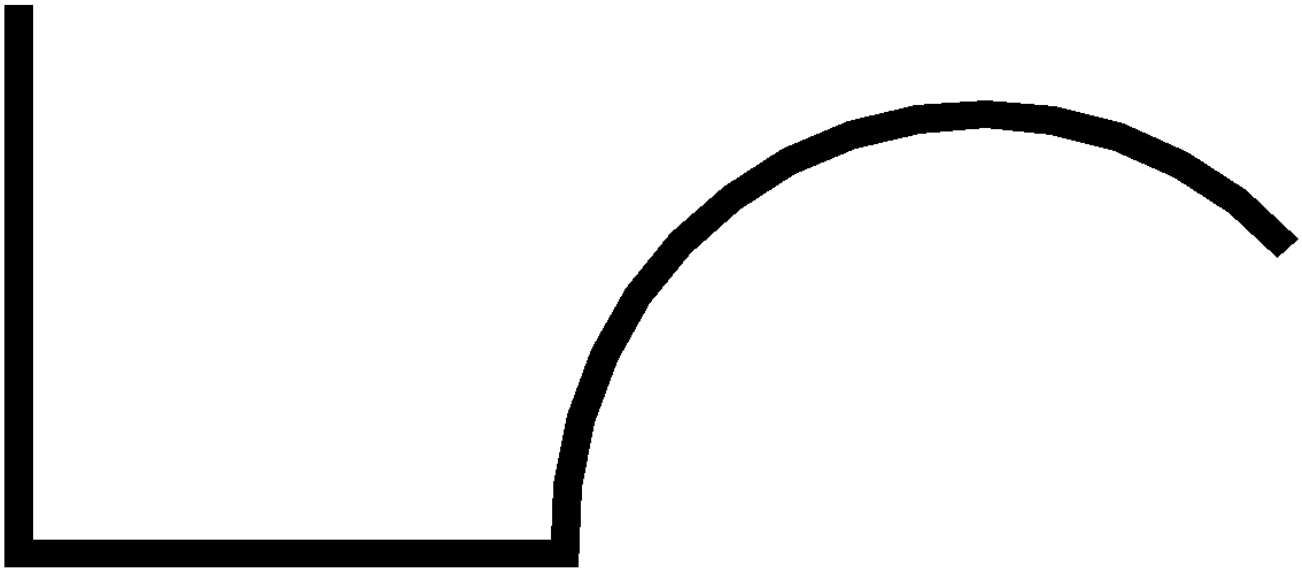
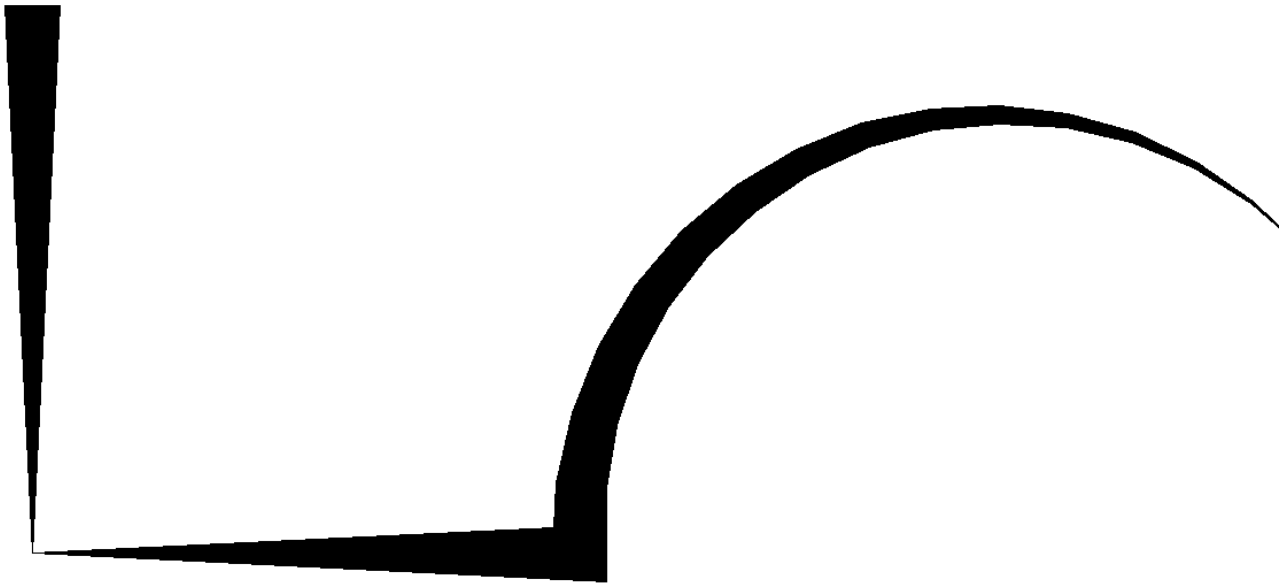


Figure 1.637



1.17.13 Splines

Command: SPLINE (6.27.65) A spline is an open or closed smooth curve defined by a set of points. You can use splines to draw curved shapes, which cannot be drawn as a polyline.

Drawing splines

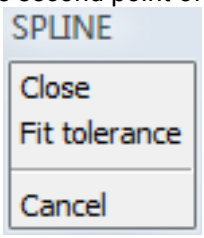
1. Do one of the following

   - Click the **Spline** tool button ( ) on the Draw toolbar.
   - Choose **Spline** in the Draw menu.
   - Type **spline** in the command bar, then press Enter.

   You are prompted: First point of spline:

2. Specify the first point of the spline.

3. Define the second point of the spline. You are prompted: Close/Fit Tolerance/<next point>: A prompt menu displays.



4. (option) Repeat step 3 to define more points.

5. Right click to stop adding more points. You are prompted: Select starting tangent point: A line that is tangent to the spline displays dynamically from the start point of the spline. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

6. Click to define the curve of the spline. You are prompted: Select ending tangent point: A line that is tangent to the spline displays dynamically from the start end of the spline. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h

7. Click to conclude the spline command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/s

8. (option) Right click to restart.

Figure 1.638

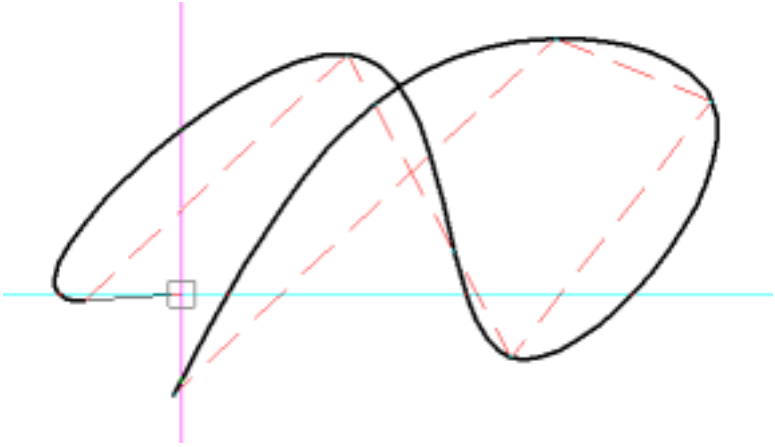


Figure 1.639

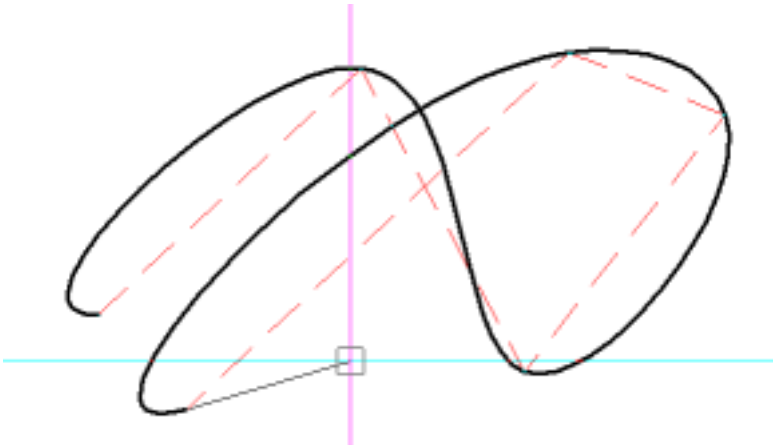
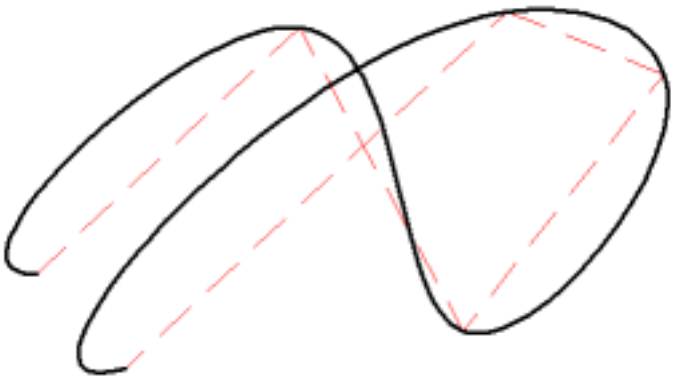


Figure 1.640



NOTES

- The red dashed lines in the above images represent the polyline through the control points of the spline.
- The **SPLFRAME** (Spline Frame) settings variable defines whether the control polygon displays for splines and spline-fit polylines.

1.17.14 Freehand sketches

Command: SKETCH (6.27.34) A freehand sketch consists of a series of straight line segments, created either as individual line entities or as a polyline. Before you begin creating a freehand sketch, you must set the length, or increment, of each segment. The smaller the segments, the more accurate your sketch, but small segments increase the file size. After you specify the length of the sketch segments, the crosshairs change to a pencil tool (). A freehand sketch line is not added to the drawing until you "write" the sketch into your drawing.

Setting the Sketch Poly (SKPOLY) system variable

Whether a chain of individual lines or a polyline is created by the **Sketch** tool is controlled through the **Sketch Poly** (SKPOLY) system variable. When the setting is **ON** the **Sketch** tool creates polylines Do one of the following:

- In the command bar type **skpoly** , then press Enter.
- In the Settings (1.13.11) dialog, go to **Drawing > Drafting > Entity Creation > Freehand Sketches**.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntit

Figure 1.641



Creating a freehand sketch

1. Do one of the following:

   - Click the **Sketch** tool button ( ) on the Draw toolbar.
   - Choose **Sketch** in the Draw menu.
   - Type **sketch** in the command bar, then press Enter.

   You are prompted: Length of segments <current length>:

2. Do one of the following to define the length of the segments:

   - Press Enter to accept the current length.
   - Type a new value in the command bar.
   - Click two points to define the length graphically.

   The length of the segments is saved to the **Sketch Increment** (SKETCHINC) system variable.

3. Click to start sketching (pen down). The crosshairs are replaced by the **Pencil** tool ( ) You are prompted: Press ENTER or click to toggle Pen/Quit without save/Record (save)/Erase/Connect/Straight/eXit with save:: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Drav

4. Move the cursor to sketch. The sketched line displays in light green.

5. (option) Click to stop sketching (pen up), then click again (pen down) to resume.

6. (option) Choose **Erase** on in the prompt menu or type **E** + Enter in the to start erasing sketch lines. The cursor changes to the **Erase** tool ( ).

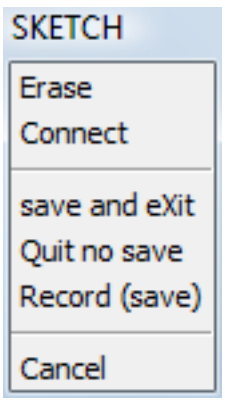
   - Move the cursor to the end of a sketch line to start erasing.
   - Click to stop erasing and resume sketching.

7. (option) Choose **Record (save)** in the prompt menu or type **R** + Enter to save the sketch lines, and keep sketching.

8. (option) Choose **save and eXit** in the prompt menu or type **X** + Enter to save the sketch lines and exit the sketch command.

9. (option) Choose **Quit no save** in the prompt menu or type **Q** + Enter to exit the sketch command without saving.

Figure 1.642



1.17.15 Circles

Command: CIRCLE (6.9.20), AI_CIRCTAN (6.5.1) The default method for drawing a circle is to specify a center point and radius. Other methods to draw circles can be found in the **Circle** submenu of the Draw menu and in the **Circles** toolbar (1.12.3). The **Ai_CircTan** command draws circles tangent to three entities.

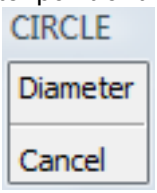
General procedure to draw a circle

1. Do one of the following.

   - Click the **Circle** tool button ( ) on the Draw toolbar.
   - Type **circle** in the command bar, then press Enter.
   - Type **C** in the command bar, then press Enter.

   You are prompted: 2Point/3Point/RadTanTan/Arc/Multiple/<Center of circle>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntiti

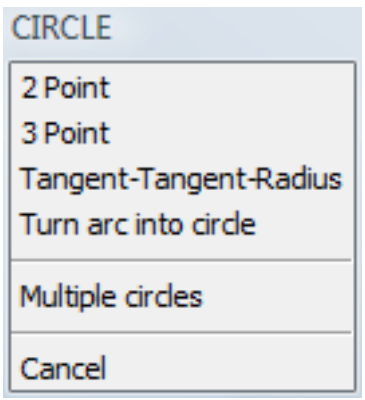
2. Specify the center point of the circle. You are prompted: Diameter/<Radius> <current radius>: The prompt menu changes:



3. Do one of the following:

   - Press Enter to accept the current radius.
   - Type a new radius and press Enter.
   - Click to define the radius graphically.

Figure 1.643



Drawing a circle tangent to three entities

1. Type **ai_circtan** in the command bar. You are prompted: Enter Tangent spec:
2. Click a linear entity. Lines, arcs, circles, polyline segments, xlines and rays are accepted. You are prompted: Enter second Tangent spec:
3. Click the second linear entity. You are prompted: Enter third Tangent spec:
4. Click the third linear entity. A circle tangent to the three selected entities is created.

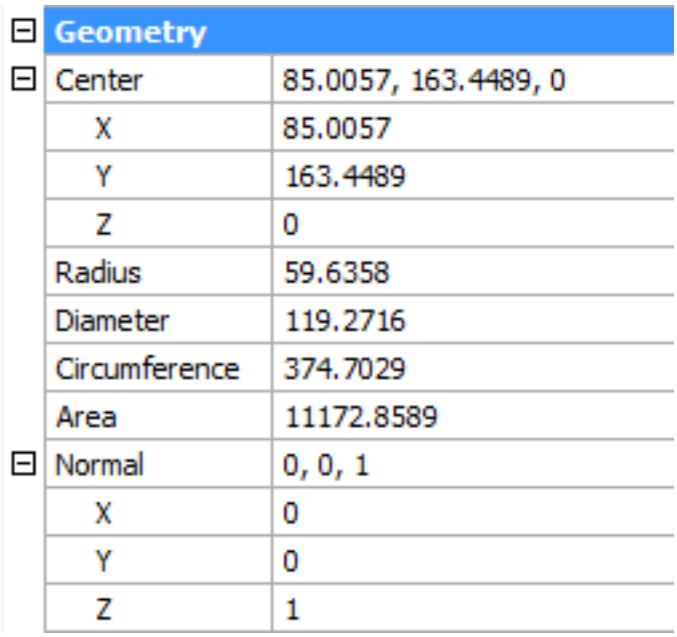
Editing a circle

1. Select the circle. The properties of the circle display in the Properties bar (1.12.4). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntities/ART/07Circle

2. Type a new value in the **Center** , **Radius** , **Diameter** , **Circumference** or **Area** field, then press Enter.

3. Press the Esc key to stop editing the circle.

Figure 1.644



Geometry	
Center	85.0057, 163.4489, 0
X	85.0057
Y	163.4489
Z	0
Radius	59.6358
Diameter	119.2716
Circumference	374.7029
Area	11172.8589
Normal	0, 0, 1
X	0
Y	0
Z	1

1.17.16 Arcs

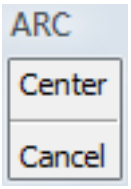
Command: ARC (6.4.12) The default method for drawing circular arcs is to specify three points: the start point, a second point and the endpoint. Other methods to draw arcs can be found in the Arc submenu of the Draw menu and in the Arcs toolbar (1.12.3).

General procedure to draw an arc

1. Do one of the following.

   - Click the **Arc** tool button ( ) on the Draw toolbar.
   - Type **arc** in the command bar, then press Enter.
   - Type **A** in the command bar, then press Enter.

   You are prompted: Enter to use last point/Center/Follow/<Start of arc>: A prompt menu displays:

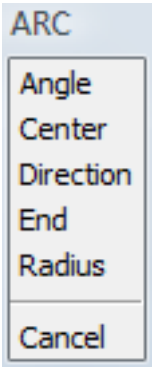


2. Specify the start point of the arc. You are prompted: Angle/Center/Direction/End/Radius/<Second point>: The prompt menu changes: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10D

3. Specify the second point of the arc. You are prompted: End point:

4. Specify the end point of the arc.

Figure 1.645



Editing an arc

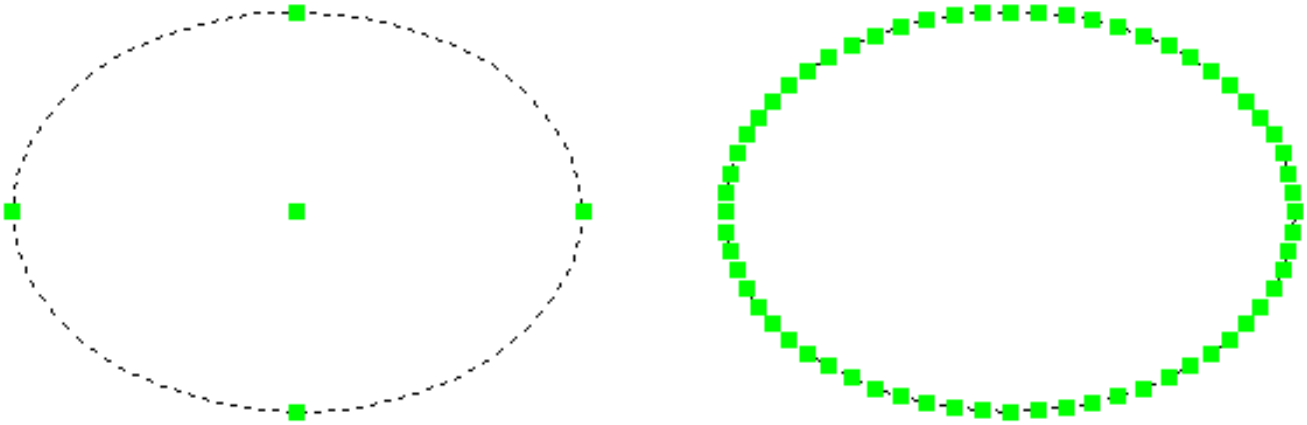
1. Select the arc. The properties of the arc display in the Properties bar (1.12.4). (\vref {fig:C:/Dev/Task-2448/images/static

2. Type a new value in the **Center** , **Radius** , **Start angle** or **End angle** , then press Enter.
3. Press the Esc key to stop editing the arc.

Figure 1.646

Geometry	
Start point	38.5381, 191.1854, 0
X	38.5381
Y	191.1854
Z	0
Center	3.7399, 153.1902, 0
X	3.7399
Y	153.1902
Z	0
End point	-47.553, 158.0457, 0
X	-47.553
Y	158.0457
Z	0
Radius	51.5223
Start angle	48
End angle	175
Total angle	127
Arc length	114.2725
Area	1884.8676
Normal	0, 0, 1
X	0
Y	0
Z	1

Figure 1.647



1.17.17 Ellipses

Command: ELLIPSE (6.13.5) The default method for drawing an ellipse is to specify the endpoints of one axis of the ellipse and then specify a distance representing half the length of the second axis. The endpoints of the first axis determine the orientation of the ellipse. The longer axis of the ellipse is called the major axis, and the shorter one is the minor axis. The order in which you define the axes does not matter. The program determines the major and minor axes based on their relative lengths. Half the major axis is called the major radius, half the minor axis is the minor radius. The ratio (minor radius divided by the major radius) defines the eccentricity of the ellipse. A circle is an ellipse with an eccentricity of 1. Other methods to draw ellipses can be found in the **Ellipse** submenu of the Draw menu and in the Ellipses toolbar. The **Polyline Ellipse** (PELLIPSE) settings variable: controls whether real ellipses or a polyline representation is created. If the variable is ON, the **Ellipse** tool creates a closed polyline, if the variable is OFF real ellipses are created. The difference between a real ellipse and a polyline representation of an ellipse is visible when you select the ellipse. A real ellipse has five handles: center point and the endpoints of the axes. A polyline representation of an ellipse is a closed polyline composed of arc segments. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntities/ART/09EllipsesTyp real ellipse (left) and the polyline representation of an ellipse (right) **Note:** Since ellipses cannot be used as a clip boundary for **external references** or to create clipped paperspace viewports (1.16.14), you must use a polyline representation to do these jobs.

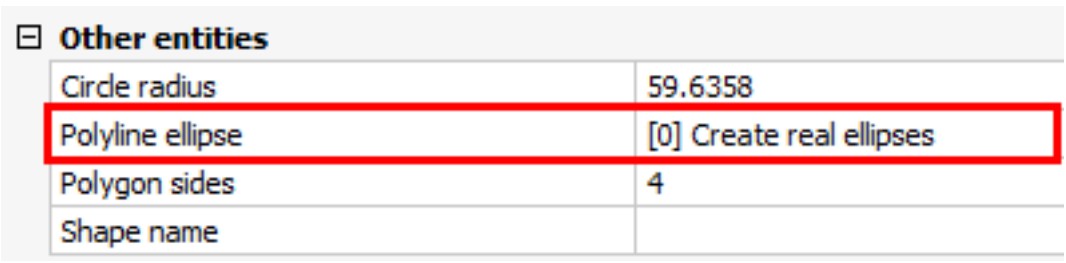
Setting the Polyline Ellipse (PELLIPSE) system variable

Do one of the following:

- In the command bar type **pellipse**, then press Enter.
- In the Settings (1.13.11) dialog, go to **Drawing > Drafting > Entity Creation > Other entities** .

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntit

Figure 1.648

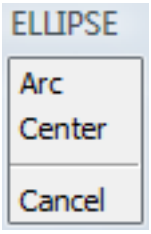


General procedure to draw an ellipse

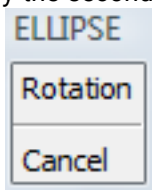
1. Do one of the following.

   - Click the **Ellipse** tool button ( ) on the Draw toolbar.
   - Type **ellipse** in the command bar, then press Enter.
   - Type **EL** in the command bar, then press Enter.

   You are prompted: Arc/Center/<First end of ellipse axis>: A prompt menu displays:



2. Specify the the first end of the ellipse axis. You are prompted: Second end of axis:
3. Specify the second end of the ellipse axis. You are prompted: Rotations/<Other axis>: A prompt menu displays:



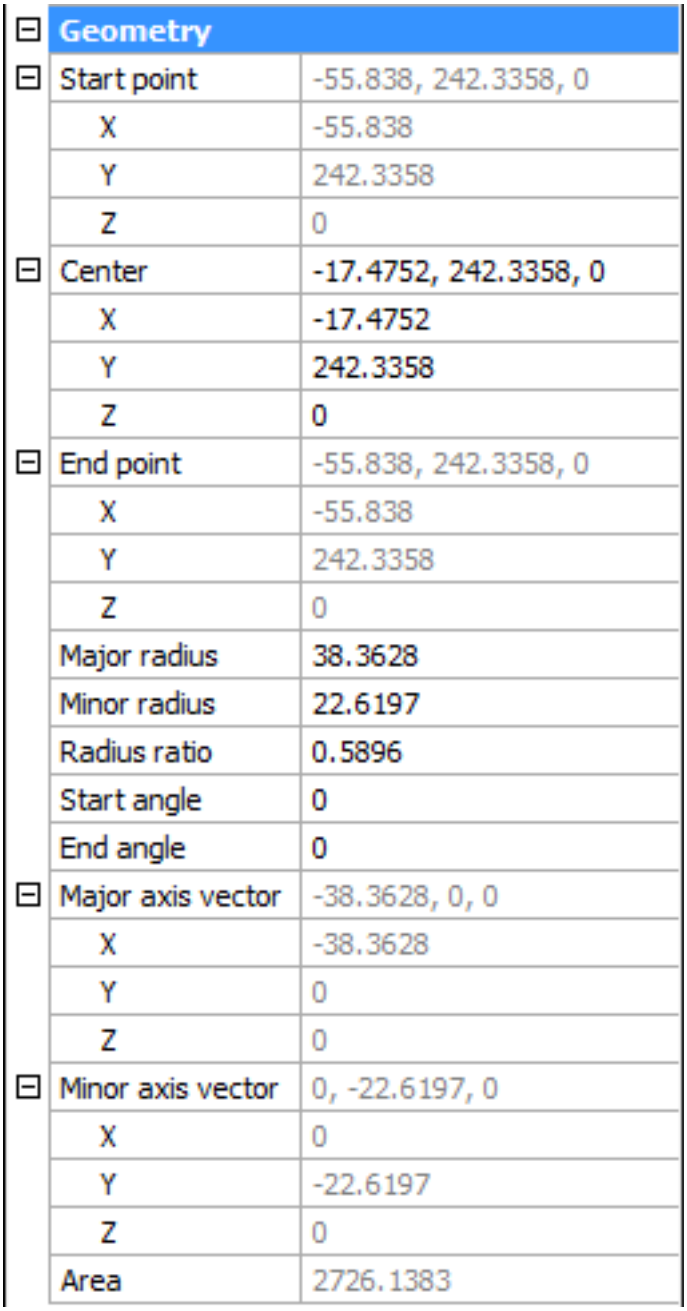
4. Specify the length of half the other axis of the ellipse.

Note: If the **Polyline Ellipse** variable is off, the **Arc** option is not available.

Editing an ellipse

1. Select the ellipse. The properties of the ellipse display in the Properties bar (1.12.4). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntities/ART/09Ellips
2. Type a new value in the **Center**, **Major radius**, **Minor radius**, **Radius ratio**, **Start angle** or **End angle** field, then press Enter.
3. Press the Esc key to stop editing the ellipse.

Figure 1.649



Geometry	
Start point	-55.838, 242.3358, 0
X	-55.838
Y	242.3358
Z	0
Center	-17.4752, 242.3358, 0
X	-17.4752
Y	242.3358
Z	0
End point	-55.838, 242.3358, 0
X	-55.838
Y	242.3358
Z	0
Major radius	38.3628
Minor radius	22.6197
Radius ratio	0.5896
Start angle	0
End angle	0
Major axis vector	-38.3628, 0, 0
X	-38.3628
Y	0
Z	0
Minor axis vector	0, -22.6197, 0
X	0
Y	-22.6197
Z	0
Area	2726.1383

1.17.18 Elliptical Arcs

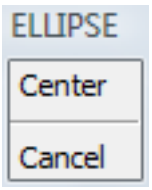
Command: ELLIPSE (6.13.5) + A An elliptical arc is a portion of an ellipse. The default method for drawing an elliptical arc is to specify the endpoints of one axis of the ellipse, and then specify a distance representing half the length of the second axis. Then you specify the start and end angles for the arc, measured from the center of the ellipse in relation to its major axis. Other methods to draw elliptical arcs can be found in the Ellipse submenu of the Draw menu and in the Elliptical Arcs toolbar.

General procedure to draw an elliptical arc

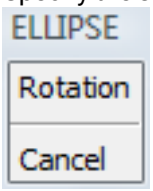
1. Do one of the following:

   - Click the **Elliptical Arc** tool button ( ) on the Draw toolbar.
   - Launch the Ellipse (1.17.17) command using the **Arc** option.

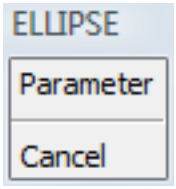
   You are prompted: Center/<First end of ellipse axis>: A prompt menu displays:

2. Specify the first end or the ellipse axis. You are prompted: Second end of axis:
3. Specify the second end of the ellipse axis. You are prompted: Rotation/<Other axis> A prompt menu displays:

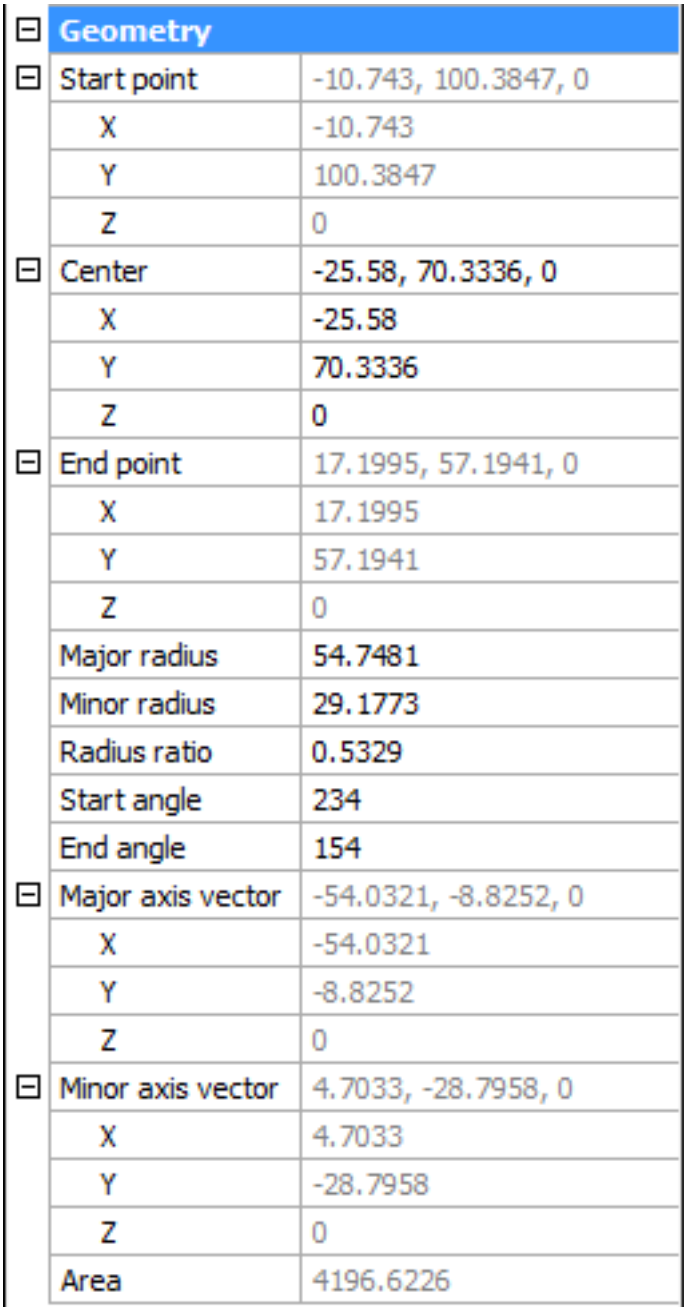
   

4. Specify the length of half the other axis of the ellipse. You are prompted: Parameter/<Start angle of arc>: The prompt menu changes:

5. Specify the start angle of the elliptical arc. The elliptical arc is drawn dynamically when you move the crosshairs. You are prompted: Parameter/Included/<End angle>: The prompt menu changes: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntities/ART/10Ellipti
6. Specify the end angle of the elliptical arc.

Figure 1.650



Geometry	
Start point	-10.743, 100.3847, 0
X	-10.743
Y	100.3847
Z	0
Center	-25.58, 70.3336, 0
X	-25.58
Y	70.3336
Z	0
End point	17.1995, 57.1941, 0
X	17.1995
Y	57.1941
Z	0
Major radius	54.7481
Minor radius	29.1773
Radius ratio	0.5329
Start angle	234
End angle	154
Major axis vector	-54.0321, -8.8252, 0
X	-54.0321
Y	-8.8252
Z	0
Minor axis vector	4.7033, -28.7958, 0
X	4.7033
Y	-28.7958
Z	0
Area	4196.6226

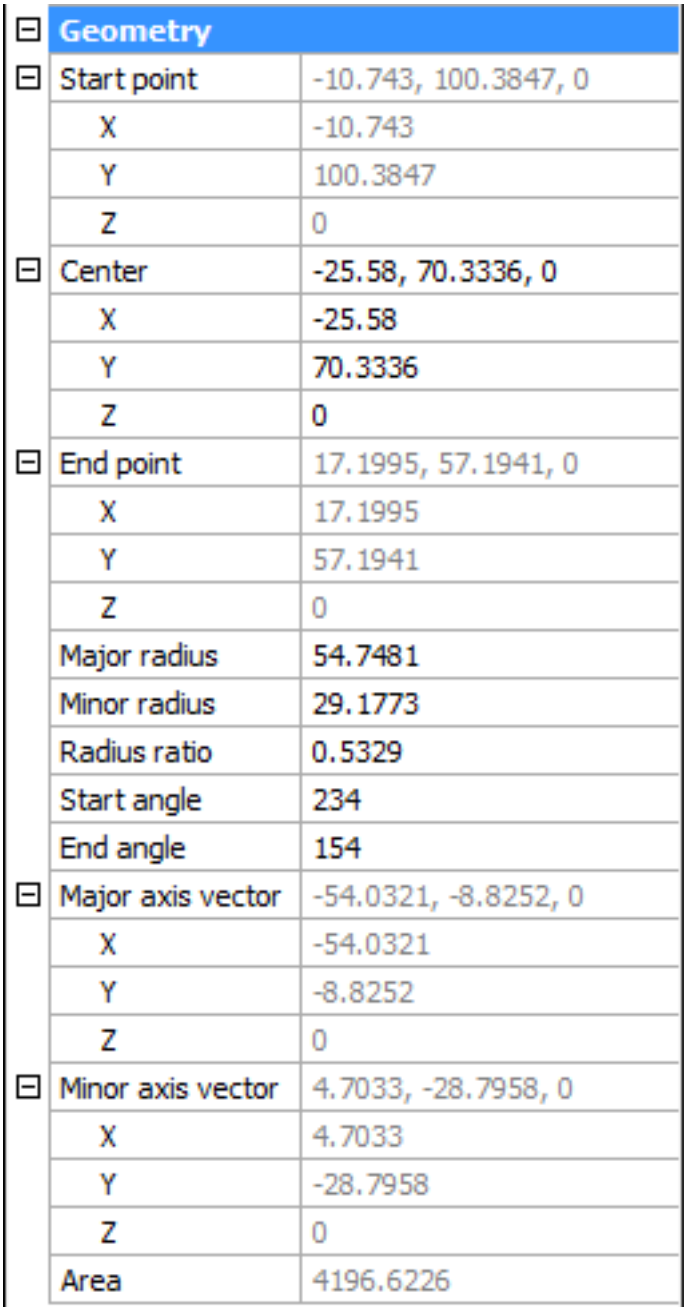
Editing an elliptical arc

1. Select the elliptical arc. The properties of the elliptical arc display in the Properties bar (1.12.4). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntiti

2. Type a new value in the **Center**, **Major radius**, **Minor radius**, **Radius ratio**, **Start angle** or **End angle** field, then press Enter.

3. Press the Esc key to stop editing the elliptical arc.

Figure 1.651



Geometry	
Start point	-10.743, 100.3847, 0
X	-10.743
Y	100.3847
Z	0
Center	-25.58, 70.3336, 0
X	-25.58
Y	70.3336
Z	0
End point	17.1995, 57.1941, 0
X	17.1995
Y	57.1941
Z	0
Major radius	54.7481
Minor radius	29.1773
Radius ratio	0.5329
Start angle	234
End angle	154
Major axis vector	-54.0321, -8.8252, 0
X	-54.0321
Y	-8.8252
Z	0
Minor axis vector	4.7033, -28.7958, 0
X	4.7033
Y	-28.7958
Z	0
Area	4196.6226

1.17.19 Rectangles

Command: RECTANG (6.26.6) A rectangle is created as a closed, four sided polyline (1.17.12). A rectangle is drawn by specifying two opposite corners. The **Dimensions** option allows to specify the width and height of the rectangle. Unless the **Rotated** option is chosen, the sides of a rectangle are always parallel to the x-axis and y-axis of the current coordinate system (1.15.18). **Note:**When Fill Mode (1.17.3) is turned off, all filled entities, such as wide polylines (1.17.12) and solids (1.17.23), display and print as outlines.

Drawing rectangles

General Procedure

1. Do one of the following

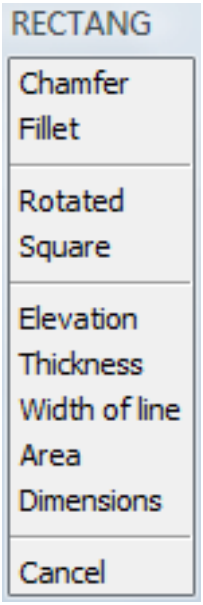
   - Click the **Rectangle** tool button () on the Draw toolbar.
   - Choose **Rectangle** in the Draw menu.
   - Type **rectang** in the command bar, then press Enter.
   - Type **rec** in the command bar, then press Enter.

   You are prompted: Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/Area/Dimensions<select first corner of rectangle>: A prompt menu opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Usr

2. Specify the first corner of the rectangle. The rectangle displays dynamically when you move the cursor.
3. Specify the opposite corner of the rectangle. The rectangle is created.
4. (option) Right click to draw more rectangles.

NOTE	Because a rectangle is created as a closed polyline, rectangles have a direction. The direction of a closed polyline is positive if it is drawn counter-clockwise and negative if it is drawn in a clockwise direction. Rectangles have a counter-clockwise direction if both the X- and Y-coordinate of the second point are higher or lower than the coordinates of the first point (= movement of the cursor is positive or negative along both X- and Y-axis). Rectangles have a clock-wise direction if the X- or Y-coordinate is higher or lower than the X- or Y-coordinate of the first point (= movement of the cursor is positive along the X-axis and negative along the Y-axis or vice versa). The **Reverse Direction** option of the **Edit Polyline** tool changes the direction of a polyline.

Figure 1.652

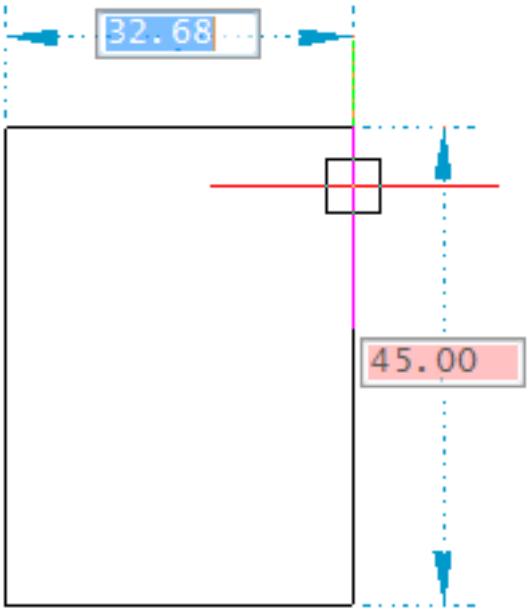


Using Dynamic Dimensions

Using Dynamic Dimensions (1.15.9) the width and length of the rectangle can be controlled dynamically.

1. Launch the **Rectang** command. You are prompted: Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/Area/Di
first corner of rectangle>:
2. Specify the first corner of the rectangle. The rectangle displays dynamically when you move the cursor. The width and height display in the dynamic dimension fields.
3. Type a value in the **Height** field.
4. Press the TAB key. The **Height** field is locked and turns red.
5. Type a value the **Width** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10D
6. Do one of the following:
   - Press Enter to create the rectangle.
   - Press the TAB key until the desired field is active, then type a different value.
7. Press Enter to create the rectangle.

Figure 1.653



1.17.20 Polygons

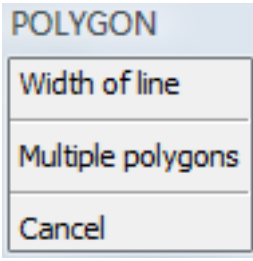
Command: POLYGON (6.24.43) Polygons are closed polylines (1.17.12) comprised of a minimum of three and a maximum of 1,024 equal-length sides. In BricsCAD you can draw polygons using the following methods:

- **Centre - Vertex** ( ): first define the center point, then the vertex (= radius of the circumscribed circle method).
- **Center - Side** ( ): first define the center point, then the midpoint of a side (= radius of the inscribed circle method).
- **Edge** ( ): define the length of the side of the polygon.

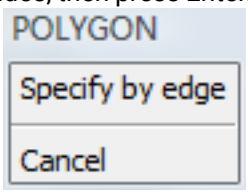
Drawing polygons

1. Do one of the following:
   - Click the **Polygon, Center - Vertex** tool button ( ) on the **Polygons** flyout of Draw toolbar.
   - Choose **Polygon** in the Draw menu.
   - Type **polygon** in the command bar, then press Enter.

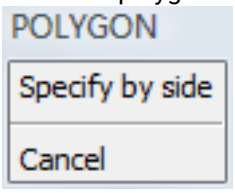
   You are prompted: Polygon: Multiple/Width of line/<Number of sides> <4>: A prompt menu opens:



2. Specify the number of sides, then press Enter. **You are prompted:** Specify by: Edge/<Center of polygon>: The prompt menu changes:



3. Specify the center of the polygon. **You are prompted:** Specify by: Side/<Select vertex point>: the prompt menu changes:



4. Specify a vertex point of the polygon. The polygon is created.

Note: When Fill Mode (1.17.3) is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.

1.17.21 Traces

Command: TRACE (6.29.24) Traces are 4-sided filled entities, created with two parallel sides. When you create a series of traces, they are L-connected to each other.

Drawing traces

1. Do one of the following:

   - Click the **Trace** tool button () on the **Polygons** flyout of the Draw toolbar.
   - Choose **Trace** in the Draw menu.
   - Type **trace** in the command bar, then press Enter.

   You are prompted: Width of trace <current width>:

2. Do one of the following:

   - Right click to accept the current width.
   - Type a new current width in the command bar.
   - Define a new current width graphically by clicking two points.

   You are prompted: Start of trace:

3. Specify the start point of the trace. **You are prompted:** Next point:

4. Specify the end point of the first trace. A line indicates the direction of the first trace. The direction of the second trace displays dynamically.

5. Do one of the following:
   - Right click to create the trace.
   - Specify the endpoint of the next trace.

6. (option) Repeat step 5 to keep adding traces.

Note: When Fill Mode (1.17.3) is turned off, all filled entities, such as traces, display and print as outlines.

1.17.22 Donuts

Command: DONUT (6.10.35) Donuts are solid, filled circles or rings created as closed, 2-segment, wide polylines. The default method to draw donuts is to specify its inside and outside diameters, and then specify its center. If the inside diameter is zero, a filled circle is created.

Drawing donuts

1. Do one of the following:

   - Click the **Donut** tool button ( ) Draw toolbar.
   - Choose **Donut** in the Draw menu.
   - Type **donut** in the command bar, then press Enter.

   You are prompted: 2Point/3Point/RadTanTan/<Inside diameter of donut> <current inside diameter>: A prompt menu opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities

2. Do one of the following:

   - Right click to accept the current inside diameter.
   - Type a new inside diameter in the command bar.
   - Define a new inside diameter graphically by clicking two points.

   You are prompted: Outside diameter of donut <current outside diameter>:

3. Do one of the following:

   - Right click to accept the current outside diameter.
   - Type a new outside diameter in the command bar.
   - Define a new outside diameter graphically by clicking two points.

   You are prompted: Center of donut:

4. Specify the center of the donut. The donut is created.

5. (option) Specify the center point of another donut.

6. Right click to conclude the **donut** command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Usr

NOTES

- When Fill Mode (1.17.3) is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.
- Set the inside diameter to 0 (zero) to create filled circles.

Figure 1.654

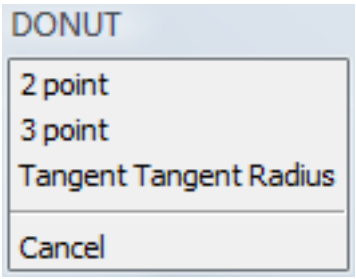


Figure 1.655

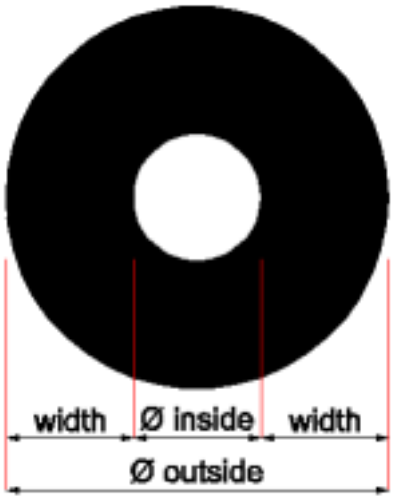
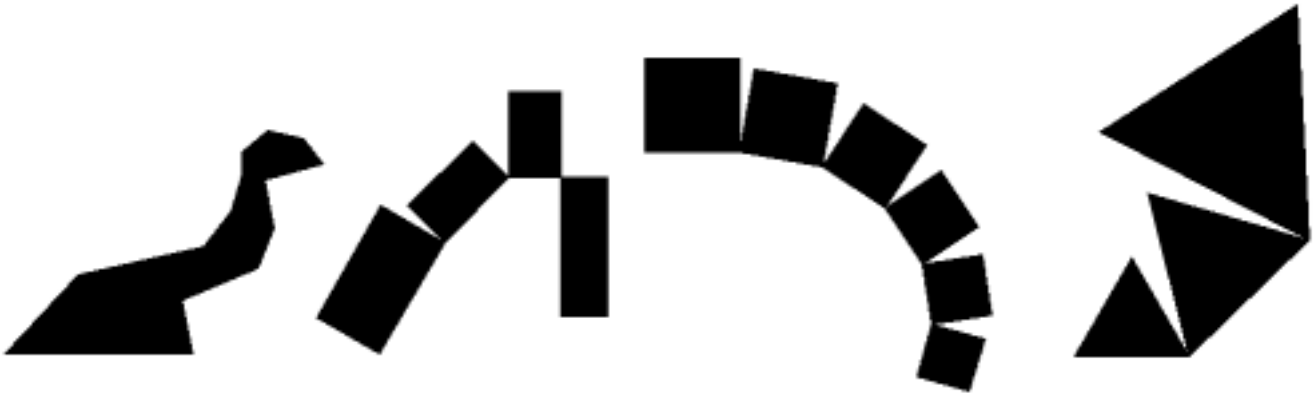


Figure 1.656



1.17.23 Solids

Command: SOLID (6.27.37) 2D Solids are rectangular, square, triangular or quadrilateral areas filled with a solid color. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2
Left to right: quadrilateral, rectangular, square and triangular solids

Drawing solids

1. Do one of the following:

   - Click the **Solid** tool button ( ) on the **Polygons** flyout of **Draw** toolbar.
   - Choose **Solid** in the **Draw** menu.
   - Type **solid** in the command bar, then press Enter.

   You are prompted: Rectangle/Square/Triangle/<First point of solid>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntities/ART/15solidp

2. Specify the first point of the solid.
3. Specify the second point of the solid.
4. Specify the third and fourth point of the solid. The solid is created.
5. (option) Repeat step 4 to add more solids. A series of connected solids is created.
6. Right click to conclude.

Note: When Fill Mode (1.17.3) is turned off, all filled entities, such as wide polylines and solids, display and print as outlines.

Figure 1.657



1.17.24 Revision Clouds

Command: REVCLOUD (6.26.29) A **Revision Cloud** is an open or closed polyline (1.17.12) composed of arcs. Revisions clouds are used to for reviewing or redlining purposes to indicate the parts of the drawing that need to be adjusted or annotated. Before you start drawing revision clouds, first define the **Arc length** and **Style** .

Drawing revision clouds

1. Do one of the following:

   - Click the **Revision Cloud** tool button ( ) on the Draw toolbar.
   - Choose **Revision Cloud** in the Draw menu.
   - Type **revision cloud** in the command bar, then press Enter.

   You are prompted: Specify start point or [Arc length/Object/Style] <Object>: A prompt menu opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntiti

2. (option) To define the **Arc length** , do one of the following:

   - Choose **Arc length** in the prompt menu.
   - Type **A** in the command bar, then press Enter.

   You are prompted: Specify minimum length of arc <current minimum length>: Do one of the following:

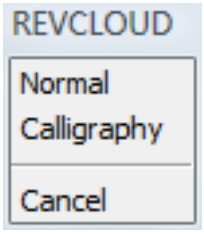
   - Press Enter or right click to accept the current minimum length.
   - Type a new value in the command bar, then press Enter.
   - Define a new minimum length graphically by specifying two points.

   You are prompted: Specify maximum length of arc <current maximum length>: Do one of the following:

   - Press Enter or right click to maximum length equal to the minimum length.
   - Type a new value in the command bar, then press Enter.
   - Define a new maximum length graphically by specifying two points.

3. (option) To define the revision cloud **Style** , do one of the following:

   - Choose **Style** in the prompt menu.
   - Type **S** in the command bar, then press Enter.

   You are prompted: Select arc style [Normal/Calligraphy] <current style>: A prompt menu opens: Do one of the following:

   - Press Enter or right click to accept the current style
   - Select a new style in the prompt menu.
   - Type **N** or **C** in the command bar, then press Enter.

4. (option) To convert an existing polyline, arc or circle to a revision cloud, do one of the following:

Figure 1.658

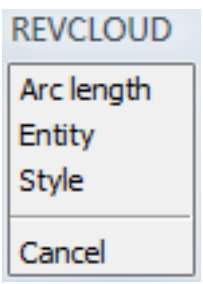


Figure 1.659

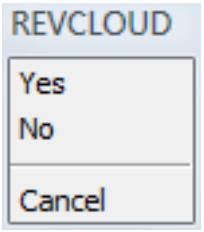


- Press Enter, then select the entity you want to convert.
- Choose Object in the prompt box, then select the entity you want to convert.

5. (option) To draw the revision cloud, click where you want to start and move the cursor. Each time the cursor movement exceeds the minimum length, an arc is added. To stop do one of the following:

   - Move the cursor over the start point to close the revision cloud.
   - Right click.

   You are prompted: Reverse direction [Yes/No] <No>: A prompt menu opens:



6. To create the revision cloud, do one of the following:

   - To accept the revision cloud, right click.
   - To reverse the direction, type **Y** or choose **Yes** in the prompt menu.
   - To leave the command without creating the revision cloud, choose **Cancel** in the prompt menu or press the Esc key.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2I
Revision Clouds: 1. Normal 2. Calligraphy 3. Reversed calligraphy (based on a rectangle)

1.17.25 Points

Command: POINT (6.24.31) Since points are dimensionless, a single pixel should be the (nearly) correct representation on the screen. But this is hardly visible, especially in a complex drawing. Therefore you can choose between a number of possible display styles. The point display style is chosen by means of the **Point Display Mode** (PDMODE) system variable. The size of the point representation style is controlled through the **Point Display Size** (PDSIZE) system variable.

Defining the point display settings

1. Open the Settings (1.13.11) dialog.
2. In the **Drawing** settings category, expand the **Entity Creation** sub-category.
3. Expand the **Points** settings group, then expand the **Point display mode** setting. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2DEntities/ART/17pointc (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/01Drawing2I Point display modes.
4. Set the **Point Display Mode** setting:
   - Choose the **point location marker** style: **point** , **none** , **X** or |
   - Choose to add a **circle** and/or a **square** .
5. Set the **Point Display Size** setting.
   - Relative: Type 0 (zero) to scale the point display style at 5% of the screen.
   - Absolute: Type the size of the point display style in drawing units.
6. Close the Settings dialog.

Note: If you need to draw a 'dot', use a Donut (1.17.22) with a zero inside diameter instead of a point entity.

Figure 1.660

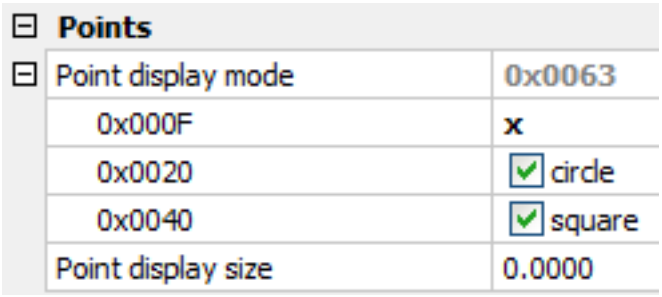
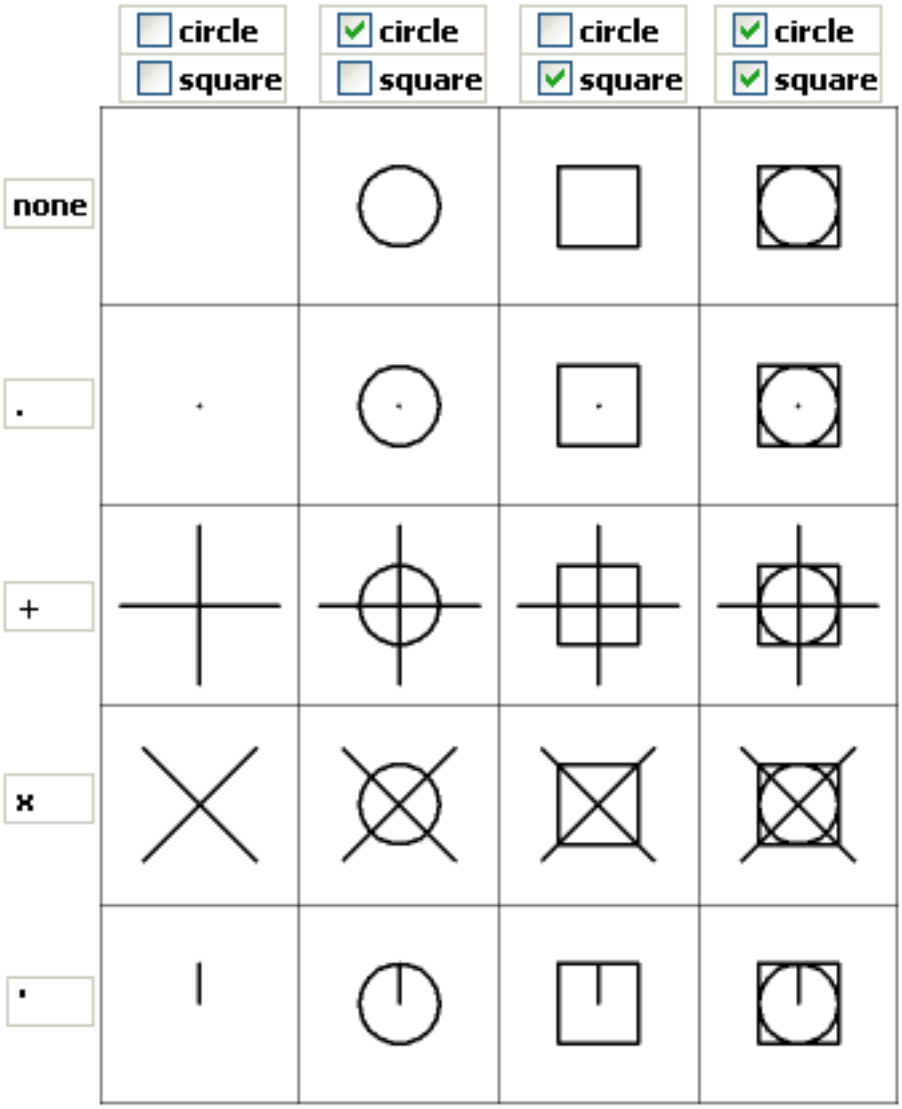


Figure 1.661

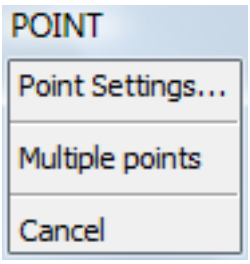

circle
square
circle
square
circle
square
circle
square
none
.
+
x
'

Drawing points

1. Do one of the following

   - Click the **Point** tool button (°) on the Draw toolbar.
   - Choose **Point** in the Draw menu.
   - Type **point** in the command bar, then press Enter.

   You are prompted: Settings/Multiple/<Location of point>: A prompt menu displays:



2. Specify the location of the point. The point is created.

Note: If the **Point Display Size** setting is set to 0 (zero), use the Regen (1.16.4) tool to resize the display of the points to 5% of the screen size after zooming in or out.

1.17.26 Hatching Overview

Commands: HATCH (6.16.1), **-**HATCH (6.16.2), GRADIENT (6.15.17), HATCHEDIT (6.16.3), -HATCHEDIT (6.16.4)
When you add hatching to a drawing, BricsCAD fills entities or enclosed areas with a predefined pattern or lines. First you specify the hatch pattern and other options, and then you choose which entities or enclosed areas that you want to hatch.

NOTES

- Depending on the pattern, hatchings are memory intensive and might take a considerable amount of time to draw and display. To improve performance, add hatching as one of the last steps when you create a drawing, or insert hatches on a separate layer that you can freeze as you continue to work on your drawing. Or use the **FILLMODE** system variable to switch off the display of hatches, gradient fills and filled entities.
- The maximum number of dashes is controlled by the **MAXHATCH** settings variable. When the number of dashes is higher than the current MAXHATCH value, the hatch cannot be created. A warning box displays offering possible choices for subsequent actions: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sour
By default the maximum number of dashes is 100 000.

The predefined and user pattern files sit in the **Support** folder of the current user. The **MEASUREMENT** setting controls which predefined hatch pattern file will be used.

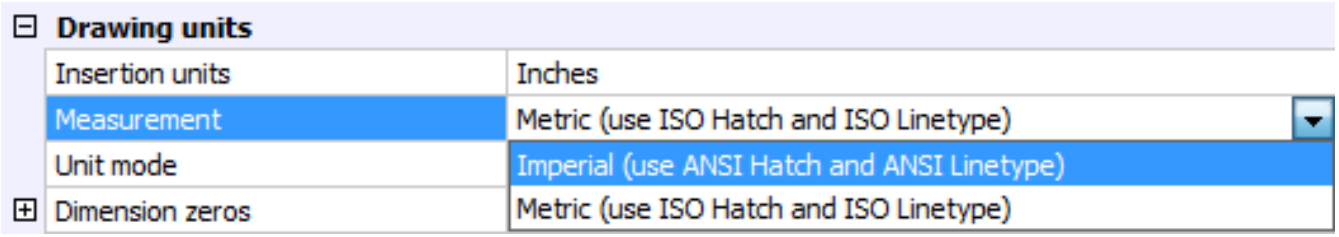
- **Imperial**: uses Default.pat
- **Metric**: uses Iso.pat

Defining the MEASUREMENT setting

Do one of the following:

- In the command bar type **measurement** , then press Enter. Type **ON** , then press Enter to set **measurement** to **Metric**. Type **OFF** , then press Enter to set **measurement** to **Imperial**.
- In the Settings (1.13.11) dialog go to **Drawing > Drafting > Drawing units.** Choose either **Metric** or **Imperial** in the combo box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntiti

Figure 1.662



Defining the MAXHATCH setting

Do one of the following:

- In the command bar type **maxhatch**, then press Enter. Type a new value and press enter.
- In the Settings (1.13.11) dialog, go to **Drawing > Drafting > Entity Creation > Hatches** . Type a new value in the Maximum hatch dashes settings field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/s

Figure 1.663

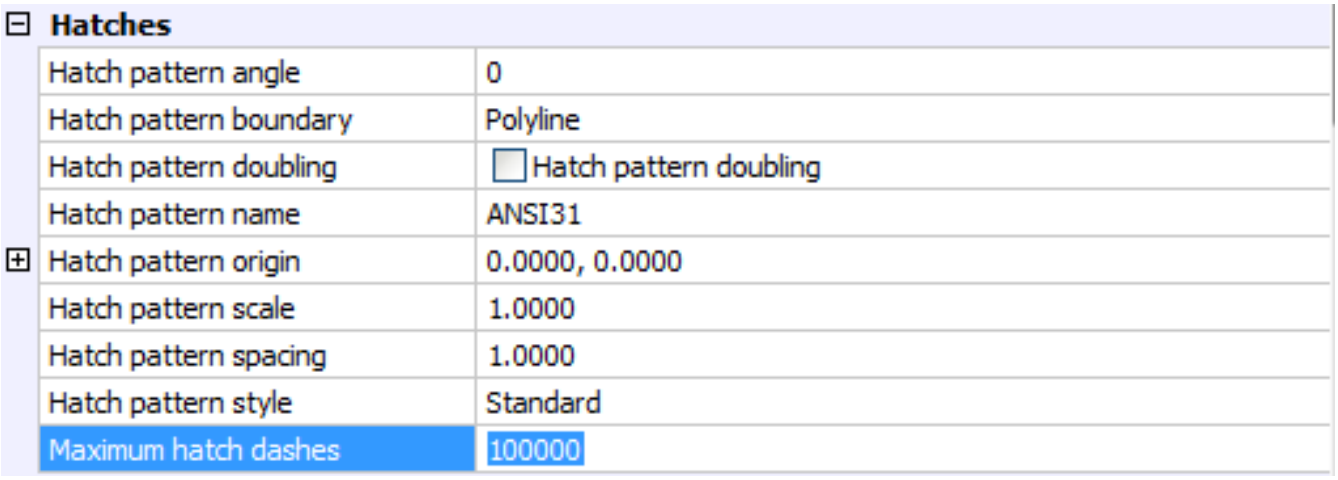
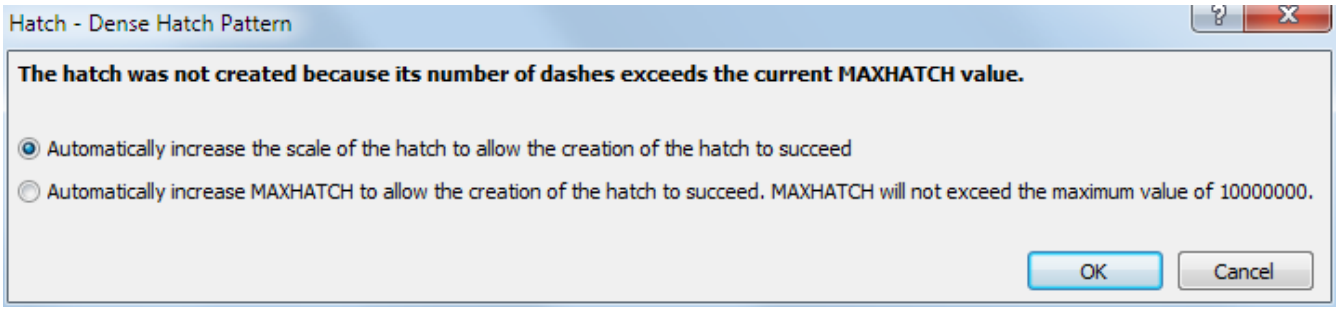


Figure 1.664


Hatch - Dense Hatch Pattern
The hatch was not created because its number of dashes exceeds the current MAXHATCH value.
Automatically increase the scale of the hatch to allow the creation of the hatch to succeed
Automatically increase MAXHATCH to allow the creation of the hatch to succeed. MAXHATCH will not exceed the maximum value of 10000000.
OK
Cancel

1.17.27 Wipeouts

Command: WIPEOUT (6.33.11) A **wipeout** is a polygonal area that hides the underlying objects using the current background color. Most commonly wipeouts are used to mask part of a drawing for clarity. For example, you may want to add text to a complicated part of a drawing. A wipeout could be used then to mask an area behind the text to increase the readability of the text. The **Frames** option of the **wipeout** command determines whether the edges of all wipeout entities in the current drawing are displayed or hidden.

To create a wipeout using points

1. Do one of the following:

   - Click the **Wipeout** tool button ( ) on the Draw toolbar.
   - Choose **Wipeout** in the Draw menu.
   - Type **wipeout** in the command bar, then press Enter.

   You are prompted: Frames/Polylines/<Start point>:

2. Specify the first vertex of the wipeout. **You are prompted:** Next point.

3. Specify the second vertex of the wipeout. **You are prompted:** Undo/<Next Point>:

4. Do one of the following:

   - Specify the third vertex of the wipeout.
   - Type **U** , then press Enter or choose **Undo** in the context menu to delete the previous vertex.

   You are prompted: Undo/Close/<Next Point>:

5. Do one of the following:
   - Specify the next vertex of the wipeout.
   - Type **U** , then press Enter or choose **Undo** in the context menu to delete the previous vertex.
   - Type **C** , then press Enter or choose **Close** in the context menu to create the wipeout.
   - Right click to create the wipeout.

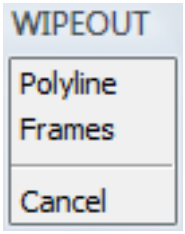
6. Repeat step 5 to create more vertices of the wipeout.

To create a wipeout from a polyline

1. Do one of the following:

   - Click the **Wipeout** tool button ( ) on the Draw toolbar.
   - Choose **Wipeout** in the Draw menu.
   - Type **wipeout** in the command bar, then press Enter.

   You are prompted: Frames/Polylines/<Start point>: The **Wipeout** context menu displays:



2. Do one of the following:

   - Choose **Polyline** in the context menu.
   - Type **P**, then press Enter.

   You are prompted: Select a closed polyline:

3. Click a closed polyline. Only polylines composed of line segments can be selected. You are prompted: Delete selected polyline? Yes/<No>:

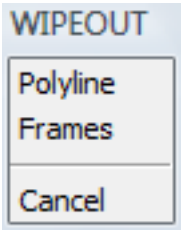
4. Do one of the following:
   - Type **Y** , then press Enter or choose **Yes** in the context menu to create the wipeout and delete the selected polyline.
   - Press Enter or choose **No** in the context menu, to create the wipeout and delete the selected polyline.

To control the display of the edges of wipeouts

1. Do one of the following:

   - Click the **Wipeout** tool button ( ) on the Draw toolbar.
   - Choose **Wipeout** in the Draw menu.
   - Type **wipeout** in the command bar, then press Enter.

   You are prompted: Frames/Polylines/<Start point>: The **Wipeout** context menu displays:



2. Do one of the following:

   - Choose **Frames** in the context menu.
   - Type **F** , then press Enter.

   You are prompted: Enter display mode: OFF/<ON>:

3. Do one of the following:
   - Choose **OFF** in the context menu or type **off**, then press Enter, to hide the edges of all wipeouts in the drawing.
   - Choose **ON** in the context menu or type **on**, then press Enter, to display the edges of all wipeouts in the drawing.

1.17.28 Creating Hatches and Gradient Fills

Commands: HATCH (6.16.1), -HATCH (6.16.2) and GRADIENT (6.15.17) The **Hatch** command fills an enclosed area in your drawing with a pattern. The **Gradient** command creates solid fills in nine different patterns and one or two colors. The area can either be a single entity such as a circle or a closed polyline or an area enclosed by a selection of entities. To create the hatch you can click inside the closed perimeter of a boundary or you can select entities.

General procedure to create hatches and gradient fills

1. Do one of the following:
   - Click the **Hatch** tool button ( ) or the **Gradient** tool button ( ) on the Draw toolbar.
   - Choose **Hatch...** or **Gradient...** in the Draw menu.
   - Type **hatch** or **gradient** in the command bar, then press enter.

   The Hatch and Gradient dialog displays.
2. Do one of the following:
   - Click the **Hatch** tab to create a hatch, then define the hatch properties.
   - Click the **Gradient** tab to create a gradient fill, then define the gradient fill properties.
3. Define the boundary for the hatch or gradient fill.
4. Set the options for the hatch or gradient fill.
5. Click the **OK** button to create the hatch or gradient fill.

To define hatch properties

1. On the Hatch and Gradient dialog click the Hatch tab. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
2. (option) Click the **Inherit properties** button ( ).
   - The **Hatch and Gradient** dialog closes temporarily. Prompts you: Select hatch entity to copy properties from:
   - Select a hatch entity. The **Hatch and Gradient** dialog reopens, displaying the properties of the selected hatch entity.
3. (option) Select the **Pattern Type** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source

   - User Defined - uses the pattern defined in this dialog box with the Angle, Spacing, and Cross Hatch options.
   - Predefined - uses patterns defined in **iso.pat** (metric units - MEASUREMENT (1.13.11) = ON) and **default.pat** (imperial units - MEASUREMENT = OFF).
   - Custom - uses patterns defined in *.pat files (one pattern definition per file).

   BricsCAD searches for *.pat files in the folders that are defined in the SRCHPATH (1.13.11) (Support File Search Path) system variable.
4. (option) If the **Pattern Type** is **Predefined** or **Custom** select a **Pattern Name** in the **Hatch pattern** palette, then double click the pattern or click the **OK** Button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/U
5. (option) If the **Pattern Type** is **Predefined** or **Custom**, do one of the following:
   - Type a scale in the **Scale** settings field.
   - Choose the scale from the drop down list.
6. (option) Do one of the following:
   - Type an angle in the **Angle** settings field.
   - Choose the angle from the drop down list.
7. (option) If the **Pattern Type** is **User defined** , specify the **Pattern Spacing**.
8. (option) If the **Pattern Type** is **User defined** , select the **Cross Hatch** option.

Figure 1.665

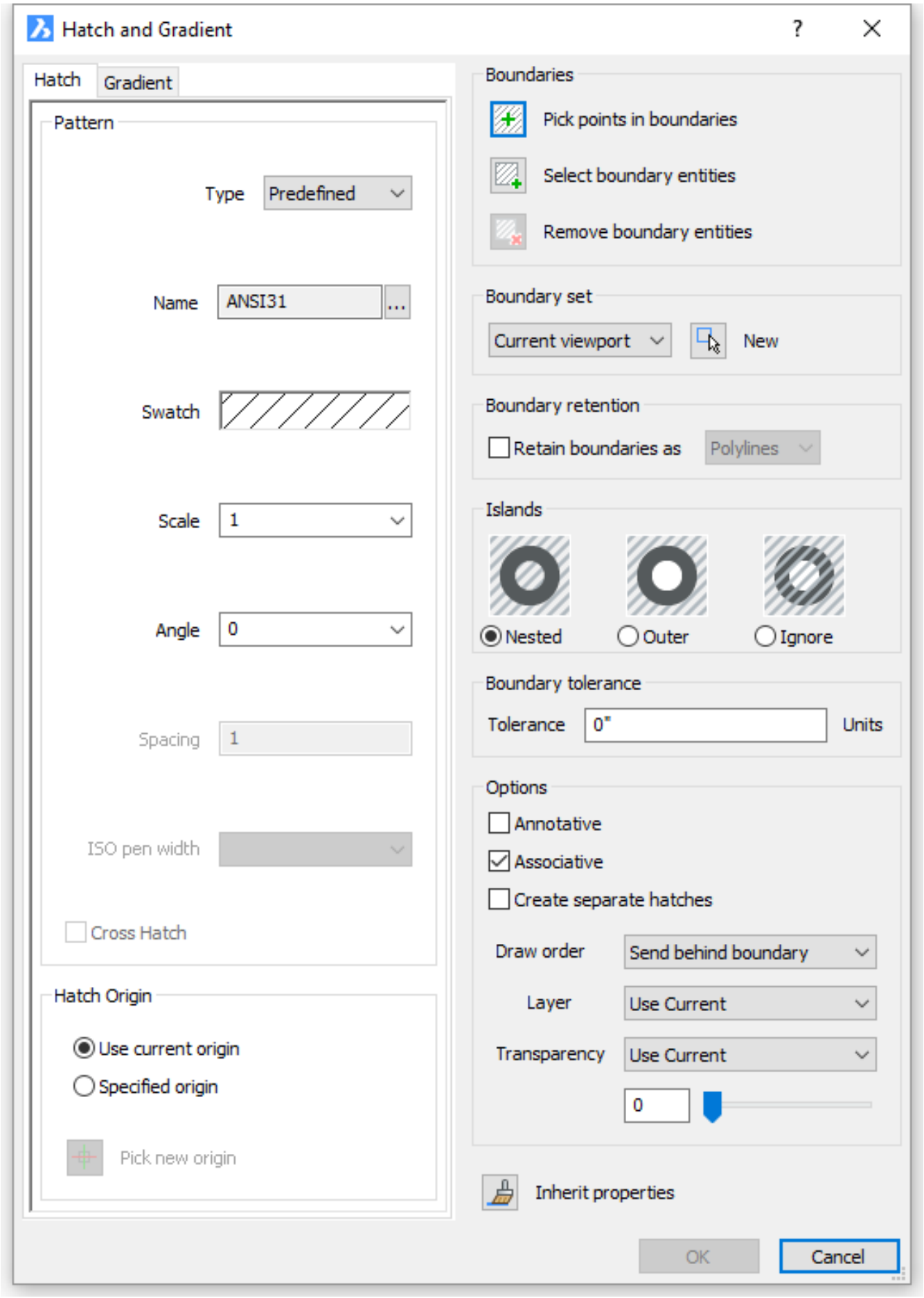

Hatch and Gradient
Hatch
Gradient
Pattern
Type
Predefined
Name
ANSI31
Swatch
Scale
1
Angle
0
Spacing
1
ISO pen width
Cross Hatch
Hatch Origin
Use current origin
Specified origin
Pick new origin
Boundaries
Pick points in boundaries
Select boundary entities
Remove boundary entities
Boundary set
Current viewport
New
Boundary retention
Retain boundaries as
Polylines
Islands
Nested
Outer
Ignore
Boundary tolerance
Tolerance
0"
Units
Options
Annotative
Associative
Create separate hatches
Draw order
Send behind boundary
Layer
Use Current
Transparency
Use Current
0
Inherit properties
OK
Cancel

Figure 1.666

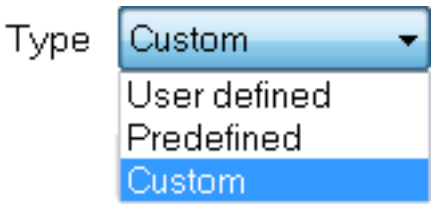
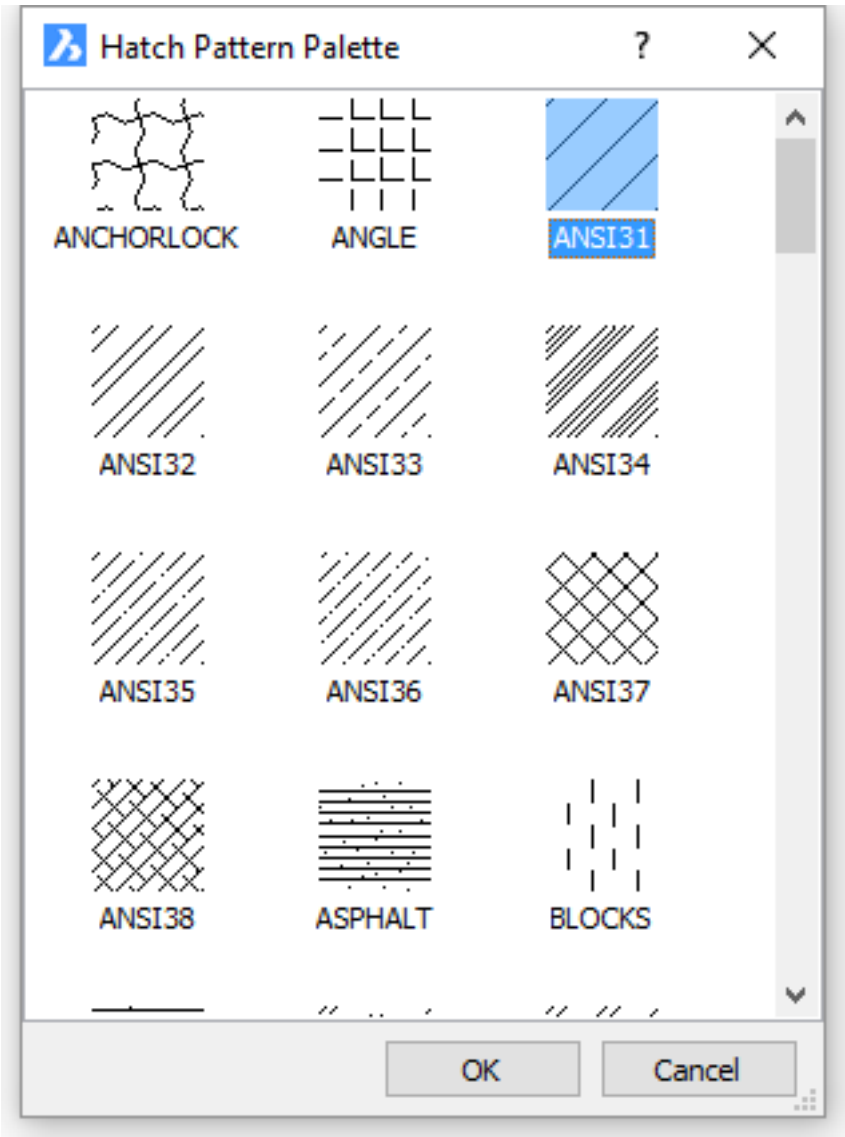


Figure 1.667



9. Define the Hatch Origin. The options are:
   - **Use current origin** : The origin as defined by the HPORIGIN (1.13.11) system variable is reused.
   - **Specified origin** : Click the **Pick new origin** button (), then pick a point in the drawing. The new origin is stored in the HPORIGIN (1.13.11) system variable.

Note: The BACKGROUNDCOLOR property of a hatch can be modified in the **Pattern / Background Color** field in the Properties (1.12.4) panel.

To define gradient fill properties

1. On the Hatch and Gradient dialog click the Gradient tab. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU

2. (option) Click the **One color** radio button to create a 'color to white' or 'color to black' gradient fill.
   - Click the colored tile to open the Select color dialog, to pick a different base color.
   - The color slider sets the transition from color to white or black. Press and hold the left mouse button to move the color slider ((\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Dra) . Move the slider to the far right to create a 'color to white' gradient fill. Move the slider to the far left to create a 'color to black' gradient fill.

3. (option) Click the Two color radio button to create 'two color' gradient fill. Click the colored tiles to open the Select color dialog to pick the base colors.

4. Select one of the 9 gradient patterns. The options are: linear, cylindrical, inverted cylindrical, spherical, hemispherical, curved, inverted spherical, inverted hemispherical and inverted curved.

5. (option) **Centered**: If centered, the gradient fill is created symmetrical; if not centered, the gradient fill is built up from the left of the boundary.

6. (option) **Angle**: Specifies the angle of the gradient fill, relative to the current UCS.

Figure 1.668

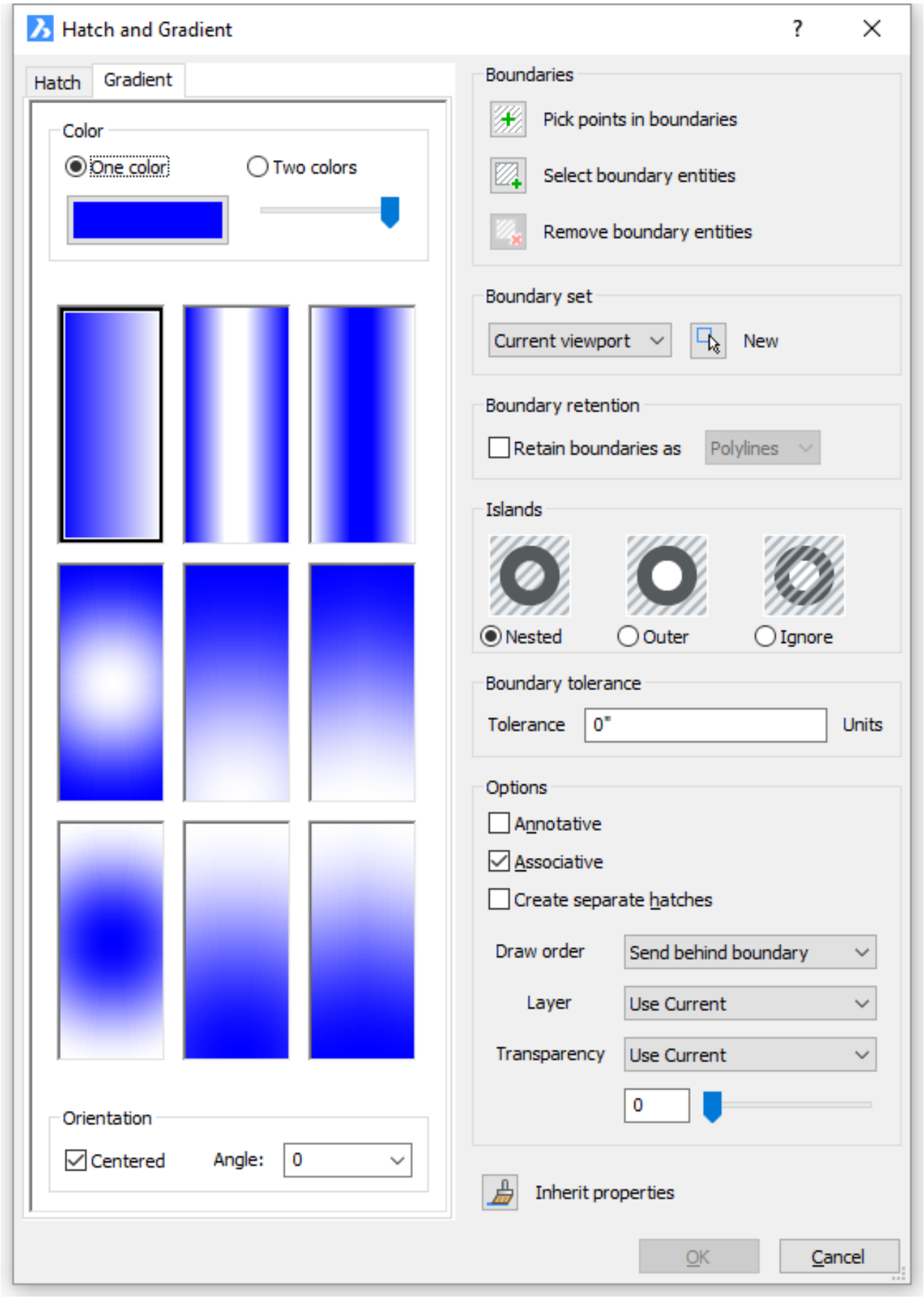

Hatch and Gradient
Hatch
Gradient
Color
One color
Two colors
Orientation
Centered
Angle:
0
Boundaries
Pick points in boundaries
Select boundary entities
Remove boundary entities
Boundary set
Current viewport
New
Boundary retention
Retain boundaries as
Polylines
Islands
Nested
Outer
Ignore
Boundary tolerance
Tolerance
0"
Units
Options
Annotative
Associative
Create separate hatches
Draw order
Send behind boundary
Layer
Use Current
Transparency
Use Current
0
Inherit properties
OK
Cancel

Figure 1.669



To define the boundary for a hatch or gradient fill

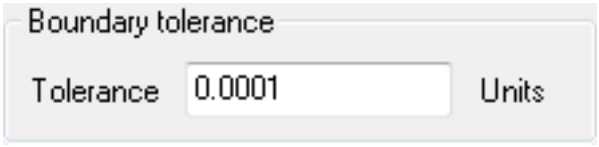
1. (option) Select the **Boundary retention** option.
2. (option) Select an **Island** option. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/
3. (option) Adjust the **Boundary tolerance**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/
4. (option) Click the **Select** button ( ) to select a **Boundary set**. The **Hatch and Gradient** dialog temporarily closes to let you select entities. (see also the note below)
5. (option) Click the **Specified Origin** radio button, then click the **Pick a new origin** button ( ). The **Hatch and Gradient** dialog temporarily closes to let you specify a new origin.
6. Do one or more of the following:
   - Click the **Pick Points** button ( ). The **Hatch and Gradient** dialog closes. You are prompted: Select a point to define a boundary or hatch area:
     - Specify a point in each of the areas you want to hatch. You are prompted: Select a point to define a boundary or hatch area:
     - Right click or press Enter to stop adding areas. The **Hatch and Gradient** dialog displays again.
   - Click the **Select Boundary Entities** button ( ). The **Hatch and Gradient** dialog closes. You are prompted: Select entities:
     - Select the entities you want to hatch.
     - Right click or press Enter to stop selecting entities. The **Hatch and Gradient** dialog displays again.
   - Click the **Remove Boundary Entities** button ( ). The **Hatch and Gradient** dialog closes. You are prompted: Select entities:
     - Select the current boundary entities to be removed from the selection.
     - Right click or press Enter to stop selecting entities. The **Hatch and Gradient** dialog displays again.
7. Define the Hatch Origin. The options are:
   - **Use current origin** : The origin as defined by the HPORIGIN (1.13.11) system variable is reused.
   - **Specified origin** : Click the **Pick new origin** button ( ), then pick a point in the drawing. The new origin is stored in theHPORIGIN (1.13.11)system variable.

Note: If a selection set was active when you launch the **Hatch** tool, this selection will be used as the **Boundary Set** if you click the **New** button in step 11 in the above procedure. In this case the Hatch and Gradient dialog closes and immediately reopens.

Figure 1.670



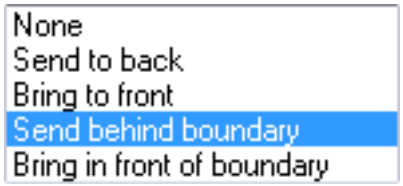
Figure 1.671



Set the Hatch and Gradient options

1. (option) Select the **Annotative** option to create an annotative hatch (1.21.29).
2. (option) Select the **Associative** option to associate the hatch with its boundary.
3. (option) Select the **Create Separate Hatches** option. In case multiple boundaries are detected or multiple entities are selected in step 9, separate hatches are created instead of a single hatch.
4. (option) Choose a **Draw order** option. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/so
This option is reset to **Send behind boundary** when the drawing is closed and reopened.

Figure 1.672



1.17.29 Editing a hatch or gradient fill

Commands: HATCHEDIT (6.16.3) and -HATCHEDIT (6.16.4) If you select a hatch or gradient fill, its properties display in the Properties bar.

To edit a hatch or gradient fill in the properties bar

1. Select the hatch or gradient fill in the drawing. The properties of the selected entity display in the Properties bar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04Worki
Hatch properties (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEnti
Gradient fill properties.
2. Select the property you want to modify in the Properties bar. The selected property field is activated.
3. Modify the selected property
4. Do one of the following:
   - Repeat steps 2 and 3 to modify another property.
   - Press the Esc key to stop editing the hatch.

NOTES

- If you select multiple hatches you can you edit them simultaneously. Properties which are different display as ***varies*** in the BricsCAD Properties bar. If you edit such property, all edited hatches will then share this property.
- Linetype (1.17.6), Linetype scale (1.17.6) and Lineweight (1.17.5) apply to **User Defined** pattern types only .

Figure 1.673

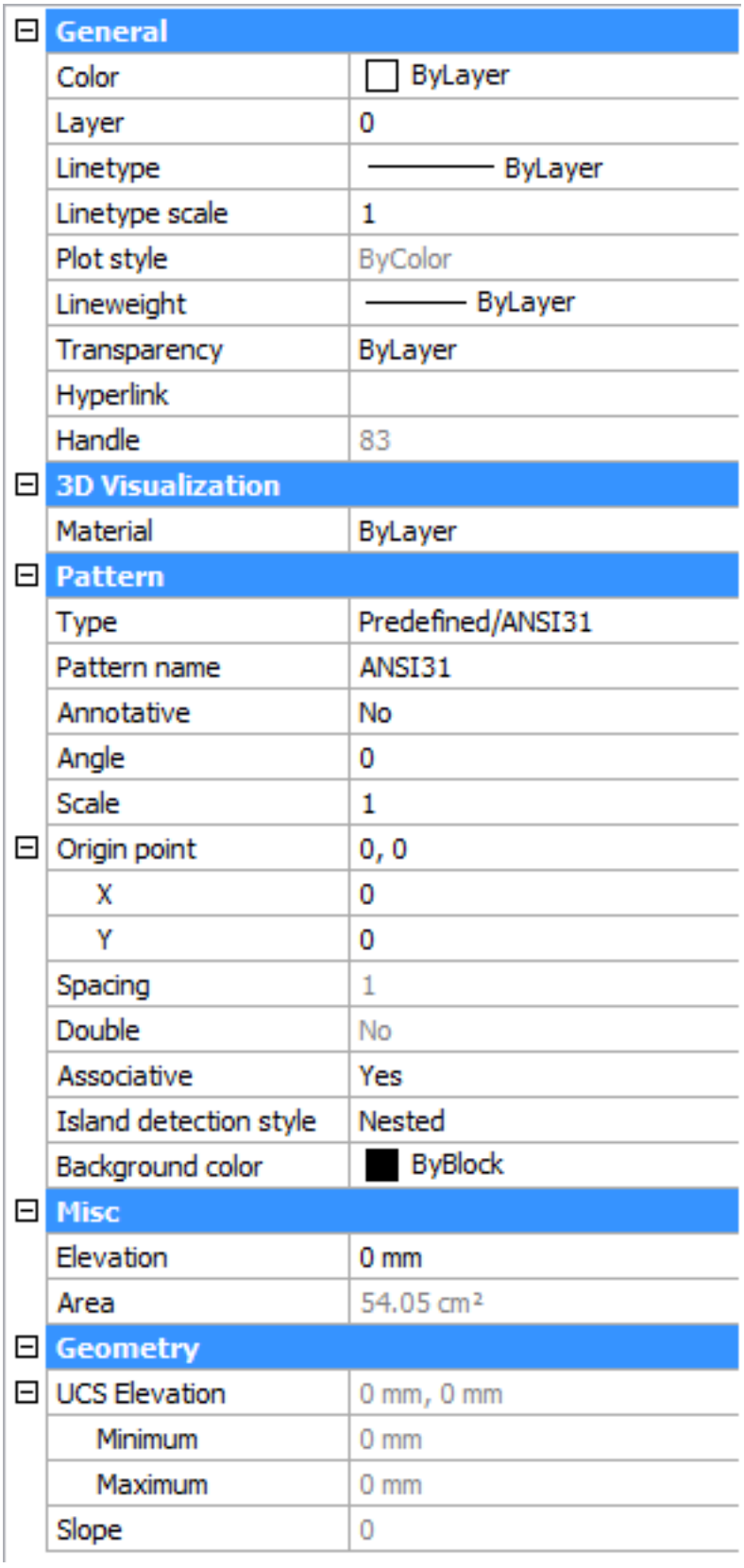
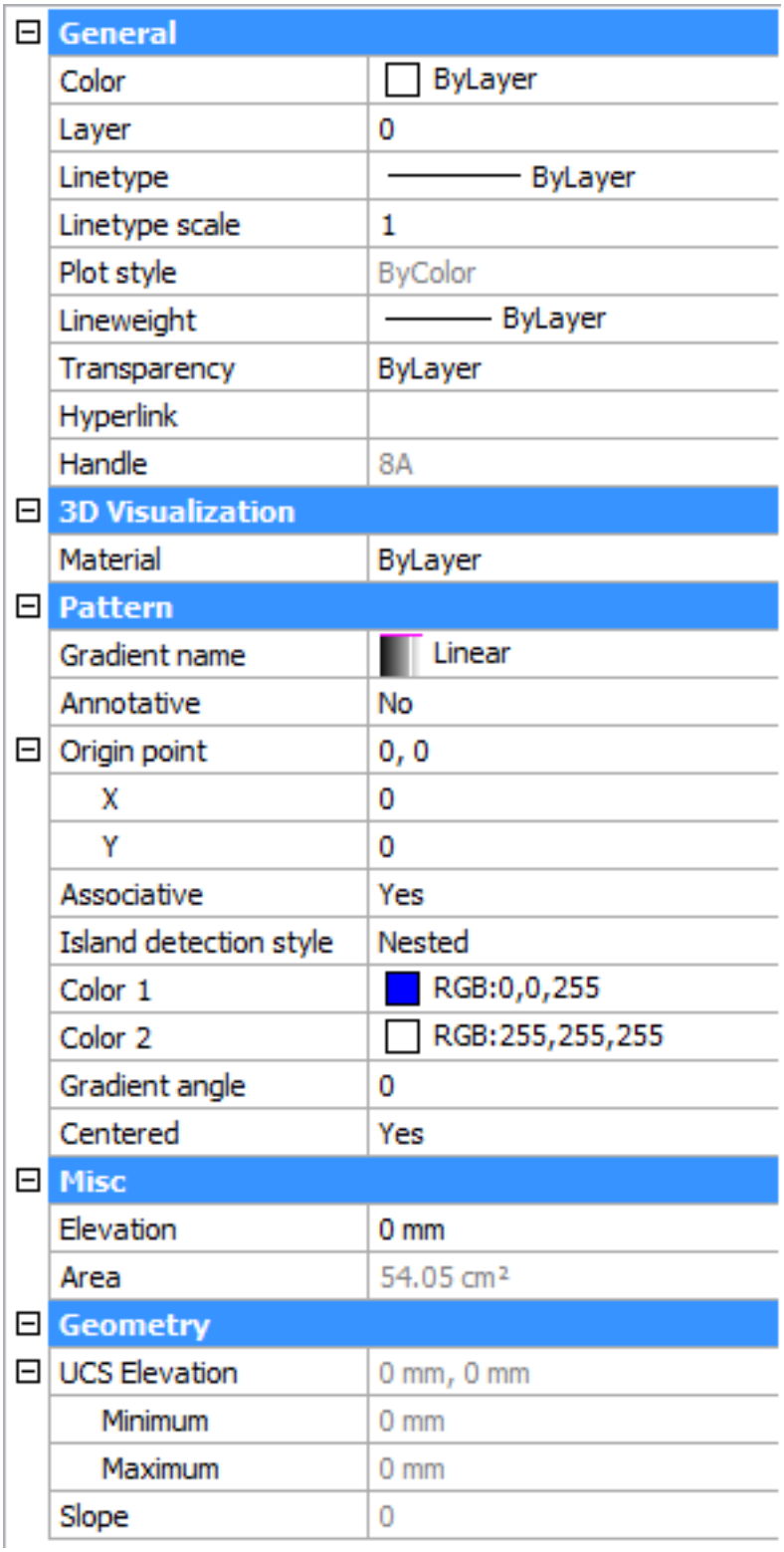

General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Plot style
ByColor
Lineweight
ByLayer
Transparency
ByLayer
Hyperlink
Handle
83
3D Visualization
Material
ByLayer
Pattern
Type
Predefined/ANSI31
Pattern name
ANSI31
Annotative
No
Angle
0
Scale
1
Origin point
0, 0
X
0
Y
0
Spacing
1
Double
No
Associative
Yes
Island detection style
Nested
Background color
ByBlock
Misc
Elevation
0 mm
Area
54.05 cm²
Geometry
UCS Elevation
0 mm, 0 mm
Minimum
0 mm
Maximum
0 mm
Slope
0

Figure 1.674


General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Plot style
ByColor
Lineweight
ByLayer
Transparency
ByLayer
Hyperlink
Handle
8A
3D Visualization
Material
ByLayer
Pattern
Gradient name
Linear
Annotative
No
Origin point
0, 0
X
0
Y
0
Associative
Yes
Island detection style
Nested
Color 1
RGB:0,0,255
Color 2
RGB:255,255,255
Gradient angle
0
Centered
Yes
Misc
Elevation
0 mm
Area
54.05 cm²
Geometry
UCS Elevation
0 mm, 0 mm
Minimum
0 mm
Maximum
0 mm
Slope
0

To edit a hatch in a dialog box

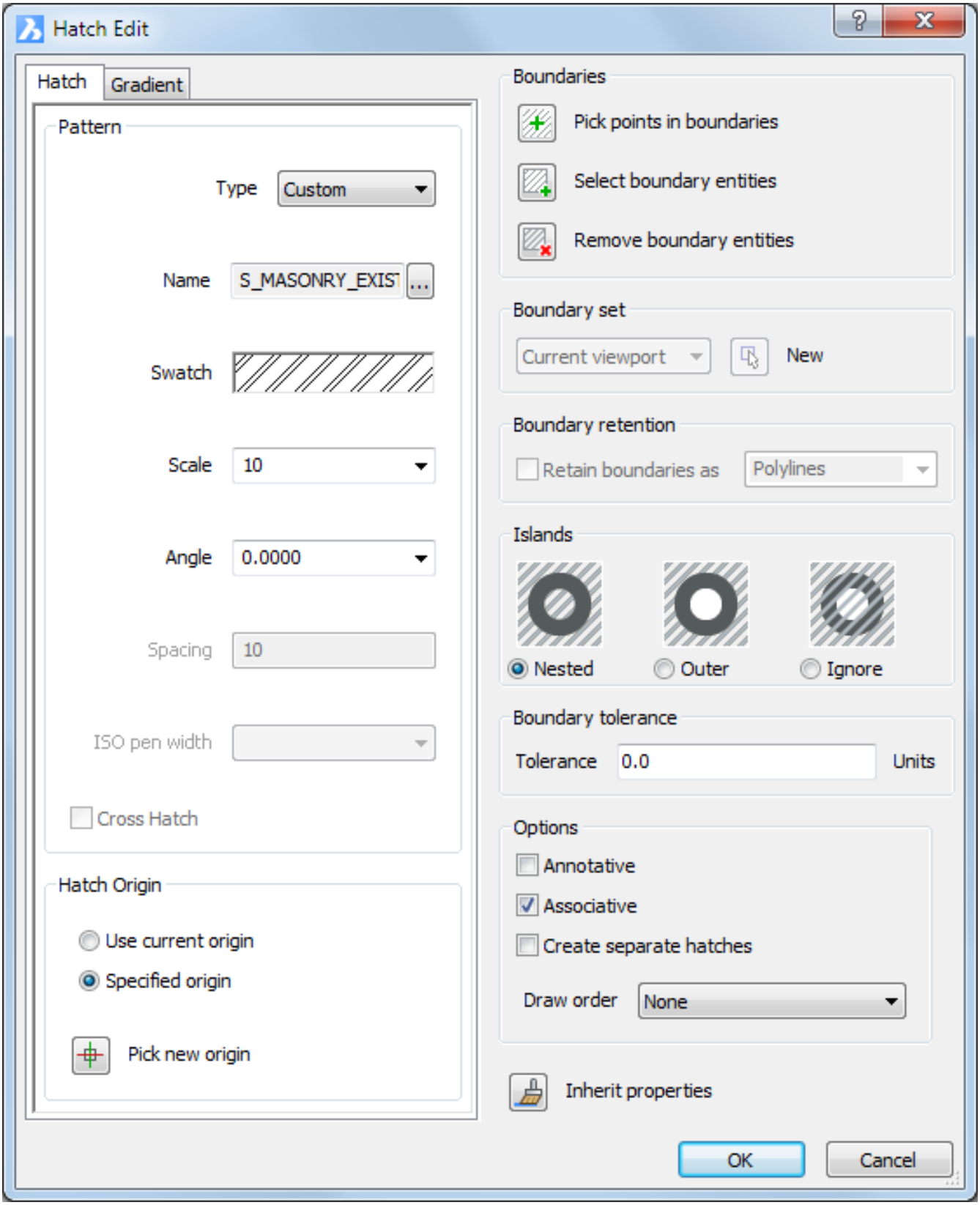
1. Do one of the following:

   - Double click a hatch or gradient fill entity.
   - Type **hatchedit** at the command prompt, then select a hatch or gradient fill entity.

   The Hatch Edit dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/1

2. (option) Select the Gradient tab to turn the Hatch into a Gradient fill.
3. Edit the Pattern settings (1.17.28).
4. Edit the Boundary settings (1.17.28).
5. Click the **OK** button to apply the changes.

Figure 1.675


Hatch Edit
Hatch
Gradient
Pattern
Type
Custom
Name
S_MASONRY_EXIS
Swatch
Scale
10
Angle
0.0000
Spacing
10
ISO pen width
Cross Hatch
Hatch Origin
Use current origin
Specified origin
Pick new origin
Boundaries
Pick points in boundaries
Select boundary entities
Remove boundary entities
Boundary set
Current viewport
New
Boundary retention
Retain boundaries as
Polylines
Islands
Nested
Outer
Ignore
Boundary tolerance
Tolerance
0.0
Units
Options
Annotative
Associative
Create separate hatches
Draw order
None
Inherit properties
OK
Cancel

To edit a gradient fill in a dialog box

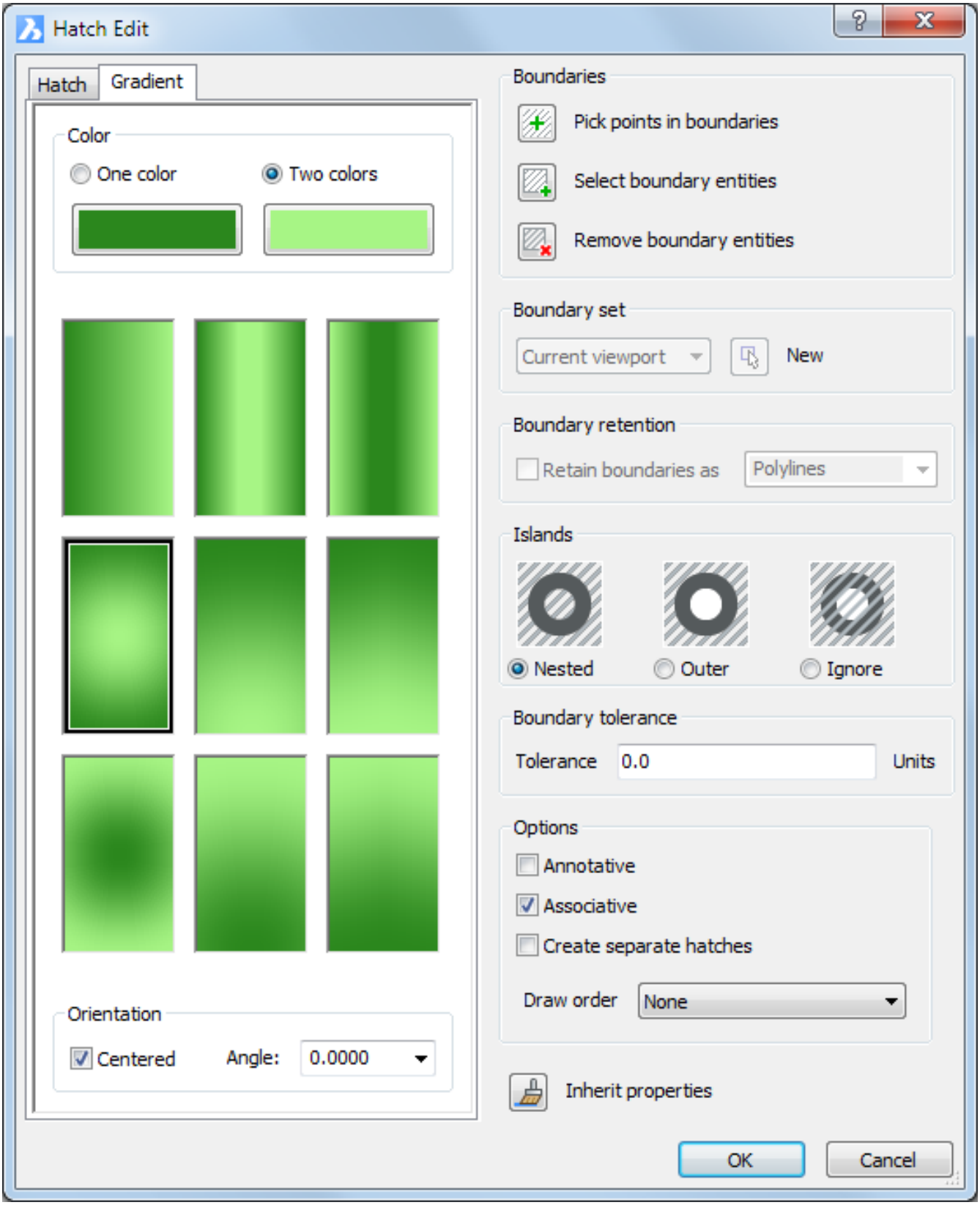
1. Do one of the following:

   - Double click a hatch or gradient fill entity.
   - Type **hatchedit** at the command prompt, then select a hatch or gradient fill entity.

   The Hatch Edit dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/1

2. (option) Select the **Hatch** tab to turn the Gradient fill into a Hatch .
3. Edit the Color settings (1.17.28).
4. Edit the Boundary settings (1.17.28).
5. Click the **OK** button to apply the changes.

Figure 1.676


Hatch Edit
Hatch
Gradient
Color
One color
Two colors
Orientation
Centered
Angle:
0.0000
Boundaries
Pick points in boundaries
Select boundary entities
Remove boundary entities
Boundary set
Current viewport
New
Boundary retention
Retain boundaries as
Polylines
Islands
Nested
Outer
Ignore
Boundary tolerance
Tolerance
0.0
Units
Options
Annotative
Associative
Create separate hatches
Draw order
None
Inherit properties
OK
Cancel

To edit a hatch or gradient fill using the Quad

1. If necessary, hit the F12 function key to activate the Quad cursor menu (1.12.5).
2. Pause the cursor of the hatch or gradient fill. The Quad displays.
3. Move the cursor to the Quad and expand the **2D Editing** command group. (\vref {fig:C:/Dev/Task-2448/images/staticbrics
4. Do one of the following:
   - Click the **Add Loop** button ( ). You are prompted: Specify internal point or [Select entities]: Click inside a closed area, or Type **S** , then press Enter to select entities. Right click or press Enter to conclude.
   - Click the **Remove Loop** button ( ). The boundaries of the selected hatch highlight. You are prompted: Select entities: Click the boundaries you want to remove. Right click or press Enter to conclude.
   - Click the **Associate** button ( ). You are prompted: Specify internal point or [Select entities]: You are prompted: Specify internal point or [Select entities]: Click inside a closed area, or Type **S** , then press Enter to select entities. Right click or press Enter to conclude.
   - Click the **Generate Boundary** button ( ). The boundary of the hatch is generated.
   - Click the **Send all hatches to back** button ( ). You are prompted: # hatch entity(ies) sent to back.

   Note: To repeat a Quad command, right click when the Quad displays.

   Tutorial: Hatch made easy using the Quad Click here (1.5.1) to watch

Figure 1.677



1.17.30 Settings for Drawing Entities

The default values and preferences to create many of the fundamental 2D drawing entities are specified by system variables. You can view and modify variables in BricsCADusing the SETVAR command at the Command line. If you prefer to change variables through the user interface, you can use the Settings dialog. It offers a single, central location to view and modify all the variables in BricsCAD. There are more than 900 of them!

BricsCAD Settings

Access Settings from the General menu on the Quad or via other methods including the Command line (SETTINGS, OPTIONS). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04Wor Settings01.png}) With so many variables, I can't possibly cover them all in this post. Instead, I'll focus on the ones for entity creation. After opening the Settings dialog box, expand the Drafting node and then expand Entity Creation. These properties apply to new entities you create. At the top of the list are general properties including color, linetype, and layer. Below that are entity-specific properties. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V Settings02.png}) Expand an entity node, such as Hatches, to view and modify its relevant properties. Select a property to display the name of the variable it controls in the lower left corner of the dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ART/Drawing-Settings03.png}) Some of the entity controls are available from other locations, such as the Entity Properties toolbar. But, you will find ALL of them in the Settings dialog box. You don't have to navigate through multiple dialog boxes in search of the control you want. And, the best part is the search capability. Enter a relevant term in the search box, such as HATCH or PICK. Relevant properties highlight as you click your way through each occurrence of the term. Even if you are completely new to BricsCAD, you can easily find the settings you're looking for! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ Settings04.png})

Figure 1.678

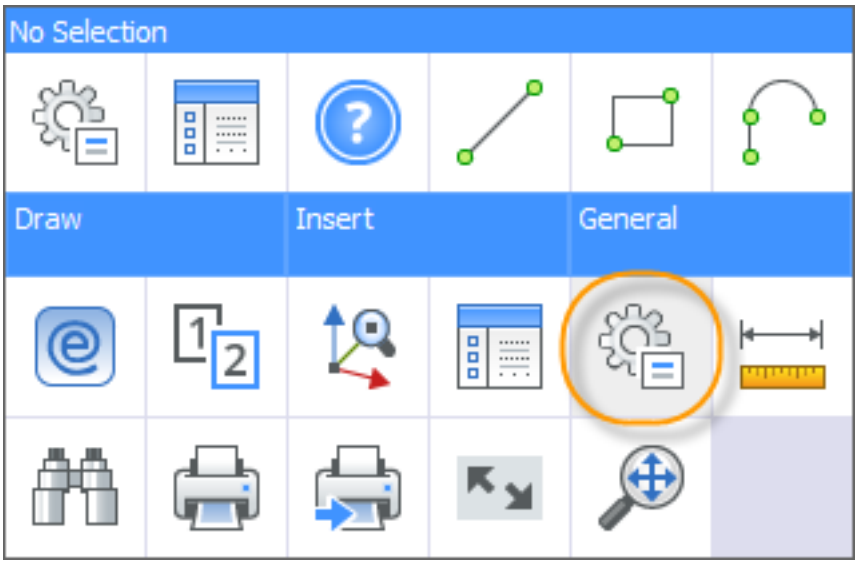


Figure 1.679

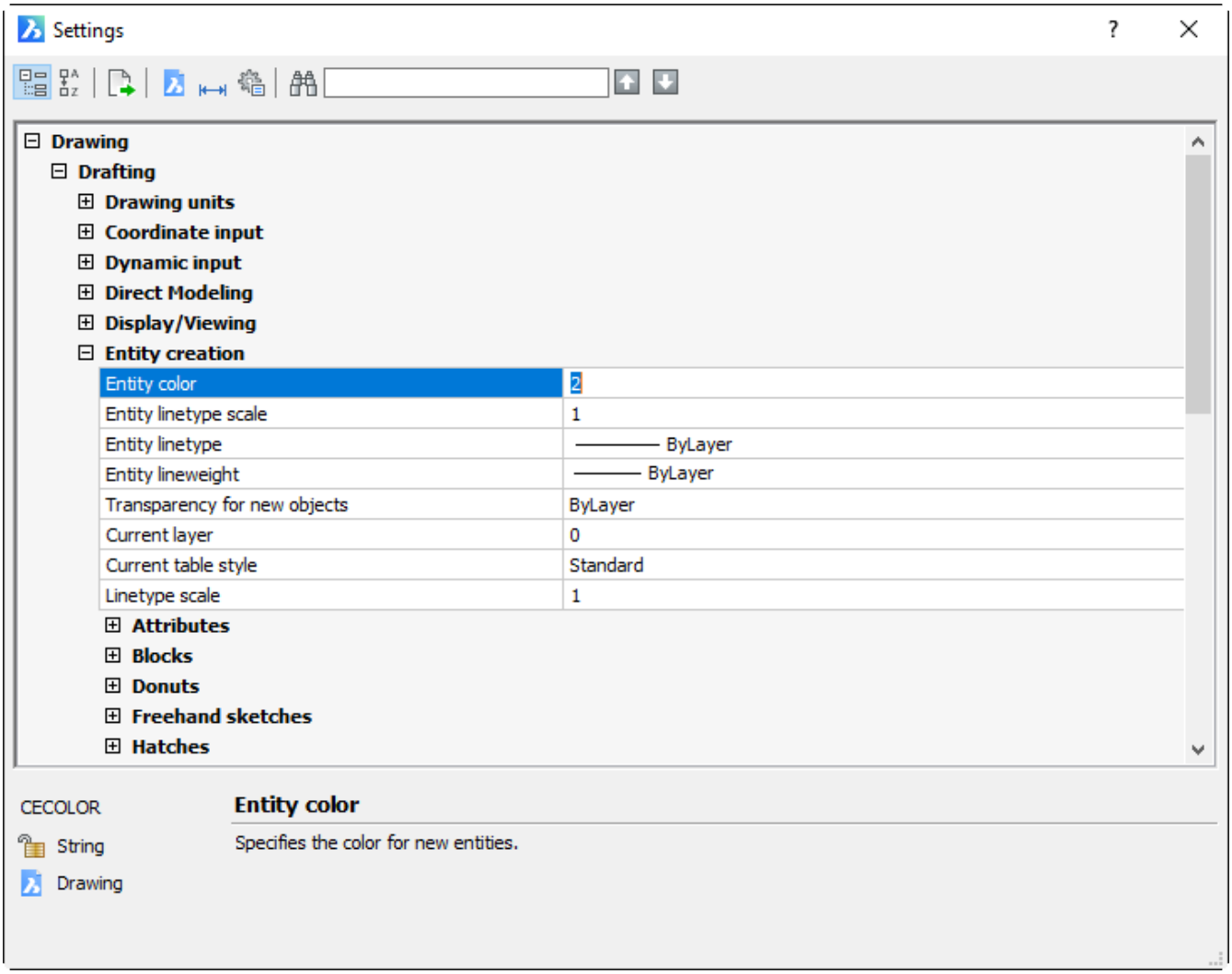


Figure 1.680

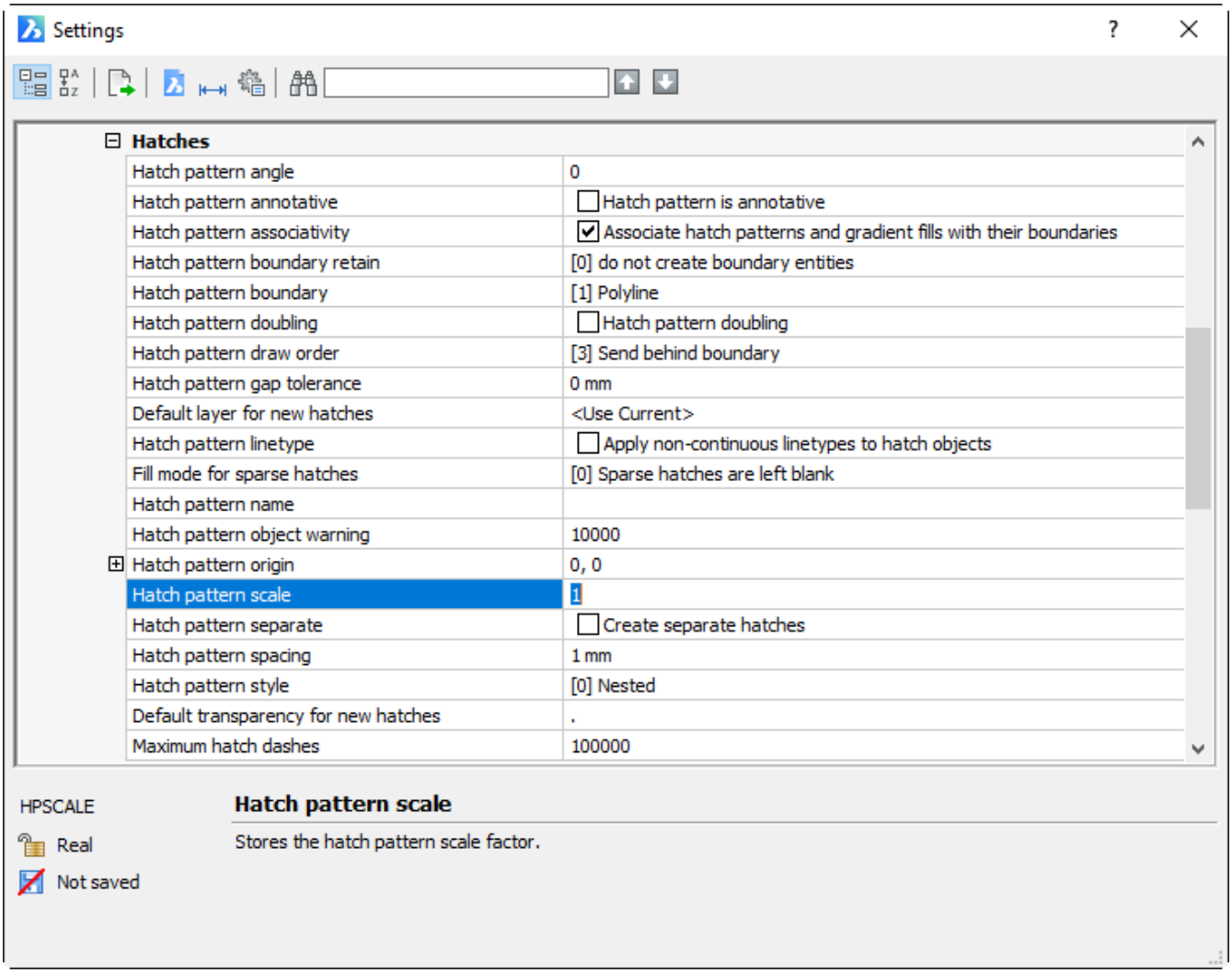
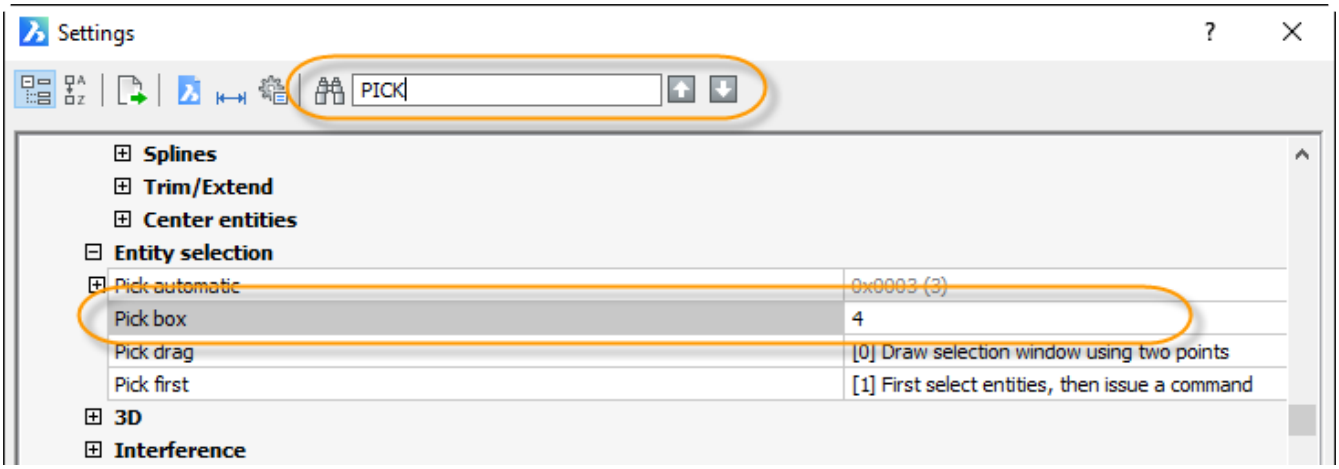


Figure 1.681



1.17.31 Working with Layers and Linetypes

Drawing Explorer

The default method to create and manage layers in BricsCAD is via Drawing Explorer. Because, like everything else in Drawing Explorer, layers are named drawing content. You can, of course, launch Drawing Explorer from the Quad and other methods including the command line. You can also launch it with the LAYER command (LA alias). Doing so automatically displays the Layers panels in Drawing Explorer. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
with-Layers-and-Linetypes01.png}) One of the primary benefits of managing layers in Drawing Explorer is the ability to preview layer geometry as you manage layer properties. It is similar to the LAYWALK tool in AutoCAD®. However, unlike AutoCAD, BricsCAD enables you to view geometry on any number of layers and then, with those layers already selected, you can change their properties. It's very cool and eliminates the need for a separate LAYWALK command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHa
with-Layers-and-Linetypes01.png}) The toolbar and right-click menu provide access to additional tools relevant for layer management. For example, you can merge layers or open and close the Filters panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ART/Working-with-Layers-and-Linetypes03.png}) Drawing Explorer is incredibly useful as a central location for managing all the named content in your drawings. However, it is a dialog box and cannot remain open while you use other commands.

Figure 1.682

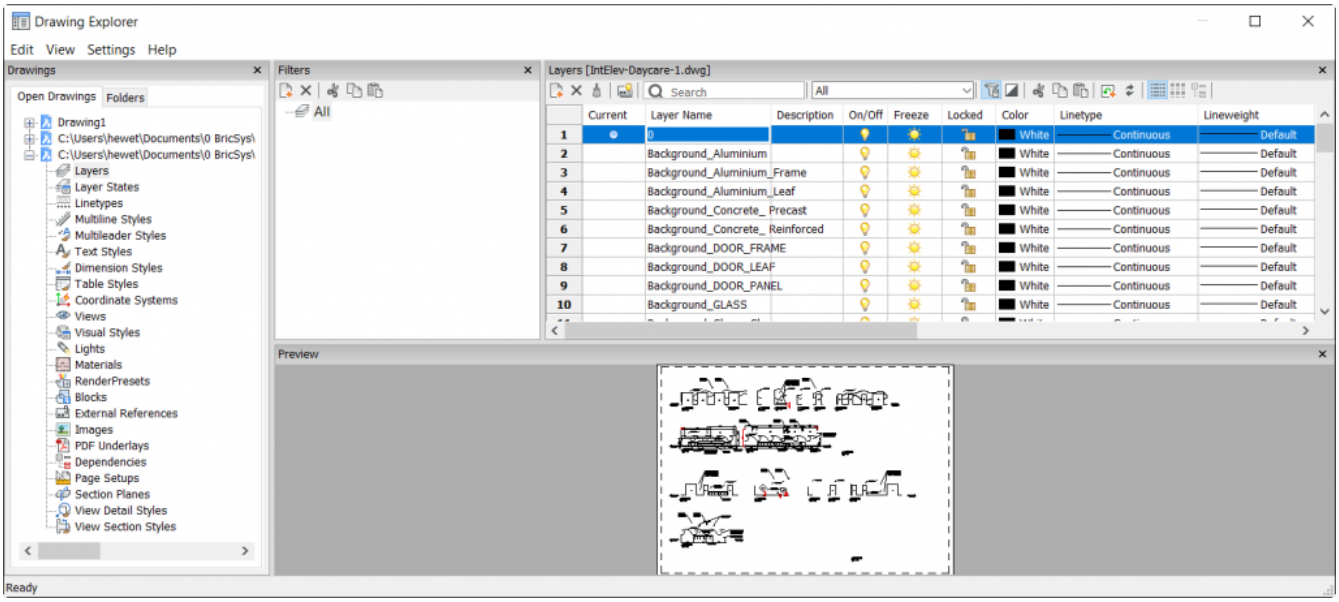


Figure 1.683

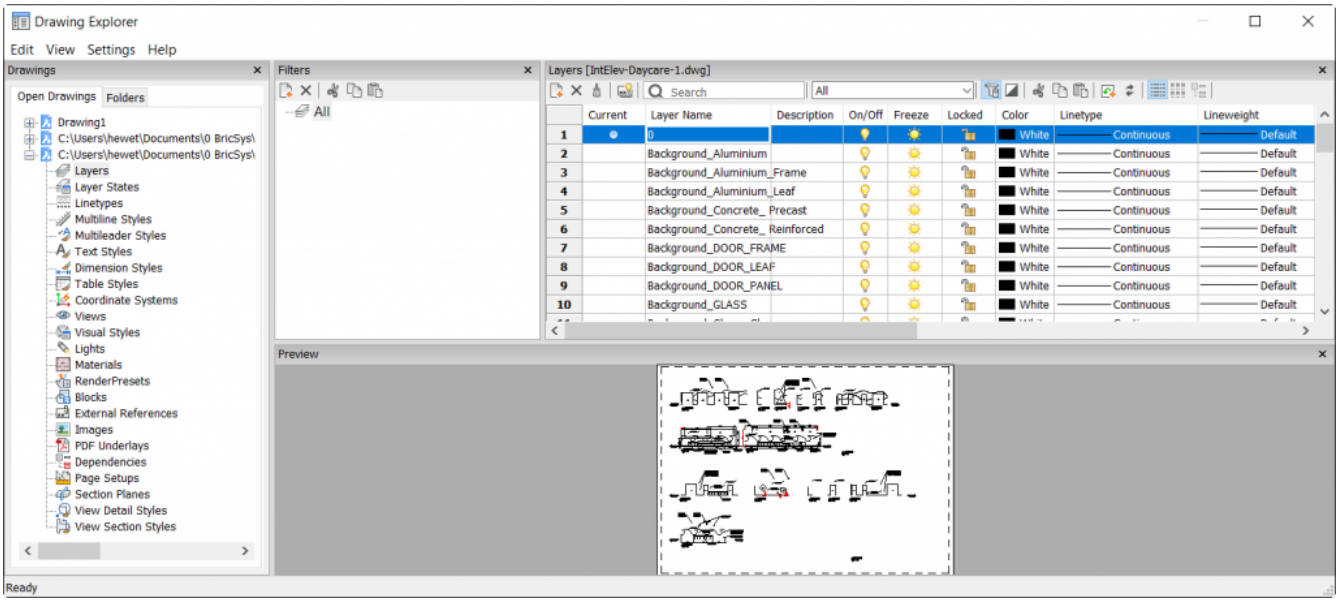
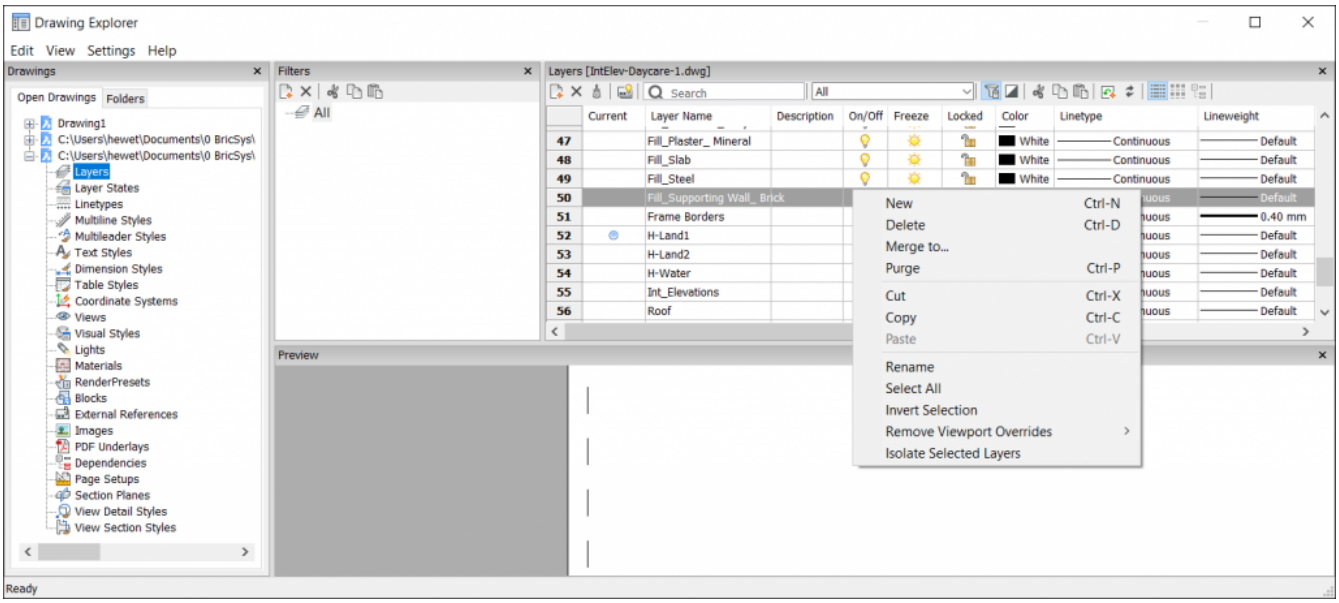


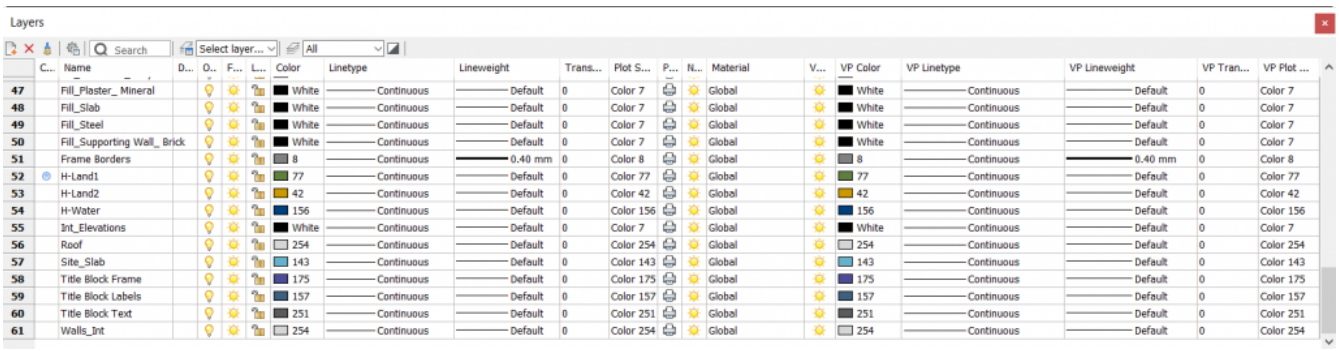
Figure 1.684



Layers Panel

If you frequently access the Layers panel, you may want to leave it open while you work. No problem! Use the LAYERSPANELOPEN command to display only the Layers panel from the Drawing Explorer. Move and resize it as you wish, and even move it to a second monitor. If you find old habits hard to break and prefer for the LAYER command and LA alias to open the Layers panel instead of the Drawing Explorer. It's easy to change and I'll show you how in a future post! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04Wo
with-Layers-and-Linetypes04.png})

Figure 1.685

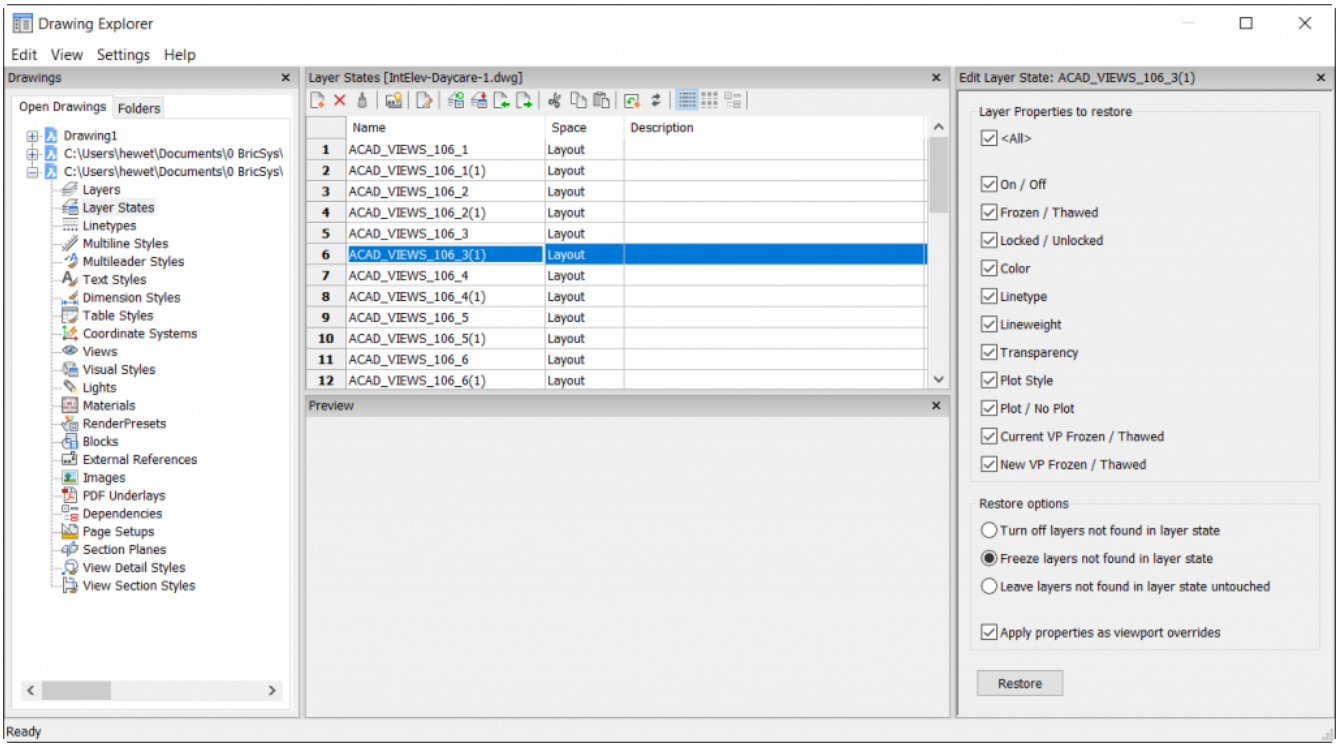


Layer States

Layer states are named content stored in a drawing. How do you think you access them in BricsCAD? If you answered, “Drawing Explorer”, you’re correct! You can, of course, open Drawing Explorer via typical methods. Or you can launch it with the LAYERSTATE command (LAS alias) to open it with the Layer States panels already displayed. It includes one panel to access layer states and another panel to modify properties for the selected state. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ with-Layers-and-Linetypes05.png})

Tutorial: Employing the Display Power of Layer States Click here (1.3.18) to watch

Figure 1.686



Linetypes

The LINETYPE command (LT alias) in BricsCAD opens the Drawing Explorer with the Linetypes panel displayed. This is where you can load new linetypes or delete existing ones. It's like the Linetype Manager in AutoCAD. However, in BricsCAD you won't find controls for linetype scaling (LTSCALE and CELTSCALE) in the Drawing Explorer because those aren't named content. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Drawin with-Layers-and-Linetypes06.png}) Instead, you can specify Linetype scales in the Settings dialog box or at the Command line like any other system variable. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sourc with-Layers-and-Linetypes07.png})

Figure 1.687

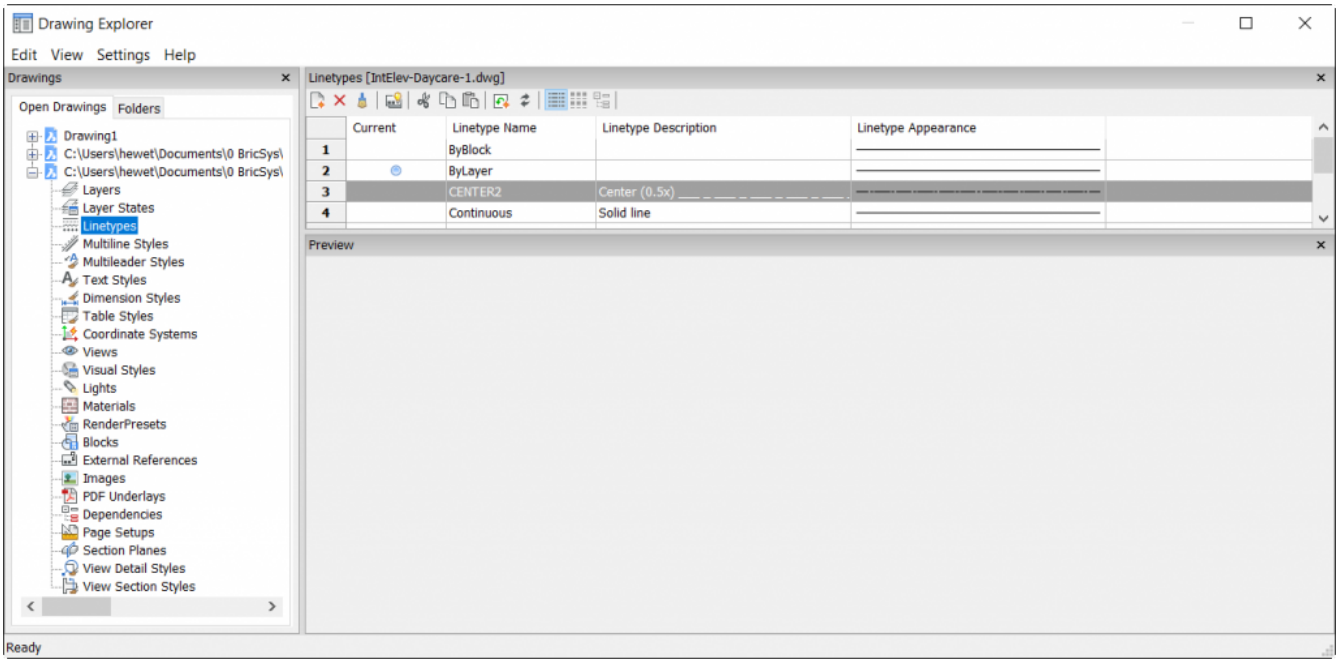
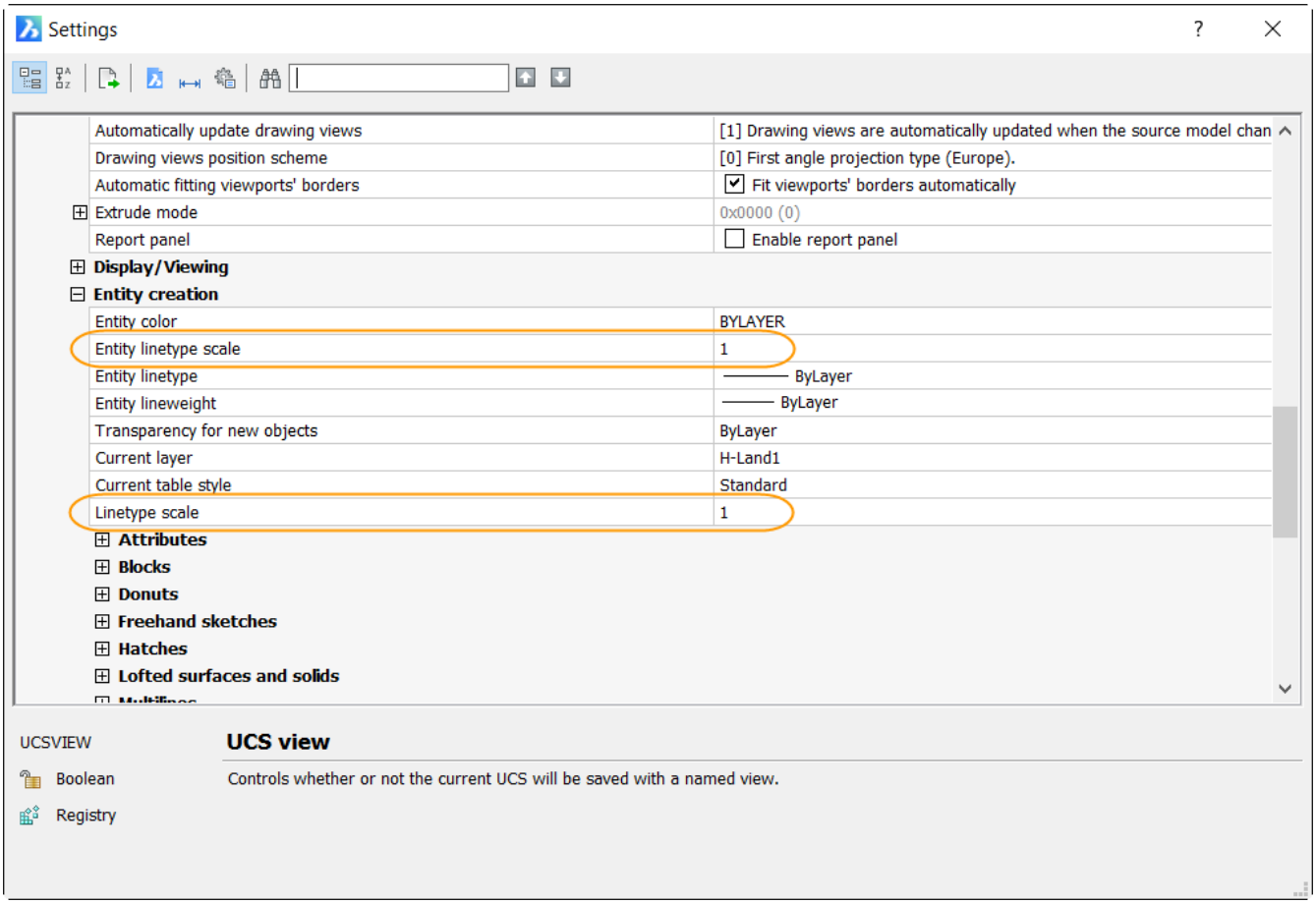


Figure 1.688



1.17.32 Editing Entities

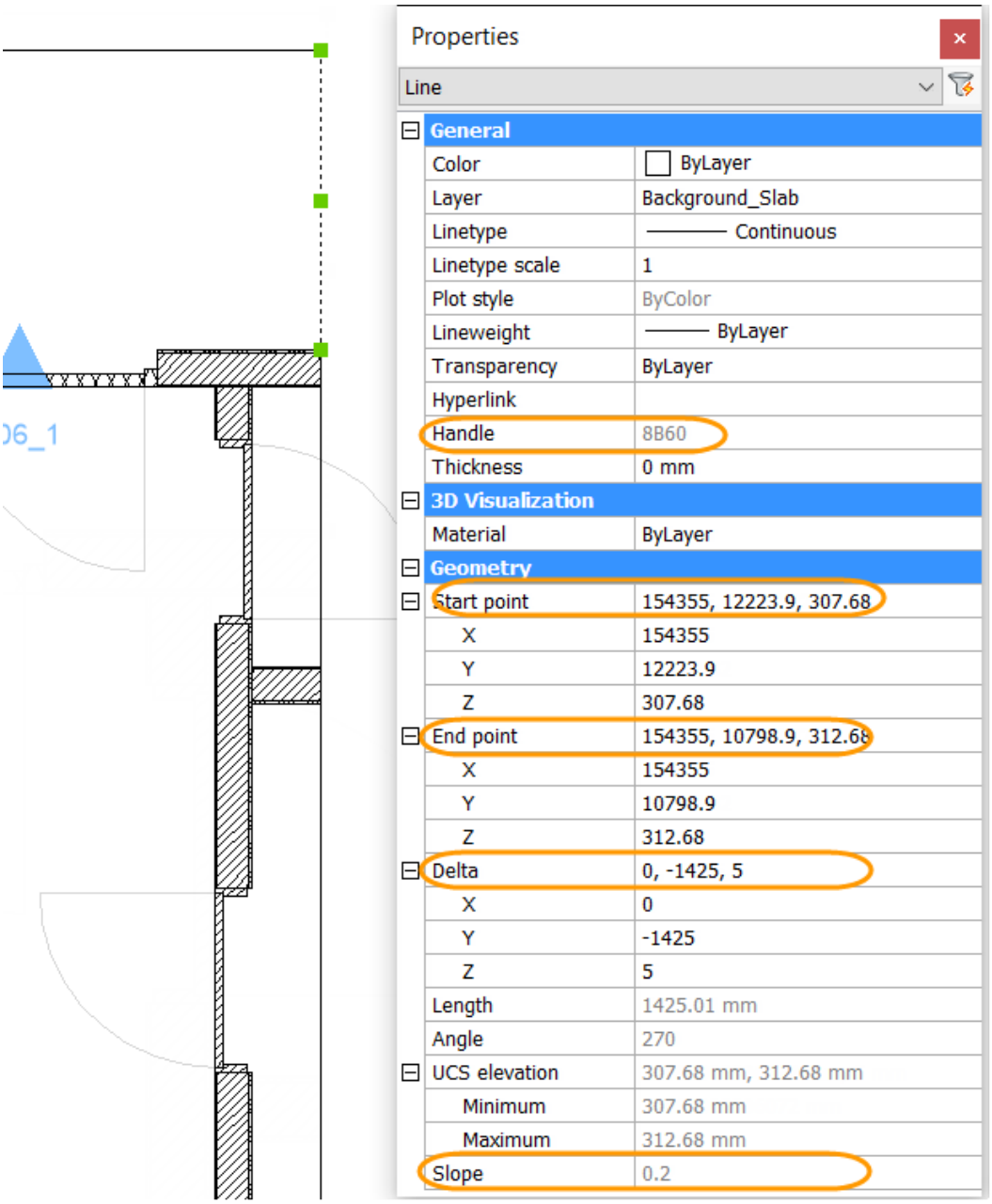
Editing in the Properties Panel

BricsCAD offers a Properties panel like the Properties palette in AutoCAD. You can access it with the PROPERTIES command (PR alias) as well as other methods such as double-clicking on an object. Since it's a panel, it can remain open while you work. The properties displayed in the panel vary depending on the type of object(s) that you've selected. In general, they match what you would expect from your AutoCAD experience. There are, however, a few notable differences. For example, the Properties panel in BricsCAD displays the Handle (read-only) of the selected entity. The entity's handle can be useful to store and recall for programing purposes. You'll find other valuable additions to the Properties panel for different objects. I can't possibly cover all of them, but below is an example of properties for a selected line. In BricsCAD you can edit a line by entering the x,y,z values of the start point, end point, or even the delta. It also displays the slope as a read-only value. While this is supposed to be a 2D drawing, it's obvious from the slope of the selected line, that this particular entity isn't flat. You could, of course, edit the z-value of its endpoint to match the start point. However, you may incorrectly assume its accuracy beyond the second decimal place. Instead, you can change the Delta z-value to 0 ensuring the line is truly flat!

Editing with Grips

BricsCAD supports traditional object grips like AutoCAD. For example, if you select a circle, you can use its center and quadrant grips to move or resize it. BricsCAD does not, however, support mutli-functional grips. Instead, much of that functionality is available on the Edit tab of the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/Entities01.png})

Figure 1.689


Properties
Line
General
Color
ByLayer
Layer
Background_Slab
Linetype
Continuous
Linetype scale
1
Plot style
ByColor
Lineweight
ByLayer
Transparency
ByLayer
Hyperlink
Handle
8B60
Thickness
0 mm
3D Visualization
Material
ByLayer
Geometry
Start point
154355, 12223.9, 307.68
X
154355
Y
12223.9
Z
307.68
End point
154355, 10798.9, 312.68
X
154355
Y
10798.9
Z
312.68
Delta
0, -1425, 5
X
0
Y
-1425
Z
5
Length
1425.01 mm
Angle
270
UCS elevation
307.68 mm, 312.68 mm
Minimum
307.68 mm
Maximum
312.68 mm
Slope
0.2

Editing with the Quad

As I mentioned in a previous post, the Quad offers different edit tools based on the type of entity. Circle edit tools are quite different, for example, from Dimension edit tools. When you select a dimension, you're offered tools to modify the dimension or dimension text. For example, you can flip the dimension arrowhead nearest to where you selected the dimension. In contrast, Circle edit tab includes tools like Trim and Hatch. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ART/Editing-Entities02.png}) The default tool changes based on your most recent editing behavior for that type of entity. You can hover over the default tool to display additional recently used tools. The recent tools can vary depending if you select the entity or simply hovered over it. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10 Entities03.png}) Pass the cursor over a tab at the bottom of the Quad to expand it with even more tools. You'll find the most common tools to edit that specific type of entity on the Edit tab. The tools may vary depending if you select the entity or simply hover over it. For example, if you select a line, the Edit tab displays the Lengthen, Break, Join, and Match tools. If you hover over a line, without selecting it, the Quad adds Trim and Extend to the tool set. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ Entities04.png}) In this example, if you hover over the line and use the Extend tool, BricsCAD automatically extends the highlighted entity to the nearest boundary based on where you hovered the cursor. It only takes one pick compared to three picks and two enters using traditional methods. Of course, traditional methods are still available. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHa Entities05.png})

Figure 1.690

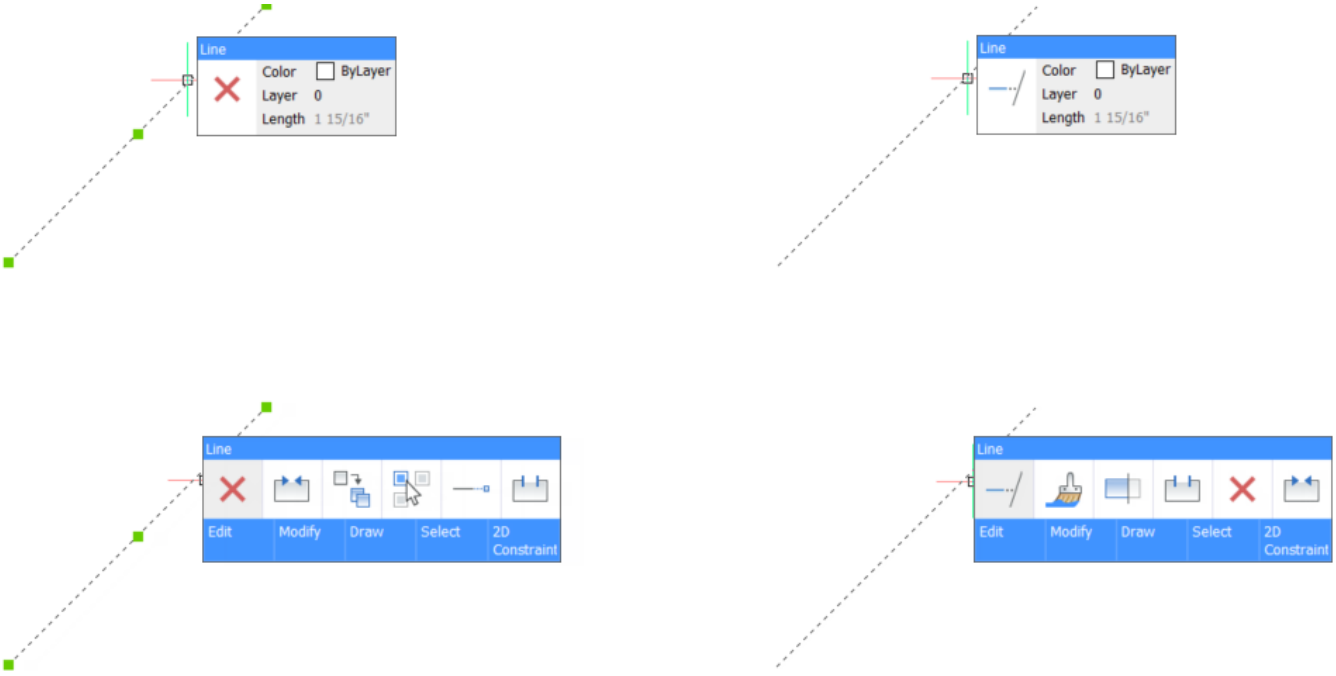


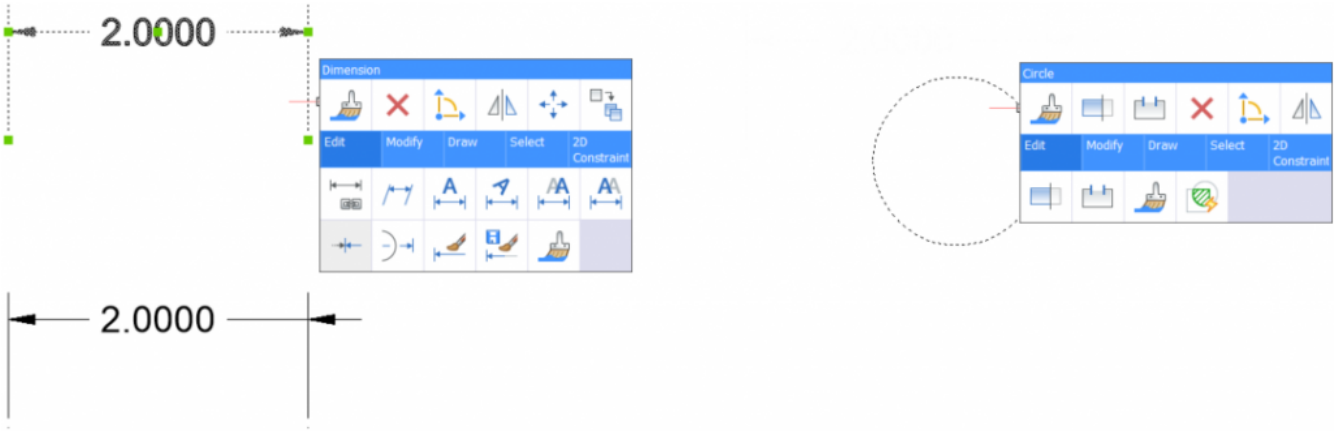
Figure 1.691



Figure 1.692



Figure 1.693


2.0000
Dimension
Edit
Modify
Draw
Select
2D Constraint
2.0000
Circle
Edit
Modify
Draw
Select
2D Constraint

1.17.33 Drawing Entities

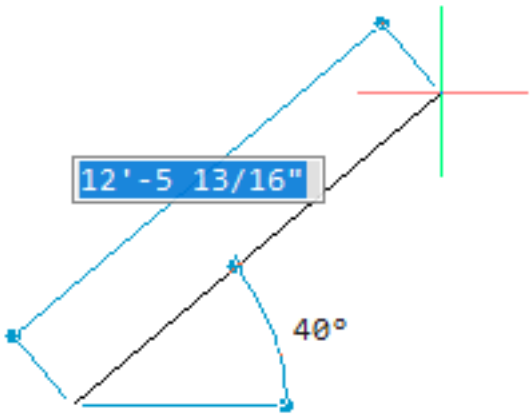
Draw Polylines, Splines, Ellipses, Elliptical Arcs, and Wipeouts

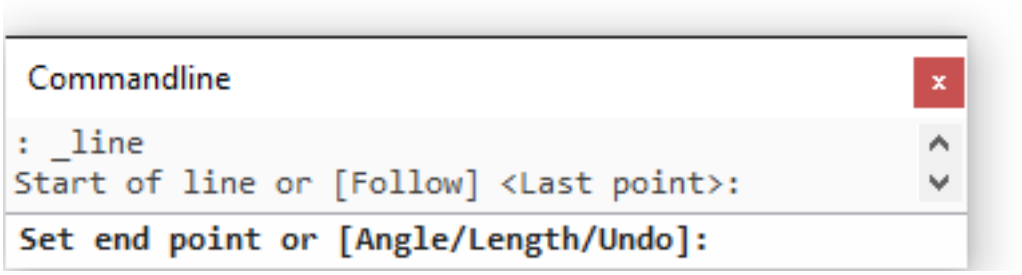
The processes to create Splines, Polylines, Ellipses, Elliptical Arcs, and Wipeouts in BricsCAD are just as you would expect in AutoCAD.

Draw Lines

The Line tool in BricsCAD works, by default, like you would expect. It includes the ability to enter the length and angle via dynamic input. However, BricsCAD also includes Command options to switch between Angle and Length. This offers you additional design flexibility even when Dynamic Input is turned off. Another minor difference is the terminology to continue a new line, arc or polyline from the last segment. In BricsCAD, the option is “Follow” instead of Continue. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkEntities02.png})

Figure 1.694

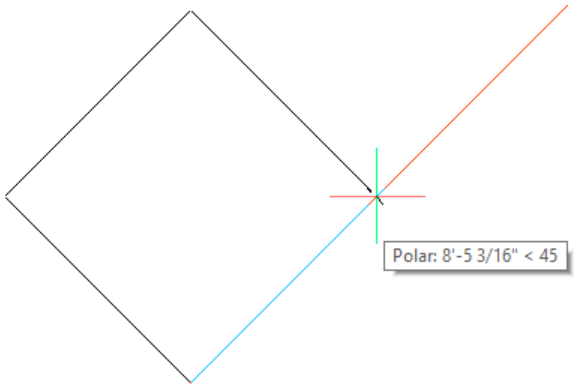


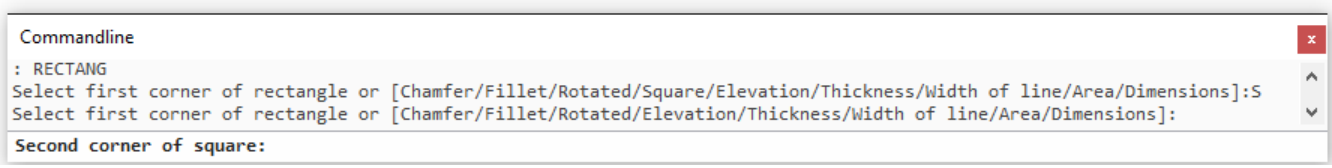


Draw Rectangles

The default behavior to draw a rectangle should be familiar to you. However, BricsCAD includes additional options offering maximum flexibility with minimal clicks. Creating a square is a great example! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ART/Drawing-Entities03.png})

Figure 1.695

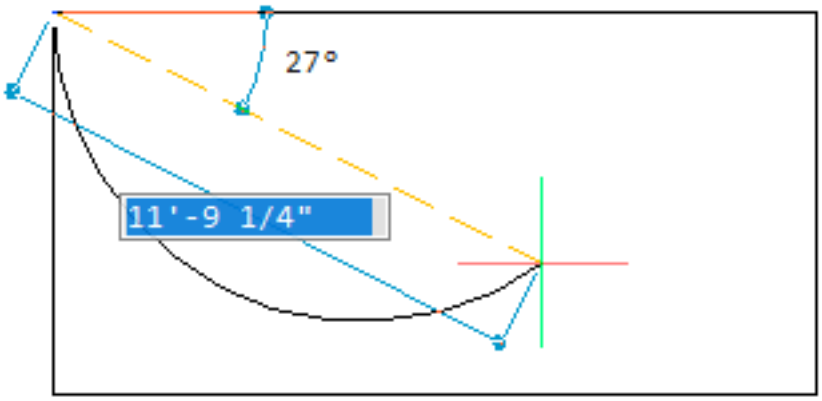




Draw Arcs

The methods to draw an Arc, by default, are as you would expect. However, BricsCAD includes all the arc creation options after the first pick. It provides more flexibility with fewer clicks. For example, you can pick the first point of an arc. Then, use the Direction option to specify its direction of tangency. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ART/DrawingEntities04.png})

Figure 1.696



Commandline

```
Enter start of arc or [Center/Follow] <Last point>:
Enter second point or [Angle/Center/Direction/End/Radius]:D
Enter initial direction for arc <Follow>:
End point:
```

Draw Circles

The options to draw a circle in BricsCAD should be familiar to you. However, the Circle command includes two extra options that you may not expect, but will certainly appreciate! One option allows you to quickly convert an arc to a circle. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithEntities05.png}) The other option, Multiple, enables you draw a circle and then copy it as many times as you like. The center of each new circle is placed in relation to the previous one. This allows you to create many circles, quickly and accurately. Very handy! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities06.png})

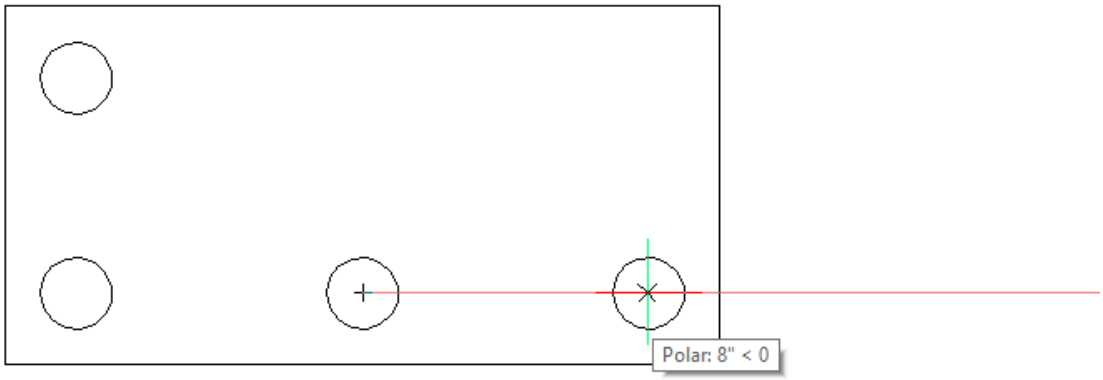
Figure 1.697



Commandline

```
:
: CIRCLE
Select center of circle or [2 Point/3 Point/TangenT-tangent-Radius/turn Arc into circle/Multiple circles]:a
Select arc to convert to circle:
```

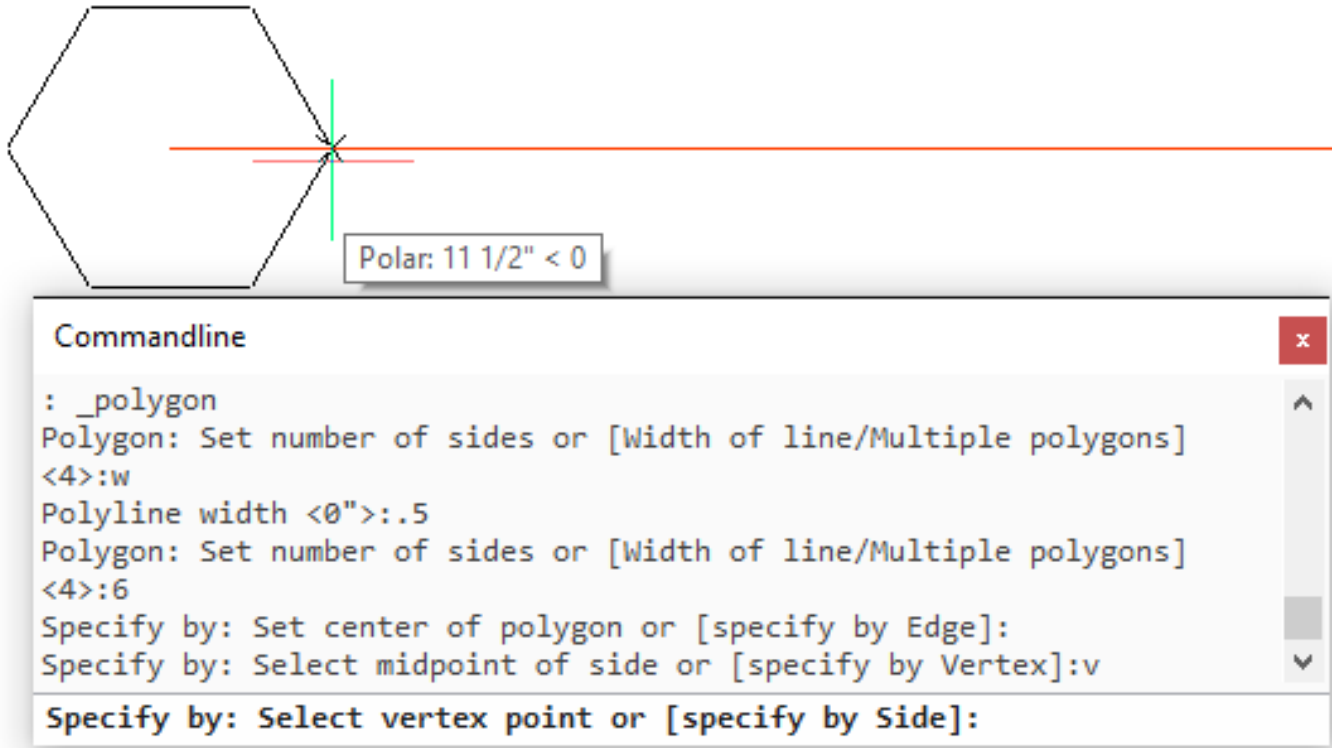
Figure 1.698



Draw Polygons

The default behavior to draw a polygon in BricsCAD works as you expect. However, it also includes an option for you to specify a line width while you draw the polygon. This reduces the need for you to modify the width property after it's created! When you create a polygon using the center option, you might notice a difference in terminology. Instead of prompting for inscribed or circumscribed, it asks if you want to select using the midpoint of the side or the vertex. Intuitive terminology! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawiEntities07.png})

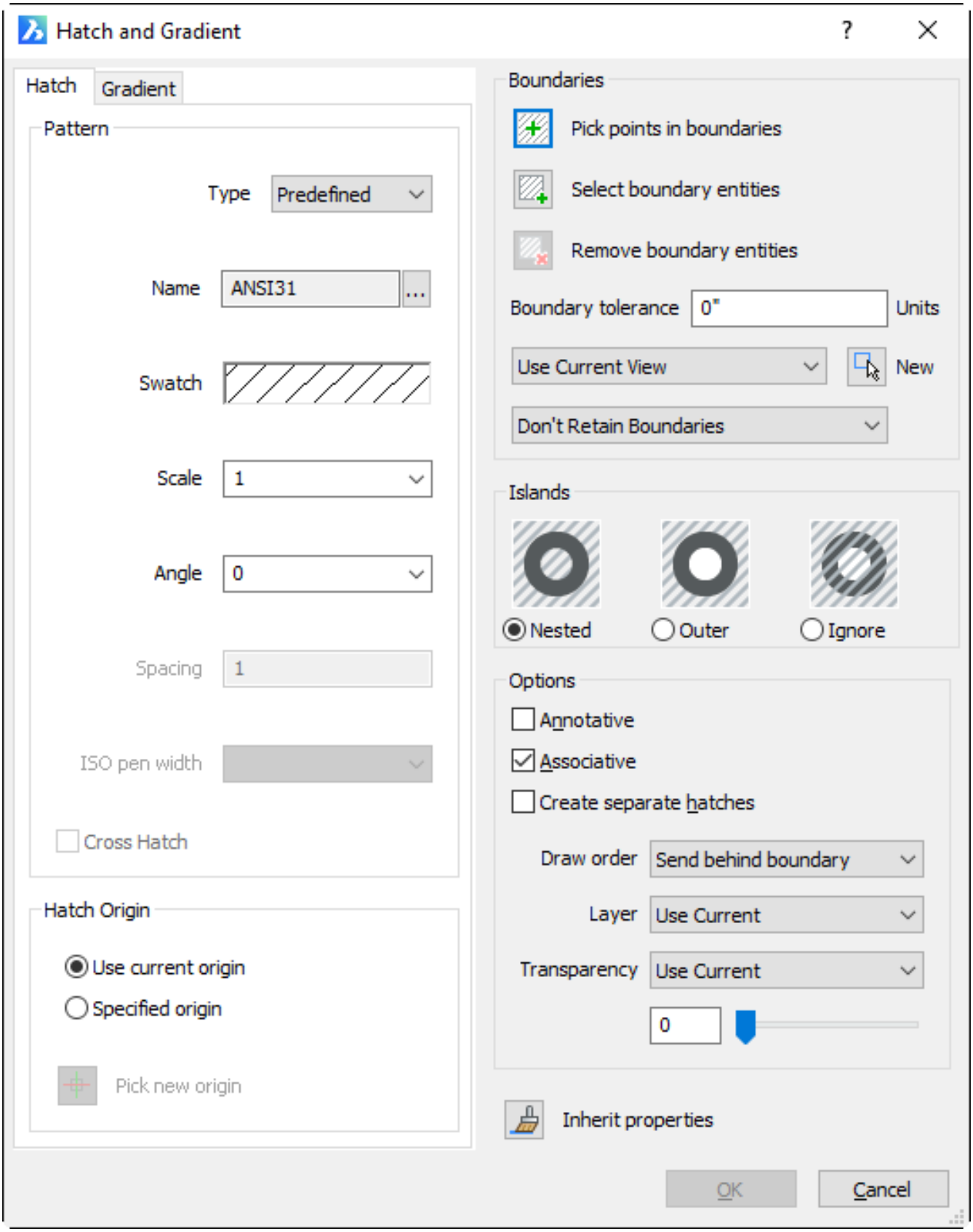
Figure 1.699



Draw Hatches

The Hatch creation tool in BricsCAD is very much like using AutoCAD without the ribbon. It displays a familiar Hatch and Gradient dialog box with the controls you need to create a new hatch or gradient object. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ART/Drawing-Entities08.png})

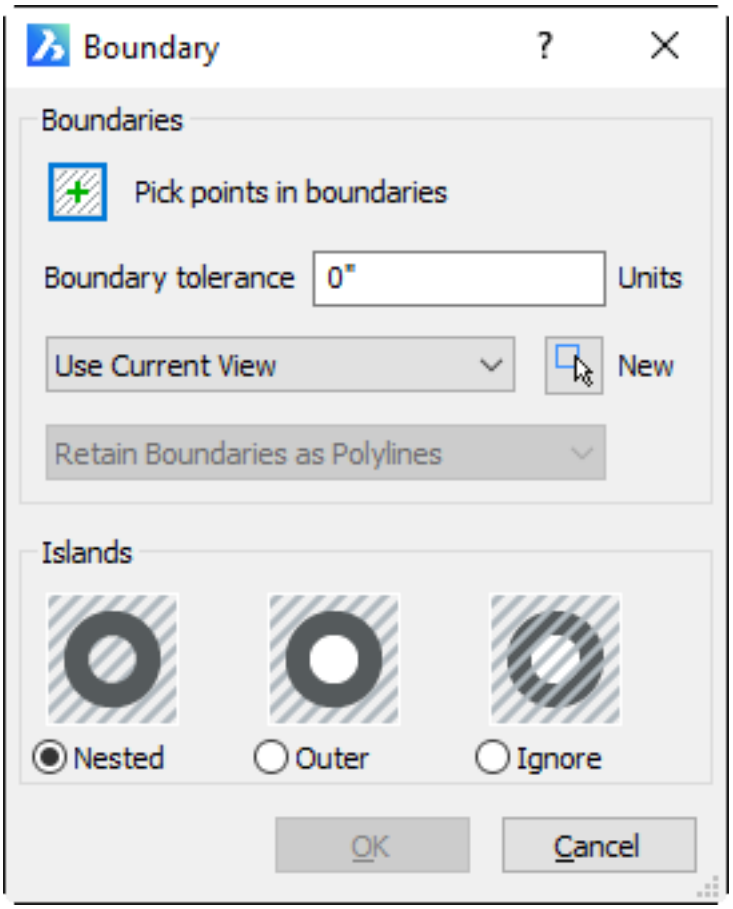
Figure 1.700


Hatch and Gradient
Hatch
Gradient
Pattern
Type
Predefined
Name
ANSI31
Swatch
Scale
1
Angle
0
Spacing
1
ISO pen width
Cross Hatch
Hatch Origin
Use current origin
Specified origin
Pick new origin
Boundaries
Pick points in boundaries
Select boundary entities
Remove boundary entities
Boundary tolerance
0"
Units
Use Current View
New
Don't Retain Boundaries
Islands
Nested
Outer
Ignore
Options
Annotative
Associative
Create separate hatches
Draw order
Send behind boundary
Layer
Use Current
Transparency
Use Current
0
Inherit properties
OK
Cancel

Draw Boundaries

The method to create a boundary in BricsCAD is very similar to AutoCAD. It displays a familiar Boundary dialog box with the controls to create a polygonal boundary. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG Entities09.png})

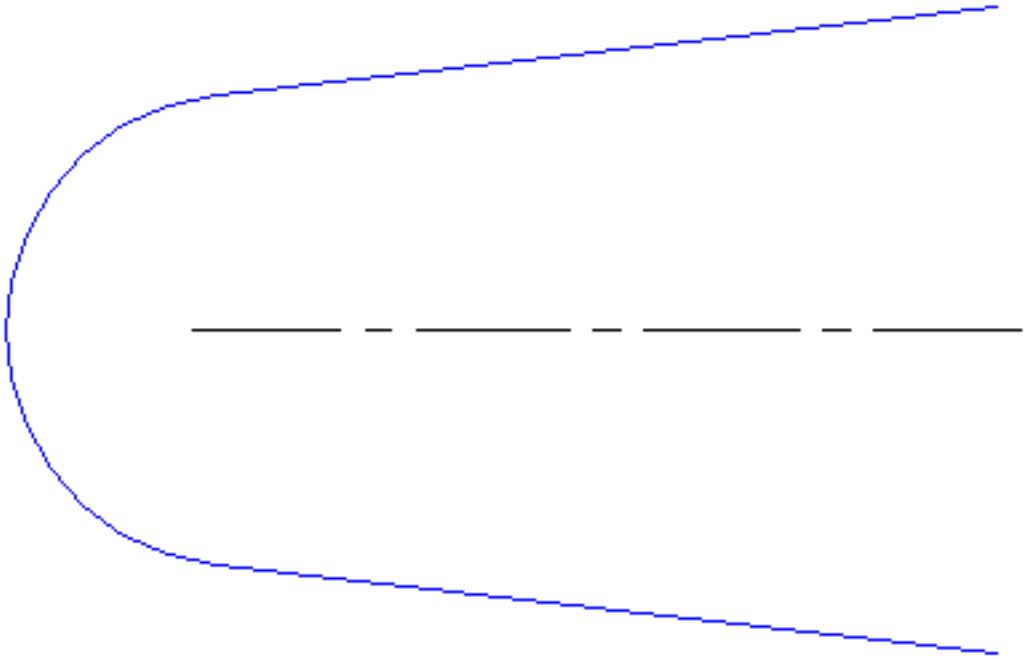
Figure 1.701



Draw Centerlines and Center marks

The process to add centerlines and center marks is as you expect. However, BricsCAD automatically applies a center linetype to centerlines. Note: The CL and CM command aliases don't launch CENTERLINE and CENTERMARK in BricsCAD. In a future post I'll show you how to easily change command aliases. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ART/DrawingEntities10.png})

Figure 1.702



Draw Points

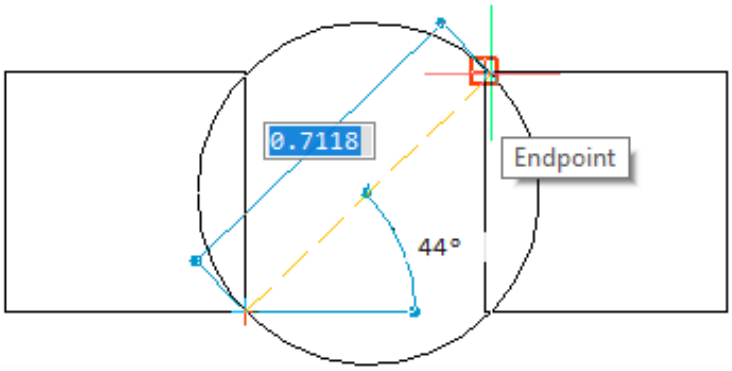
The default process to draw points in BricsCAD is simple and familiar. However, the POINT command in BricsCAD offers a Multiple option. This allows you to insert multiple points without having to relaunch the command each time.

Draw Donuts

You have never seen a donut like this before! By default, it behaves as you expect. But, unlike the donuts of your past, BricsCAD donuts come in many flavors! You're not required to create and place the donut only by its center point. You can choose from familiar circle creation options including 2 Point, 3 Point, and Tangent, Tangent, Radius. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/Entities12.png})

When you use any of those options, you are prompted to specify the width and diameter of the donut. That's more intuitive than specifying the inside and outside radius! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/Entities13.png})

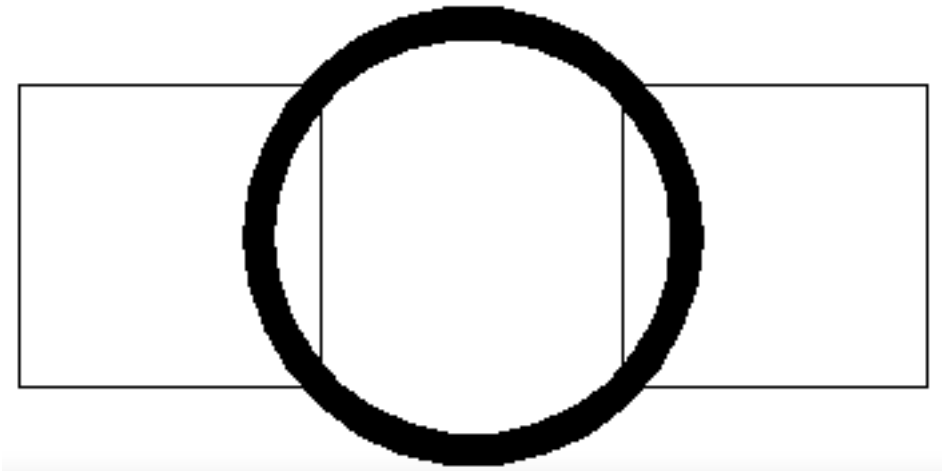
Figure 1.703



Commandline

```
: DONUT
Inside diameter of donut [2 Point/3 Point/Tangent Tangent Radius] <0.5000>:2p
Width of donut <0.2500>: .05
First point on diameter:
Second point on diameter:
```

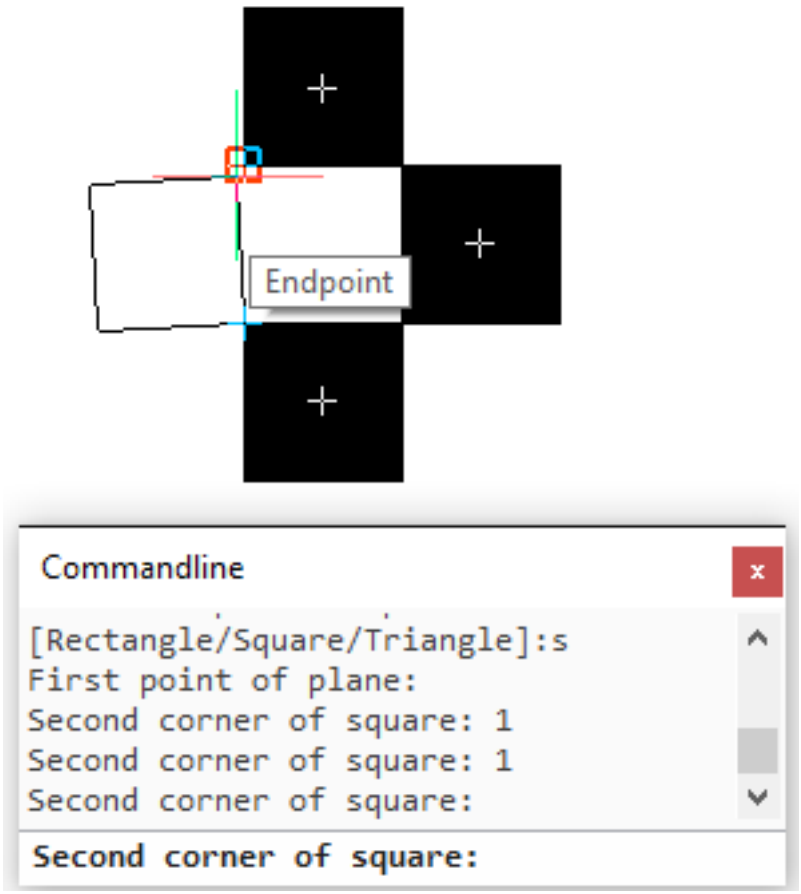
Figure 1.704



Draw Solids

BricsCAD offers three additional optionsin addition to the default expected behavior for drawing solids. You can create rectangular, square, or triangular solids. After you draw one of these shapes, you can create more of the same shapes with minimal clicks. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawinEntities14.png})

Figure 1.705



Draw Xlines and Rays

Xline and Ray work as expected. You might, however, notice the Parallel option for Xlines. It's the same as the Offset option in AutoCAD.

1.17.34 Drawing with Styles

Draw Multilines

The default behavior to draw a multiline is just as you expect. In addition to the familiar command options, BricsCAD offers a Follow option. Follow enables you to create a new multiline that's adjacent and colinear to the last mline you created. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04Workin
Style02.png})

Figure 1.706



Commandline

```
Set end point or [Angle/Length/Follow/Close/Undo]:
:
: MLINE
Current Settings: Justification = Top, Scale = 20.00, Style = Standard
Start of line or [Justification/Scale/STyle/Follow] <Last point>:f
Length of line:
```

Draw Single-line Text

Like AutoCAD®, BricsCAD includes two text tools: Single- and Multi-line. The default process to create single line text is just as you would expect. BricsCAD, however, displays all the available options on the Command line so you can view and specify options with minimal effort. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG Style03.png})

Figure 1.707



Commandline

```
: TEXT
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>:
Height of text <50>:
Rotation angle of text <0>:
Text: NOTES:
```

Draw Multiline Text

You create mtext (multiline text) in BricsCAD much like in AutoCAD. However, after you pick the first corner to place the text box, you have the option to enter specific values for the width and height of the text box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/Style04.png}) And, instead of displaying the text editing tools on the ribbon, BricsCAD displays a Text Formatting toolbar right above the mtext box. It includes familiar text formatting controls such as line spacing, columns, and numbering. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/Style05.png})

Figure 1.708

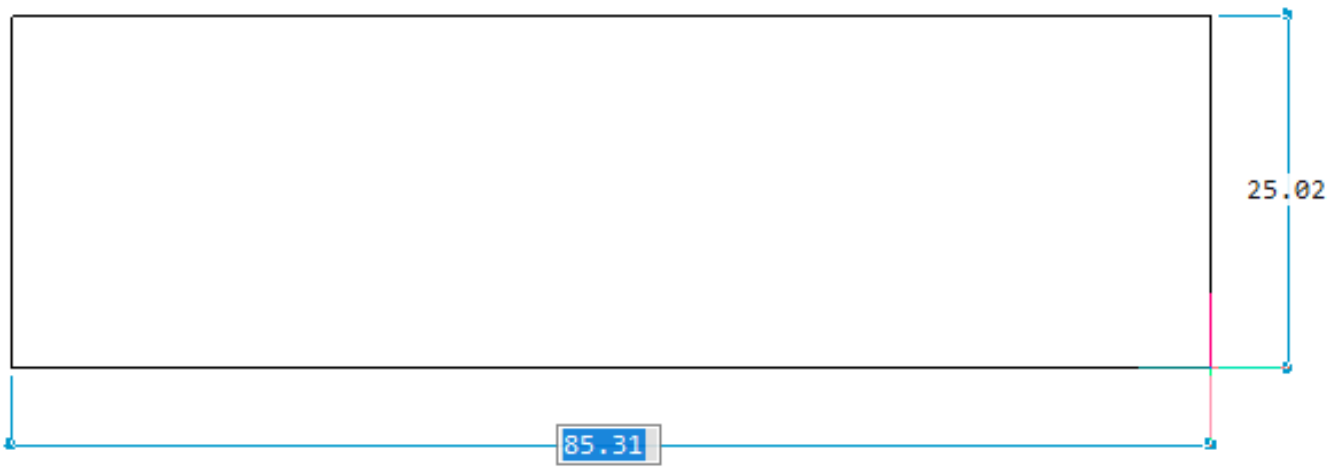
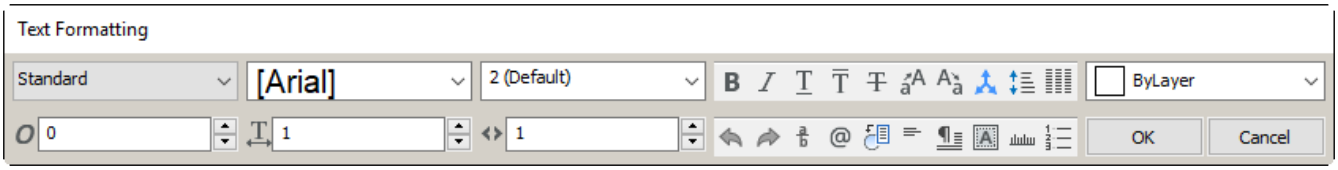


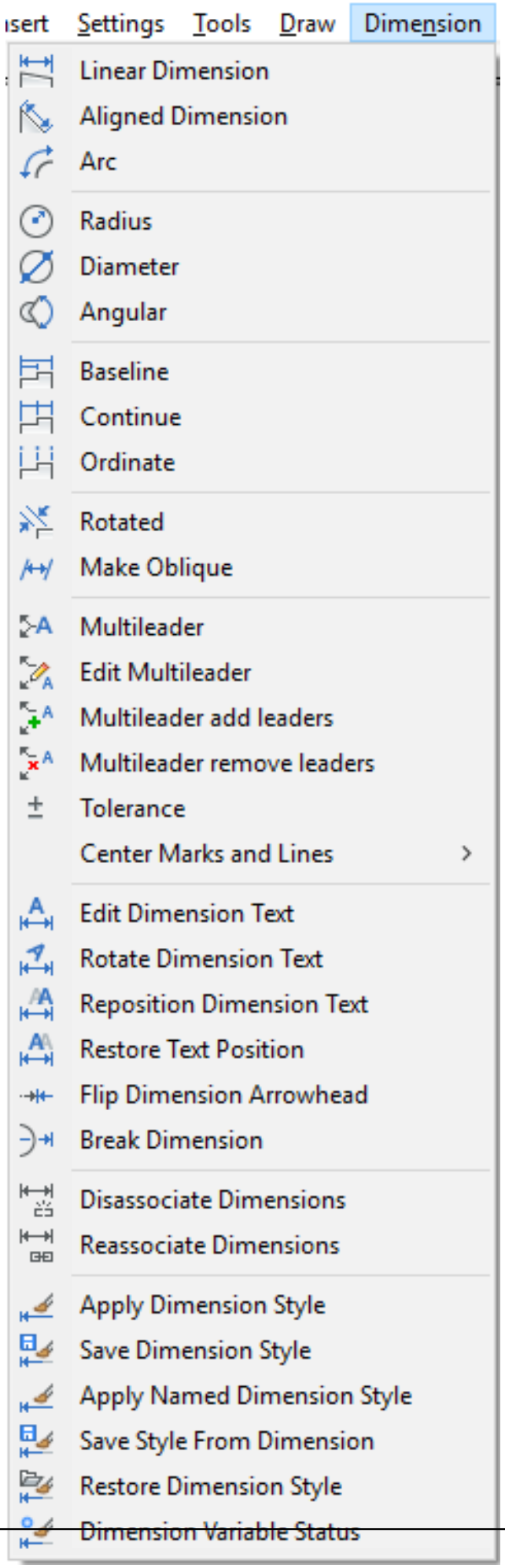
Figure 1.709



Draw Dimensions

BricsCAD includes the most common dimensioning tools, Linear and Aligned, in the Quad. You can access the entire set of familiar dimensions from other locations including the Command line. And, of course, you can add them to the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkWithDimensions/DimStyle06.png})

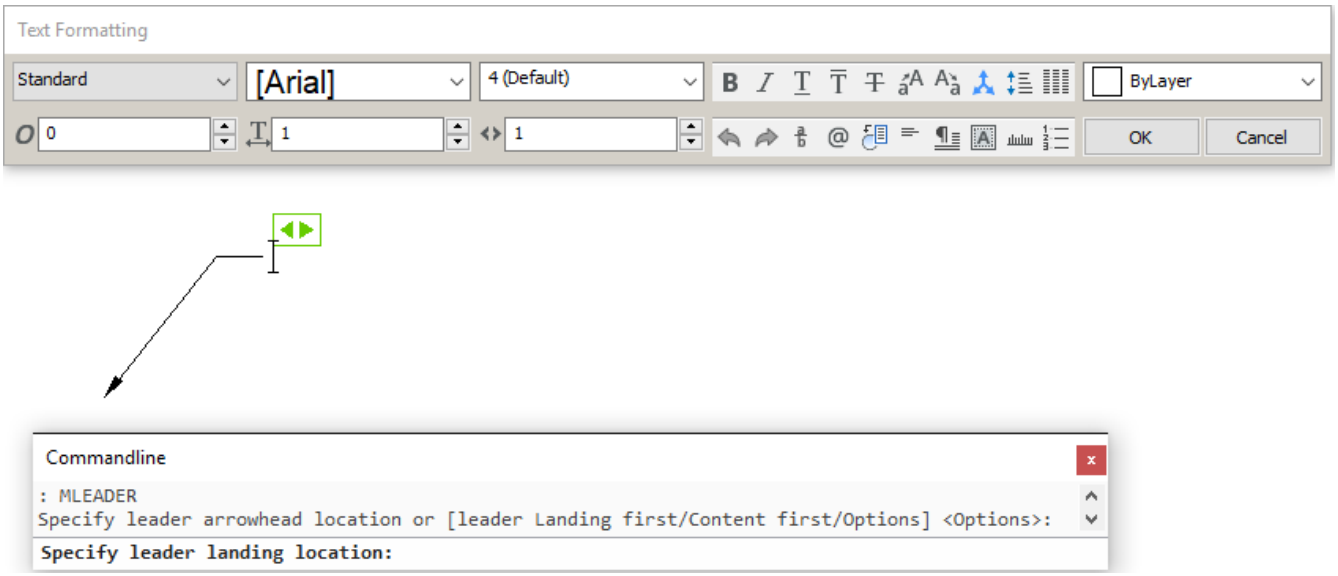
Figure 1.710


Insert
Settings
Tools
Draw
Dimension
Linear Dimension
Aligned Dimension
Arc
Radius
Diameter
Angular
Baseline
Continue
Ordinate
Rotated
Make Oblique
Multileader
Edit Multileader
Multileader add leaders
Multileader remove leaders
Tolerance
Center Marks and Lines
Edit Dimension Text
Rotate Dimension Text
Reposition Dimension Text
Restore Text Position
Flip Dimension Arrowhead
Break Dimension
Disassociate Dimensions
Reassociate Dimensions
Apply Dimension Style
Save Dimension Style
Apply Named Dimension Style
Save Style From Dimension
Restore Dimension Style
Dimension Variable Status

Draw Multileaders

You draw a multileader in BricsCAD very much like you draw one in AutoCAD. First you place the leader lines. And then you enter the text. However, after you pick points to place the leader, BricsCAD displays the Text Formatting tools near the leader rather than on the ribbon. The Text Formatting toolbar includes all the tools you need to format multiline text as part of your leader. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10 Style07.png})

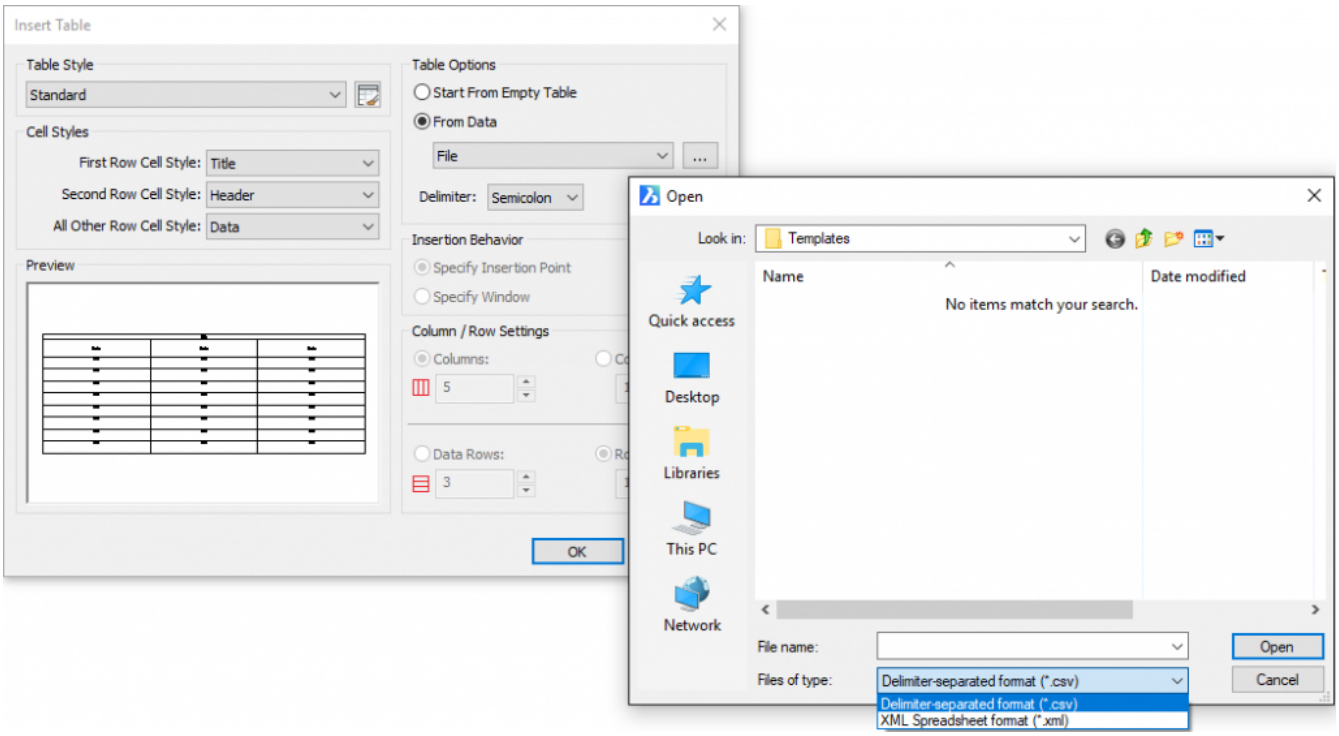
Figure 1.711



Draw Tables

The process to create a table in BricsCAD is similar to AutoCAD LT. The Insert Table dialog box offers the most popular table creation options. It includes the ability to import data from CSV and XML files. Note: BricsCAD does not support data linking directly from Excel spreadsheets. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Style08.png})

Figure 1.712



Defining Styles in the Drawing Explorer

All the drawing tools I covered in this post are dependent on style definitions. They work as you expect from your AutoCAD experience. However, the way you define styles is different. At first glance it may seem overwhelmingly different. But, if you take just a few minutes to explore the style definitions in BricsCAD I think you'll appreciate the differences! In BricsCAD all style definitions are managed from a central location, the Drawing Explorer. You can access the Drawing Explorer from the General menu on the Quad. It's also accessible via other methods including the Command line (EXPLORER). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Drawir Style09.png}) If you use familiar methods such as the MLSYTLE, STYLE, DDIM, MLEADERSTYLE, and TABLESTYLE commands, BricsCAD automatically expands the relevant style information in the Drawing Explorer. In the image below, Text Styles is expanded. Select any of the other styles to view and modify those definitions. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/04WorkingwithHatches/ Style10.png})

Tutorial: BricsCAD Spreadsheet Import Techniques Click here (1.2.9) to watch

Figure 1.713

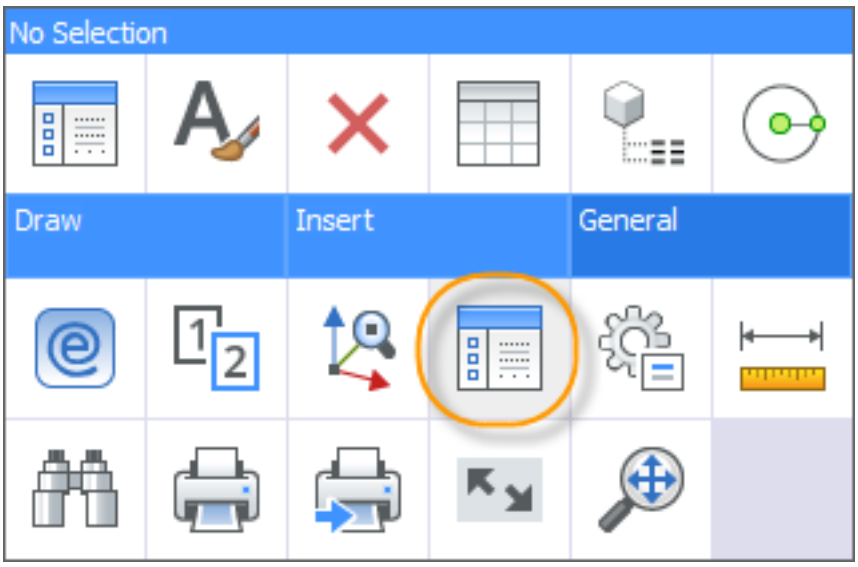
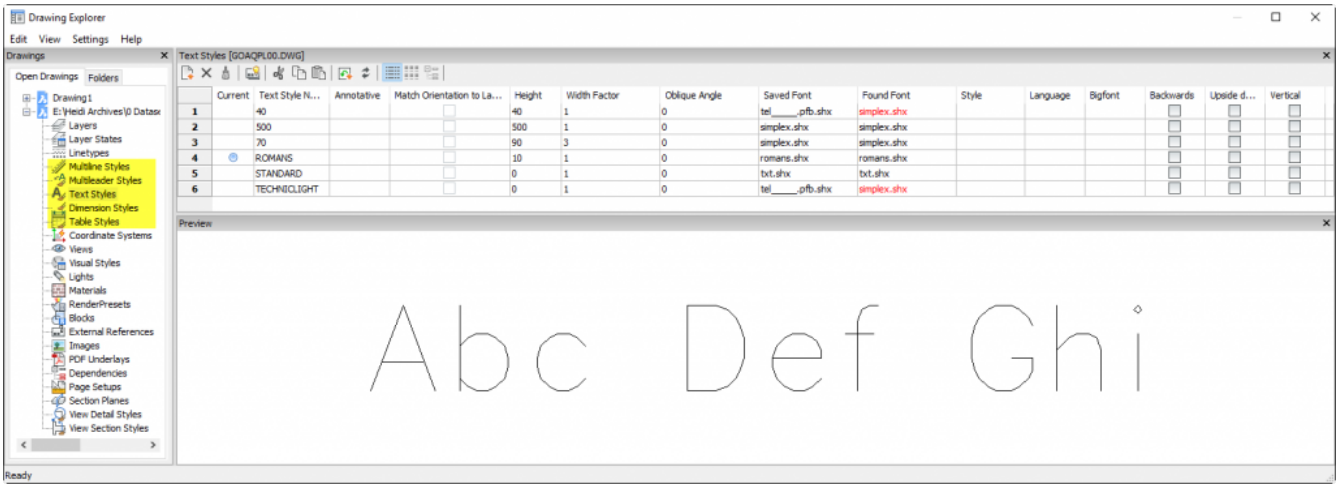


Figure 1.714



1.18 3D Modeling

1.18.1 Direct Modeling Overview

BricsCAD provides advanced easy-to-use functions for working with ACIS solid and surface geometry, including:

- The creation of solid and surface geometry using Extrude (6.13.21) (dmExtrude (6.12.13)) and Revolve (6.26.30) (dmRevolve (6.12.22)) commands applied to 2D contours, 3D curves, and edges of existing solids.
- The creation of fillets (dmFillet (6.12.14)) and chamfers (dmChamfer (6.12.4)) on existing solid and surface geometry.
- Modifying (dmPushPull (6.12.20)) or deletion (Erase (6.13.8)) of fillets and chamfers.
- Direct interactive manipulation of solid and surface faces using PushPull (dmPushPull (6.12.20)), Move (6.21.30) (dmMove (6.12.17)) and Rotate (6.26.34) (dmRotate (6.12.24)) operations.
- The creation or removing of a volume when 2D contour attached to a face of a solid is being pushed or pulled.
- The removing of holes in surfaces by deleting its edges (Erase (6.13.8)).
- Deletion (Erase) and copying (dmCopyFaces (6.12.8)) of protrusion or depression features, such as holes, pockets, extrusions, ribs, and their combinations within the same 3D solid or from one 3D solid to another.

All these operations are performed in interactive mode: the intermediate result of the operation is shown dynamically, according to user input defined by mouse movement. Depending on the value of the DELOBJ (1.13.11) (Delete Entity) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities (see table below).

Value	Result
0	Retain defining entities
1	Delete profile entities; retain paths and guide curves
2	Delete all defining entities
-1	Prompt to delete profile entities; retain paths and guide curves
-2	Prompt to delete all defining entities

The key feature of direct editing capabilities provided by BricsCAD consists in preserving of design intent. Design intent can be expressed by 3D dimensional and geometrical constraints which can be explicitly specified by user (see 3D Constraints (1.19.4)). 3D dimensional constraints (Distance, Radius, and Angle) imposed for faces and edges give user full parametric control over the solid model, different configurations and scales of the model can be obtained simply by changing the parameter of a dimensional constraint. Geometrical constraints (Coincident, Tangent, Parallel and others) define how these changes influence the geometry of the model. However, there is no need to introduce all the geometrical constraints manually since most of them can be recognized automatically by BricsCAD (see Design Intent (1.18.3)) to provide the expected result. Another feature of Direct Editing operations is that they can be easily applied with the Quad cursor menu (1.12.5). This tool provides a set of operations for the currently selected entities. At the same time the Quad provides the other basic solid editing operations such as copy, boolean union and subtract. Direct modeling operations apply to all ACIS solid geometry, whether created in BricsCAD or imported, and can be combined with regular solid modeling operations, for example, SOLIDEDIT (6.27.39).

Figure 1.715



Figure 1.716



1.18.2 Direct Modeling Operations

This section describes direct modeling operations offered by BricsCAD. These operations are available from the Quad (1.12.5) tool. Alternatively, you can use **Direct Modeling** Menu or **Direct Modeling** and **Deform** toolbars: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar30DirectMo (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar30-1Deform.png}) The **Align UCS** tool ( ) launches the UCS (6.30.2) command with the **Face** option selected. If the UCSDETECT (1.13.11) system variable is ON, the UCS is aligned automatically to the currently highlighted face. The UCS-icon changes accordingly. Press the TAB key to select an obscured face. Click the **DUCS** field in the Status Bar (1.12.2) to toggle the dynamic UCS (1.15.18) feature. The following direct modeling operations are available in BricsCAD:

Icon

Command

Applies to

Description  dmPushPull (6.12.20) Any face of a 3D solid or surface Adds volume to, or removes volume from the solid by cursor movement or direct distance input.  dmMove (6.12.17) Cylindrical, spherical, conical and toroidal faces of a 3D solid or surface. Linear and circular edges of a 3D solid or surface. Whole 3D solids, surfaces, and other entities. Moves the selected geometry using a vector in the XY- plane of the current UCS. When a face or an edge of a solid or surface is moved, adjacent faces and edges are adjusted to preserve the correct solid/surface topology.  dmRotate (6.12.24) Planar, cylindrical, spherical, conical and toroidal faces of a 3D solid or surface Whole 3D solids, surfaces, and other entities. Rotates the selected geometry around an axis. When a face of a solid or surface is rotated, adjacent faces and edges are adjusted to preserve the correct solid/surface topology.  dmSimplify (6.12.27) 3D solids, surfaces, and their faces Simplifies geometry and topology of selected entities or their subentities.  dmStitch (6.12.28) Surfaces Stitches disjoint surfaces into a single 3D solid or a single surface.  dmAudit (6.12.2) 3D solids and surfaces Checks and fixes geometrical and topological errors in ACIS solids and surfaces.  UCS (6.30.2) Planar faces of 3D solids and surfaces Aligns the UCS to the selected face. dmExtrude (6.12.13) Closed 2D entities and detected closed boundaries* Creates solids by extruding the selected 2D entities or boundary.  Planar faces of 3D solids and surfaces. Adds volume to, or removes volume from the solid. Optionally the added volume is created as a separate solid Closed 2D entities and detected closed boundaries on a planar face of a solid Creates solids or surfaces by extruding the selected 2D entities or boundary. Optionally the created volume is unified with or subtracted from the solid. dmRevolve (6.12.22) Closed 2D entities and detected closed boundaries* Creates a solid or a surface by revolving a 2D entity about an axis.  Planar faces of solids and surfaces Adds volume to, or removes volume from the solid. Optionally the added volume is created as a separate solid. Closed 2D entities and detected closed boundaries on a planar face of a solid Creates solids or surfaces by revolving the selected 2D entities or boundary. Optionally the created volume is unified with or subtracted from the solid.  xFaces (6.34.4) Faces of 3D solids and surfaces Extracts a set of 3D solid or surface faces as a new 3D surface object.  dmCopyFaces (6.12.8) A set of 3D solid faces Copies a feature (protrusion or depression) represented by the selected faces from one 3D solid to another, or to a different location on the same 3D solid. dmThicken (6.12.31) Faces of 3D solids and surfaces Creates new 3D solids by thickening faces by a given value.  Surfaces Creates a new 3D solid by thickening the whole surface. Wire entities (lines, polylines, circles, ellipses, arcs, helices, splines) Creates a new 3D solid by sweeping a circle along the selected wire.  dmFillet (6.12.14) Sharp edges of a 3D solid or surface. Creates a fillet between adjacent faces sharing a sharp edge(s).  dmChamfer (6.12.4) Sharp edges of a 3D solid or surface Creates a chamfer between adjacent faces sharing a sharp edge(s). Erase (6.13.8) Faces of a solid or a surface. Whole 3D solids and surfaces. Deletes the selected entities. When faces of a solid entity are deleted, the gap is filled by extending the adjacent faces. When faces of a surface entity are deleted, the gap is filled only if the faces form a fillet.  Free edges of a surface (a surface edge is called free if it has only one adjacent face) If the selected free edges belong to one or more holes in the surface, then these holes are deleted and the gap if filled automatically. Edges between coplanar faces. Deletes the edges between coplanar faces, thus unifying the faces.  dmExtrude (6.12.13) 3D curves and edges of 3D solids and surfaces Creates surfaces by extruding the selected curves and edges in a particular direction, which is defined heuristically, but can also be set by the user.  dmRevolve (6.12.22) 3D curves and edges of 3D solids and surfaces Creates surfaces by revolving ~~the selected curves and edges around the selected axis.~~  ~~dmDeformPoint (6.12.11) Faces of a 3D solid or a~~ surface Deforms the selected set of faces by moving a point lying on one of them  dmDeformMove (6.12.10)

Tutorial: 3D Direct Modeling Basic Commands Click here (1.6.26) to watch

Selection Preview

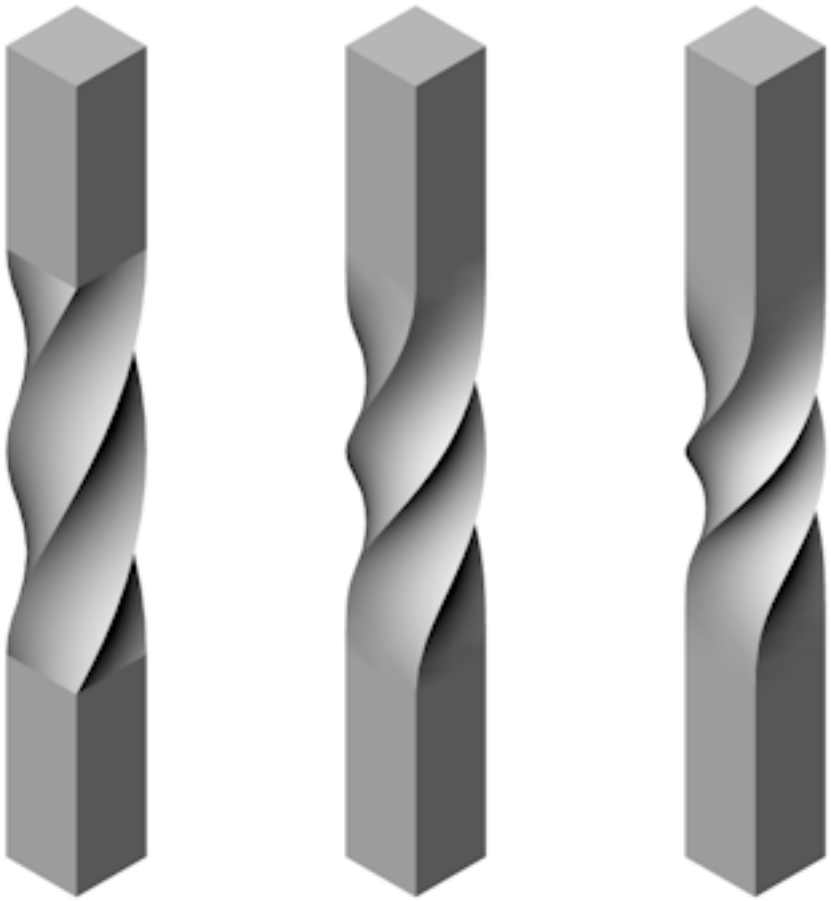
When the Quad cursor menu (1.12.5) is active, the SELECTIONPREVIEW (1.13.11) system variable is neglected, unless a command prompts to select an entity (see Selection Settings (1.23.5)).

Selecting geometry

- Hit the TAB key to select obscured geometry.
- The SELECTIONMODES (1.13.11) system variable controls which subentities highlight: edges, faces and/or detected boundaries.
- If a direct modeling command is entered at the command prompt or selected in a menu or toolbar, you need to press the Enter key (or right click) to conclude the selection procedure.
- If launched from the Quad cursor menu (1.12.5), the command is executed when you click to confirm the selection of the currently highlighted geometry.
- Window selection (1.23.6): Pressing the CTRL key during window selection allows to select faces or edges of 3D solids and surfaces. An icon next to the cursor indicates what will be selected.
- The dmSelect (6.12.26) command allows to select edges and faces based on their geometrical and topological properties:
  - Edges of a particular length;
  - Edges forming a concave/convex network;
  - Edge loops representing the border of a set of connected faces;
  - Faces of a particular area;
  - Smoothly connected faces (G1 - tangent faces, or G2 - curvature continuity); (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/DmTwis
    G0: No continuity (left), G1: Tangent Faces (middle) and G2: Curvature continuity (right)
  - Fillets of a particular radius;
  - Cylindrical protrusions of a particular radius;
  - Protrusion or depression features, such as holes, pockets, extrusions or ribs.

The value of a particular length, area or radius can be taken from a selected edge/face.

Figure 1.717



To control dimensions

When using the dmPushPull (6.12.20) command a dynamic dimension displays. Make sure the Quad (1.12.5) and Dynamic Dimensions (1.15.9) or ON.

1. Hover the cursor over the face you want to move. The face highlights.
2. Choose **PushPull** () in **Direct Modeling** command group of the Quad cursor menu (1.12.5). The selected face is attached to the cursor and a dynamic dimension field displays the relative distance. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/PushPullRel
3. (Option) repeatedly hit the TAB key to see the distance with respect to an internal face, parallel to the selected face (left) or the bottom face (right). Press and hold the SHIFT key while hitting the TAB key to reverse the display order of reference faces. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingS
4. Type a value in the dynamic dimension field, then press Enter.

Figure 1.718

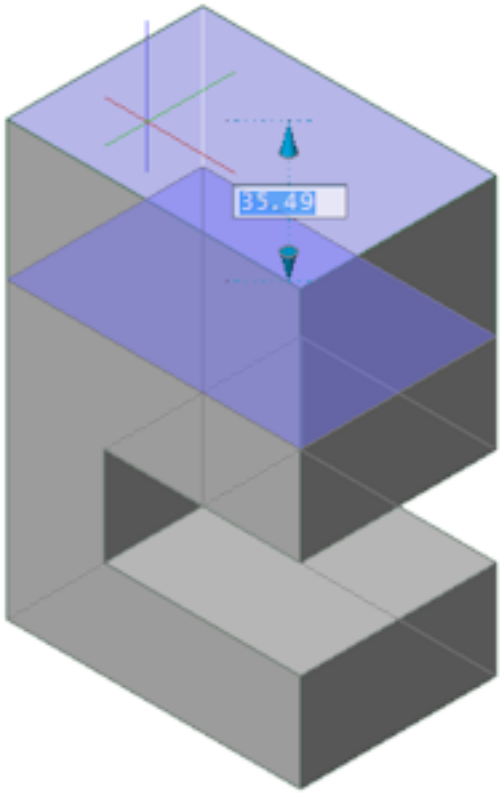


Figure 1.719

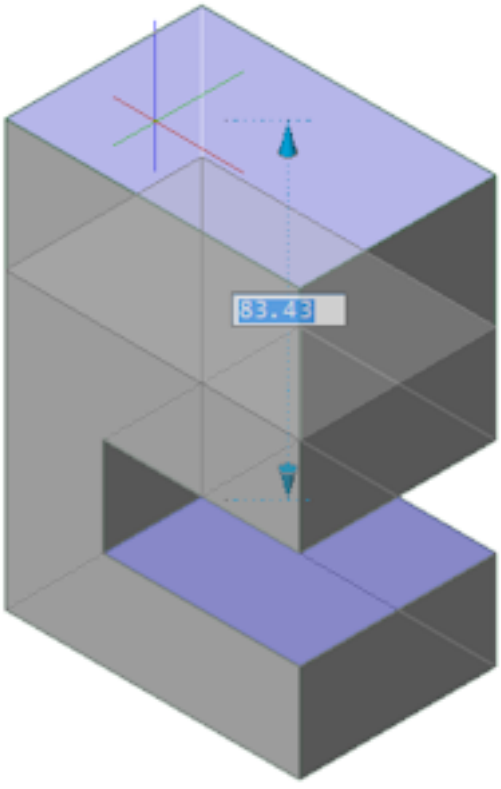
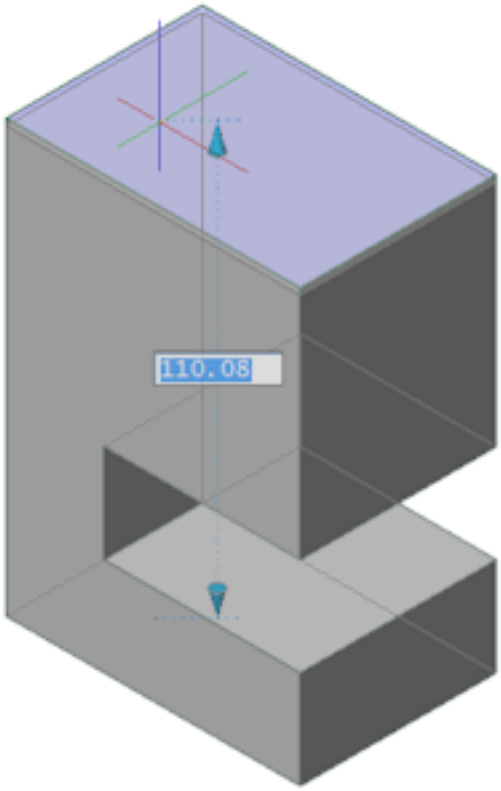


Figure 1.720



Making holes

Typically holes are made using the SUBTRACT (6.27.56) command. When push-pulling a closed 2D entity that lies on a face of a solid, a hole is created when you push the 2D entity through the solid.

To make a (circular) hole through a solid

1. Make sure Dynamic UCS (1.15.18) is active.
2. Launch the Circle (1.17.15) command.
3. Move the cursor to the face of the solid you wan to create the circular hole in. The UCS automatically aligns to the face the cursor is on.
4. Draw a circle on the face of the solid.
5. Hover over the circle, then choose **Extrude** ( ) in the **Model** command group of the Quad cursor menu (1.12.5). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03D
6. Push the circle inside the solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/1
7. Do one of the following:
   - Type a distance in the dynamic field to define the depth of the hole.
   - Push the circle through the solid to create a through-hole.
8. (option) Choose **Push/Pull** ( ) in the **Model** command group of the Quad cursor menu (1.12.5) to modify the radius of the hole.
9. (option) Use the dmDistance3d (6.12.12) command to apply distance constraints between the side faces of the solids and the center of the cylindrical surface of the hole to control the position of the hole.
10. (option) Use the dmRadius3d (6.12.21) command to control the radius of the hole.

Figure 1.721

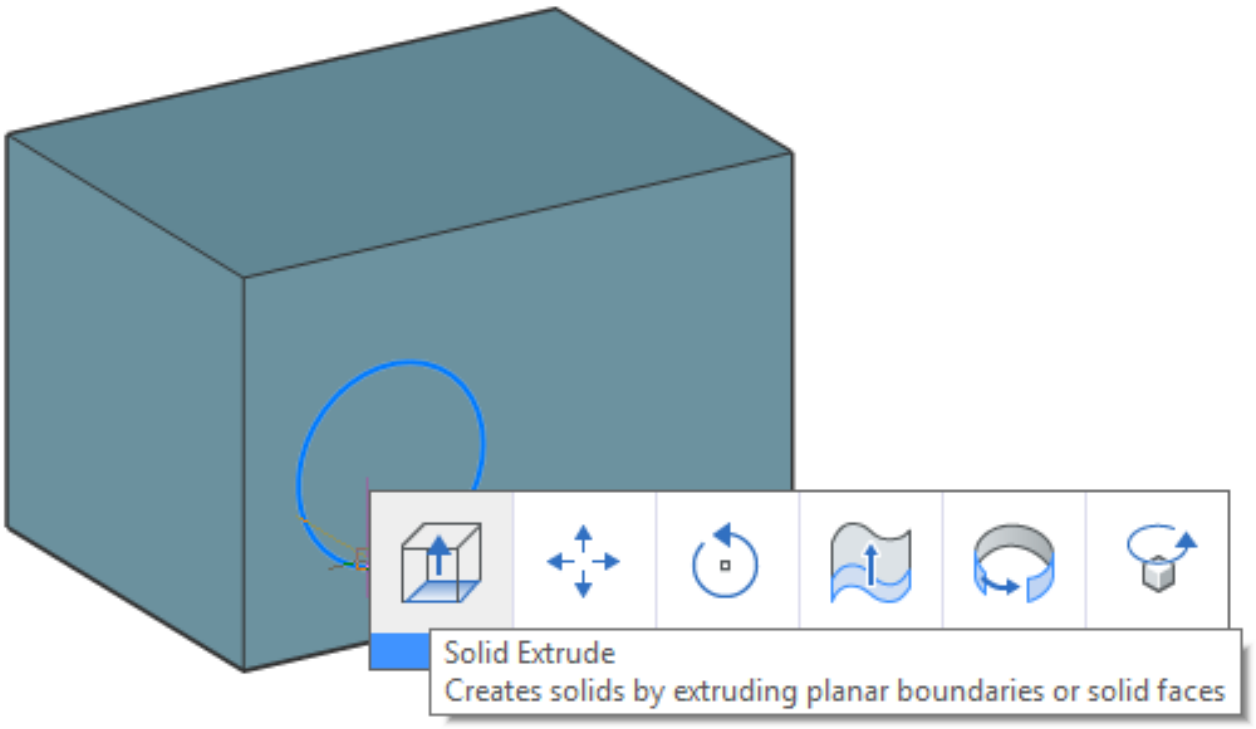


Figure 1.722

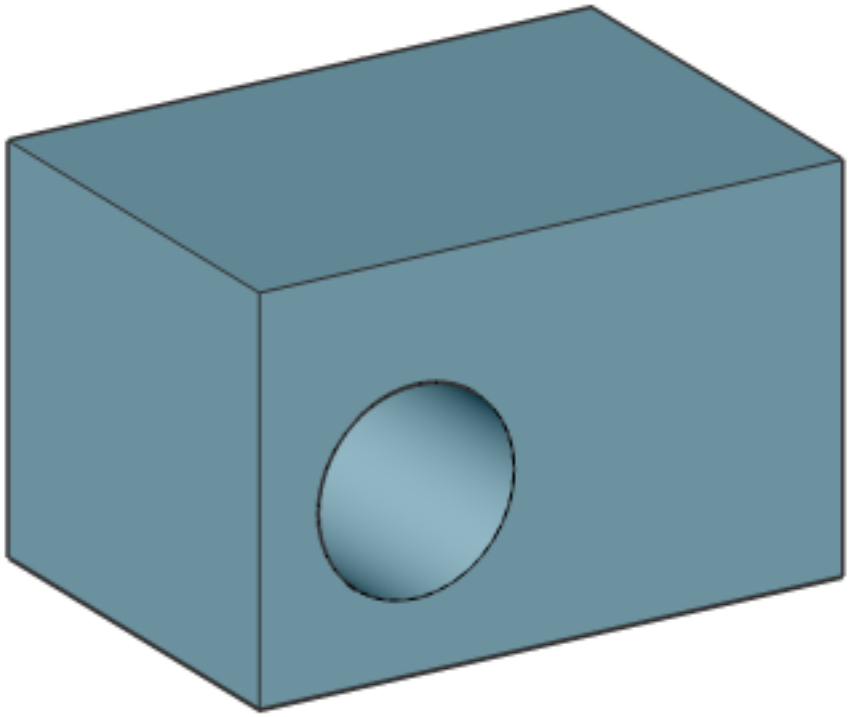
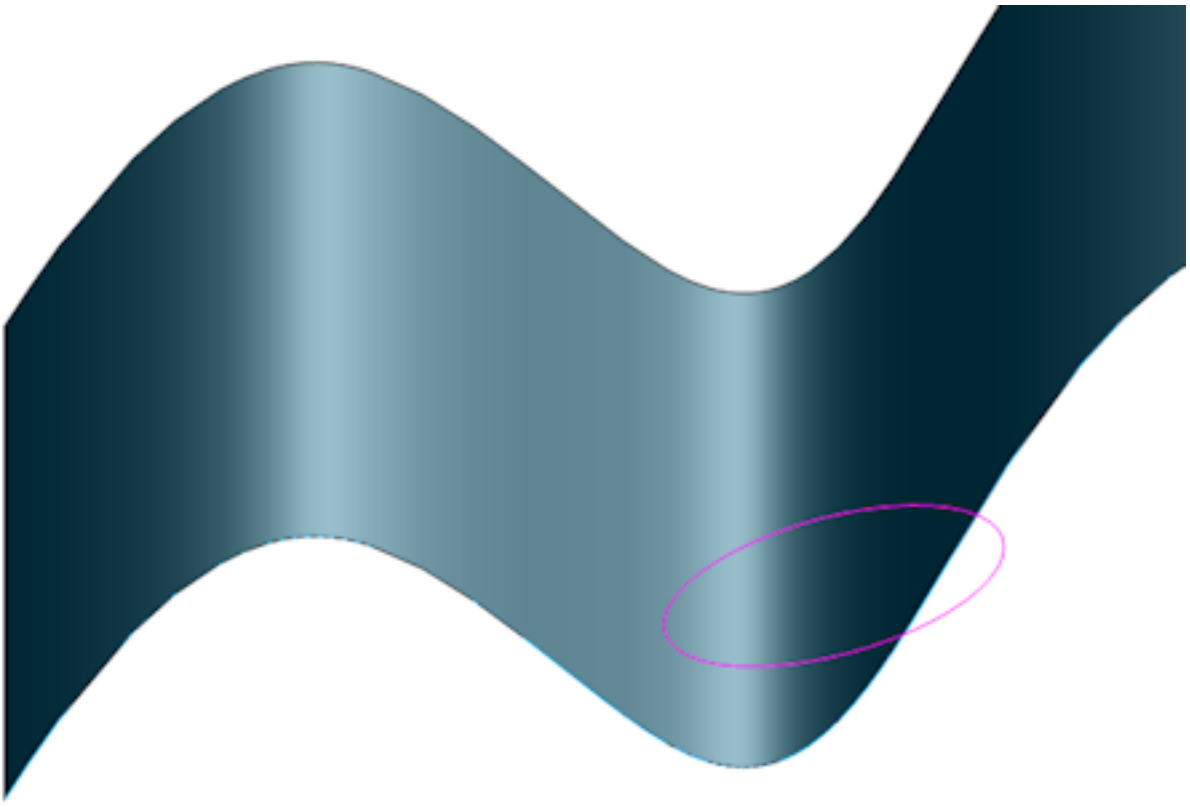


Figure 1.723



To make a hole in a surface

1. Create a 2D closed entity, e.g. an ellipse. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/
2. Hover over the 2D entity.
3. Choose **Extrude** ( ) in the **Model** command group of the Quad cursor menu (1.12.5).
4. Extrude the 2D entity in the direction of the surface. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
5. Press the CTRL key twice to select the **Subtract** option of the DmExtrude (6.12.13) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/HoleInSurfac
6. Click to create the opening.

Figure 1.724

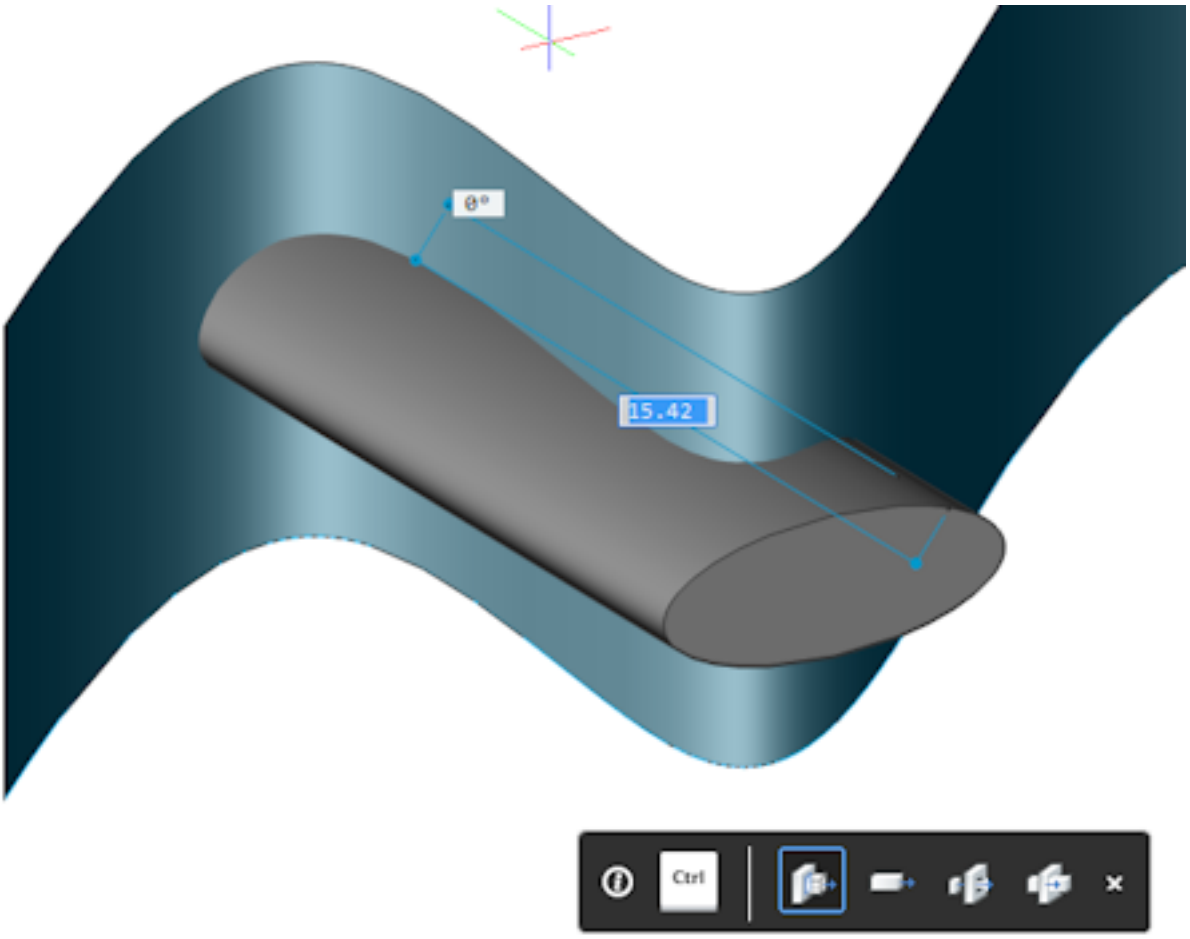
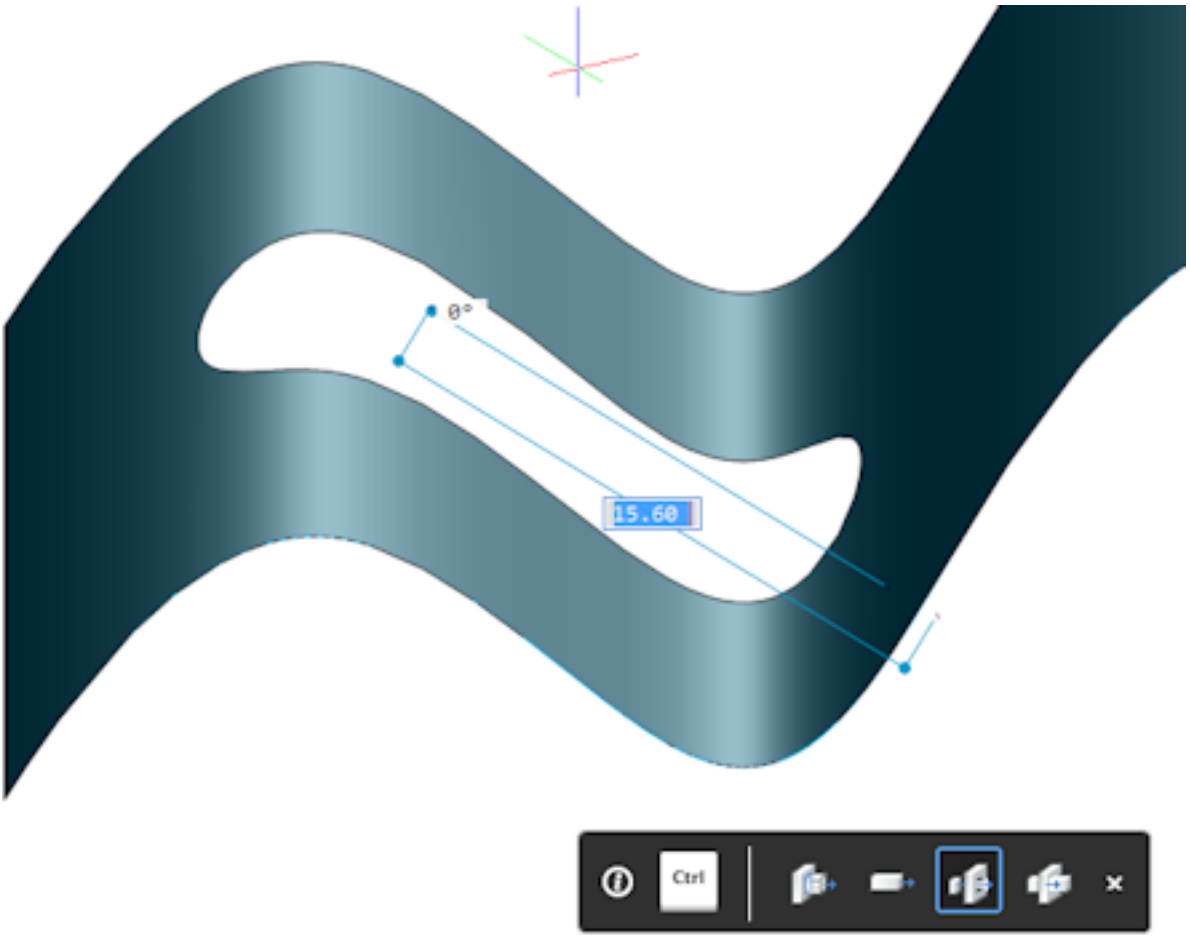


Figure 1.725



Deleting features

To delete a hole, you must delete all internal faces and/or surfaces of the hole.

To delete a circular hole

1. Hover over the cylindrical surface of the hole.
2. Choose **Delete** ( ) in the **Model** command group of theQuad cursor menu (1.12.5).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/
To delete a rectangular hole

1. Hover over one of the visible internal faces of the hole. The face highlights.
2. Choose **PushPull** ( ) in the **Model** command group of theQuad cursor menu (1.12.5) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/DeleteRecta
3. Drag the selected face to close the hole.

To delete a niche or recess

1. Hover over the back (bottom) face of the recess. If the face is obscured in the current view, hit the TAB key untill the face highlights.
2. Choose **PushPull** ( ) in the **Model** command group of theQuad cursor menu (1.12.5). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/DeleteReces
3. Drag the face outside the solid.

To delete a hole in a surface

1. Hover over the edge of the opening. The edge highlights.
2. Choose Erase (6.13.8) ( ) in the **Model** command group of theQuad cursor menu (1.12.5)... (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/DeleteHoleI
to delete a depression (hole or pocket) or a protrusion feature (extrusion or rib).
3. Select one or more faces of the feature you want to delete.
4. Click the **Protrusion** ( ) or **Depression** ( ) tool button in the **Select** panel of the **Solid** ribbon tab or in the **Select** command group of the Quad (1.12.5).
5. Click the **Erase** ( ) tool button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/

Figure 1.726

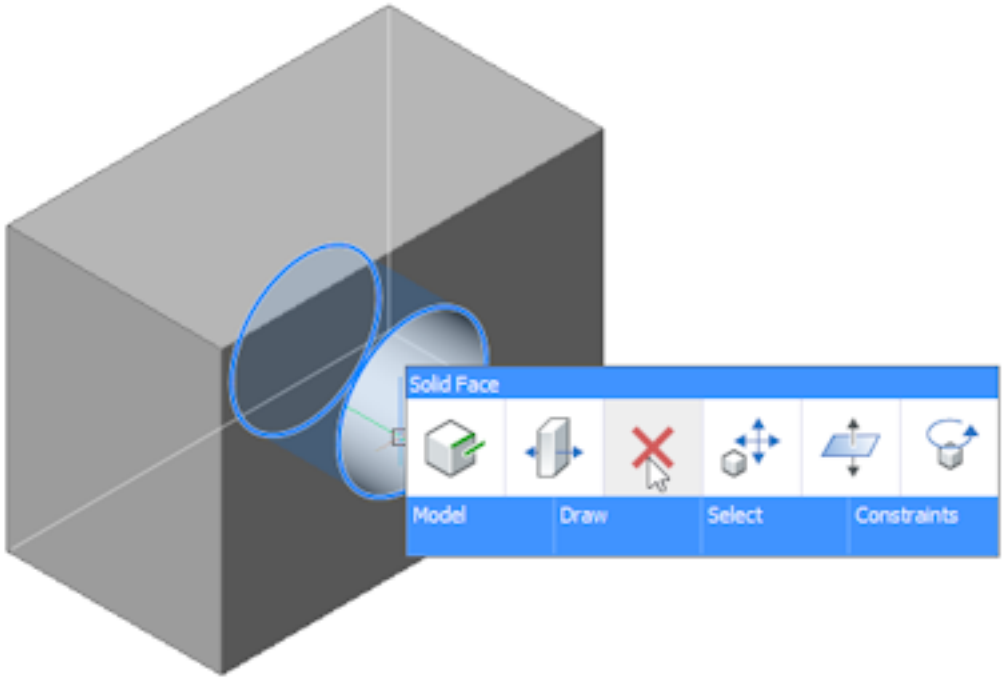


Figure 1.727

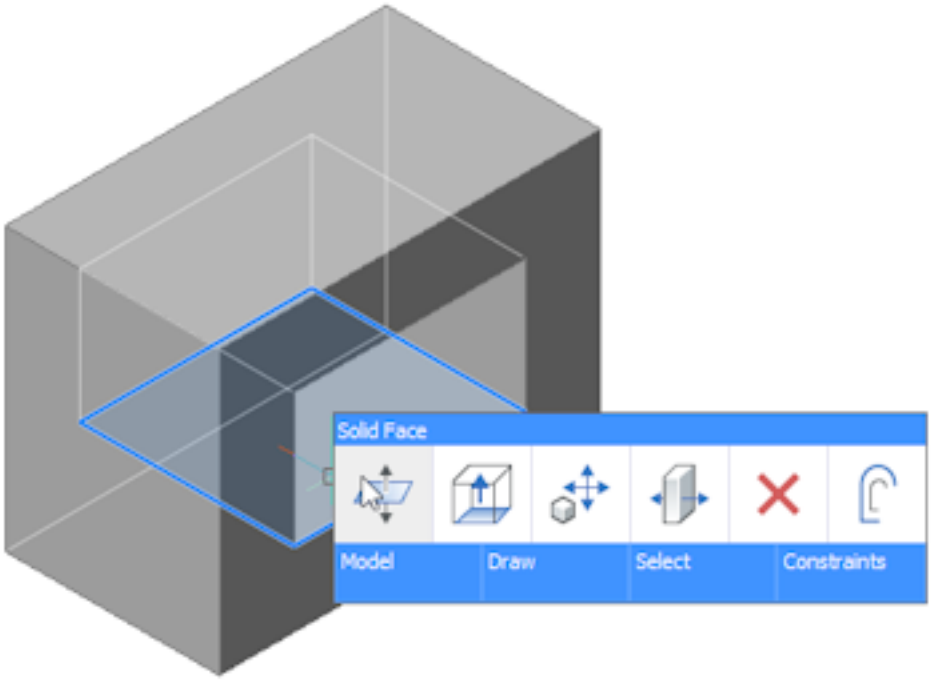


Figure 1.728

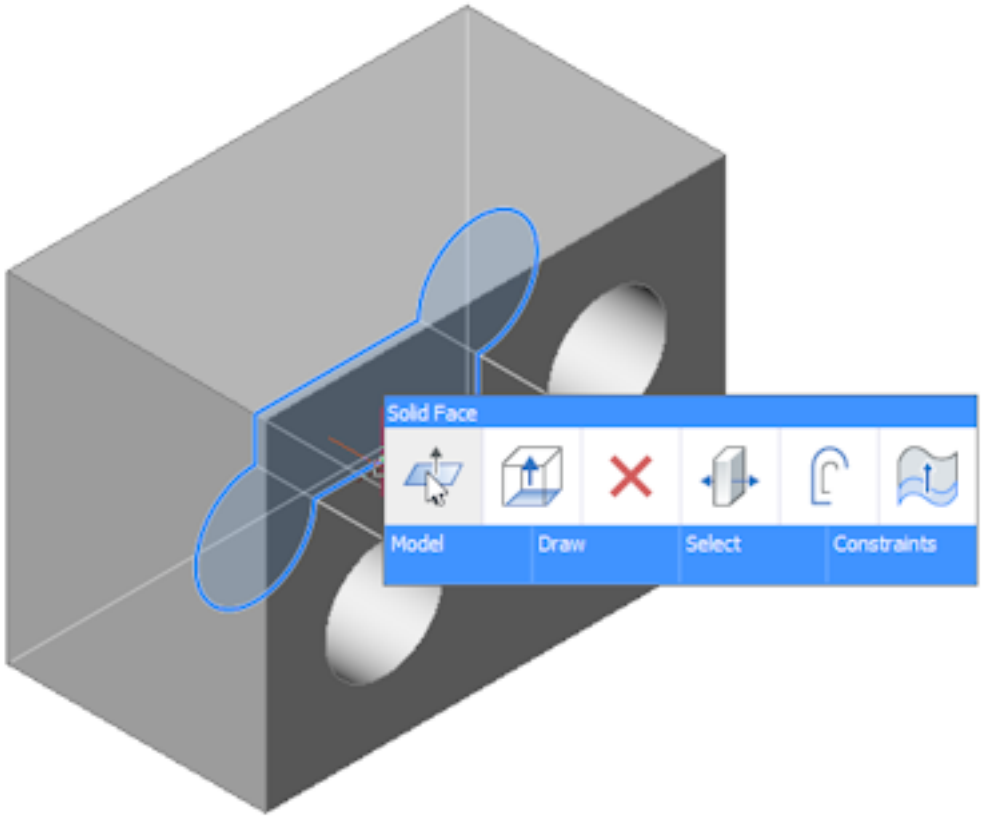


Figure 1.729

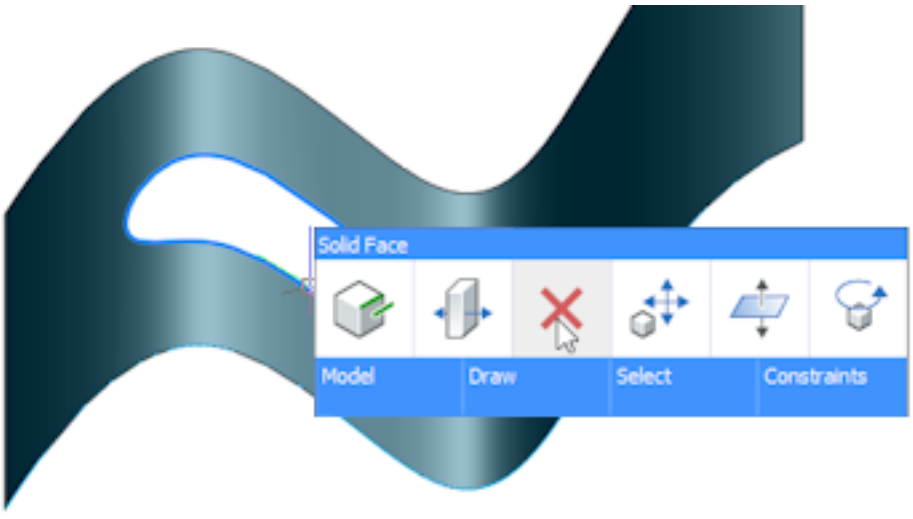
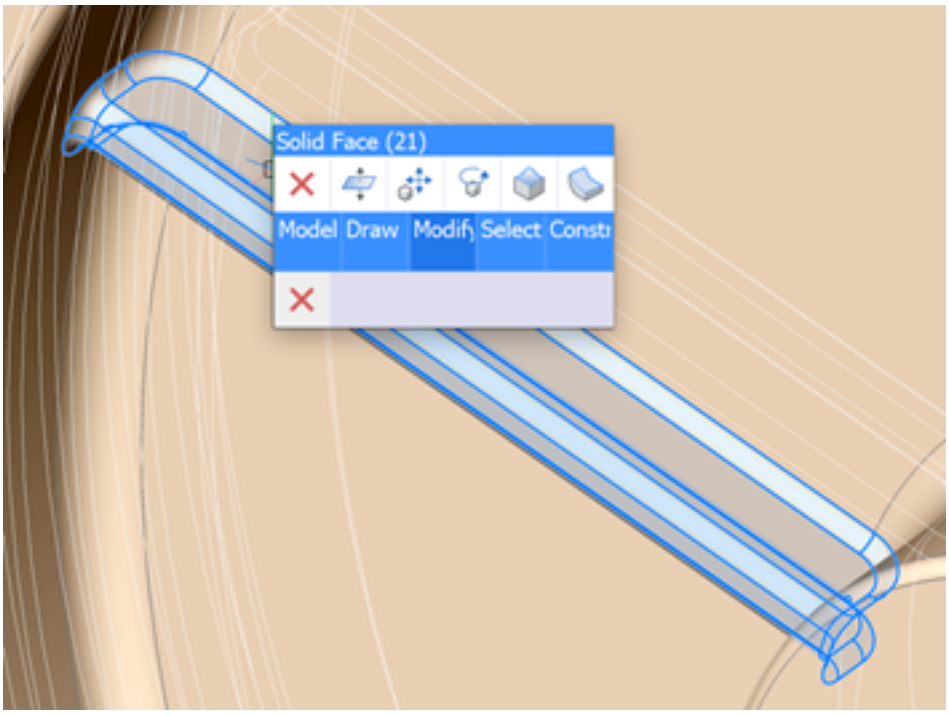


Figure 1.730



To remove edges between coplanar faces

When two or more faces are made coplanar, e.g. using the PushPull command, it might be necessary to remove the shared edges.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/
Two depressed faces ...	... moved to the top face, creating three coplanar faces.

1. Select the three faces.
2. Choose **Delete** (✕) in the **Model** command group in the Quad (1.12.5). (\vref {fig:C:/Dev/Task-2448/images/staticbricsys
   The shared edges are deleted and the three selected faces are unified. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys

Figure 1.731

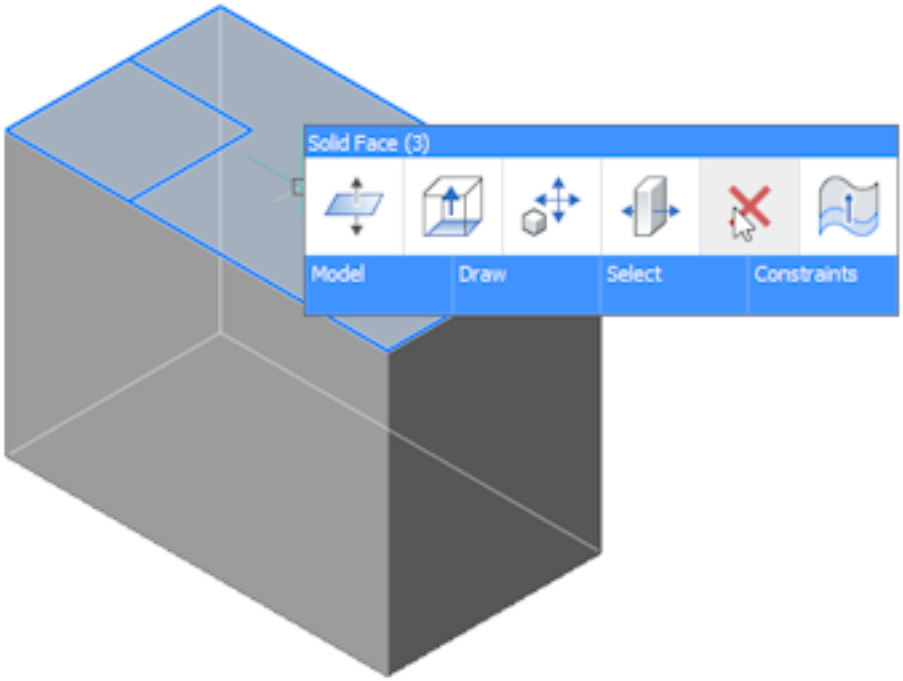
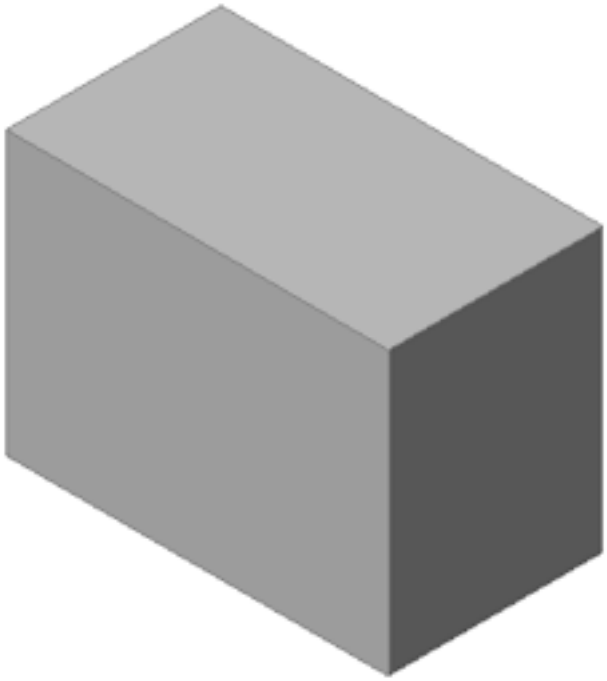


Figure 1.732



Copying Features

the dmCopyFaces (6.12.8) command allows to copy one or more features of a 3D solid, such as depressions (holes, grooves, pockets) or protrusions (extrusions, ribs) to a new location on the same or a different 3D solid. To copy a feature do the following:

1. Select all faces of the entire feature(s) (see Selecting Geometry (1.18.2)).
2. Make sure dynamic UCS (1.15.18) is enabled. It is recommend to always keep DUCS enabled when using the Direct Modeling toolset.
3. Click the **Copy Faces** ( ) tool button:
   - on the ribbon: **Solid | Edit | Copy Faces** .
   - on the **Direct Modeling** toolbar.
   - in the menu bar: **Model | Direct Modeling | Copy Faces You are prompted:** Enter base point:
4. Specify the base point. **You are prompted:** Select insertion point or [Rotate/mUltiple/MOde]:
5. Optionally:
   - Use the **Rotate** option to rotate the faces around Z axis of the UCS.
   - Use the **Multiple** option to insert the multiple copies of the selected feature.
6. Specify the insertion point. The feature is copied from the base point to the insertion point taking into account the dynamic UCS used to select both points.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar30-1Deform.png}) Select the faces of the entire feature	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar30-1Deform.png}) Specify base point
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar30-1Deform.png}) Specify insertion point	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar30-1Deform.png}) The feature is copied

Deformable modeling

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar30-1Deform.png}) Using the tools on the **Deform** toolbar, you can create complex aesthetic shapes from simpler ones by deforming a set of adjacent faces of 3d solid or a surface. The faces are always deformed under the following conditions:

- Their external boundary (= edges, which are adjacent to only one of the selected faces) remain unchanged;
- Their internal boundaries (= edges, which are adjacent to two of the selected faces) is deformed together with the faces;
- The initial continuity between deformed faces (G1 – tangent faces, or G2 – curvature continuity) is kept during deformation.

For a twist deformation it is possible to specify the continuity between the deformed and the fixed portion of the entities. Deformation can be defined in several ways:

- By twisting the whole entity or its portion;
- By moving a point lying on a face in any direction in 3D space;
- By moving or rotating one or more edges;
- By replacing an edge with a given 3D curve.

Figure 1.733



Figure 1.734



Tutorial: Use surface modeling and deformable modeling on imported geometry Click here (1.6.35) to watch

Figure 1.735

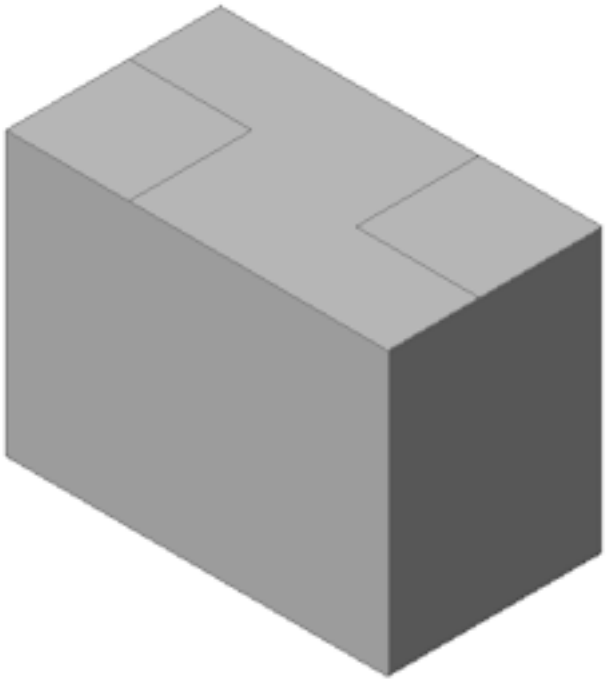


Figure 1.736

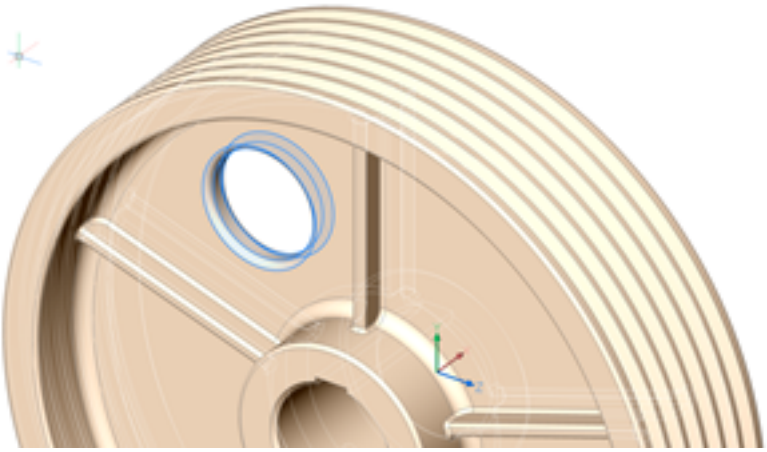


Figure 1.737

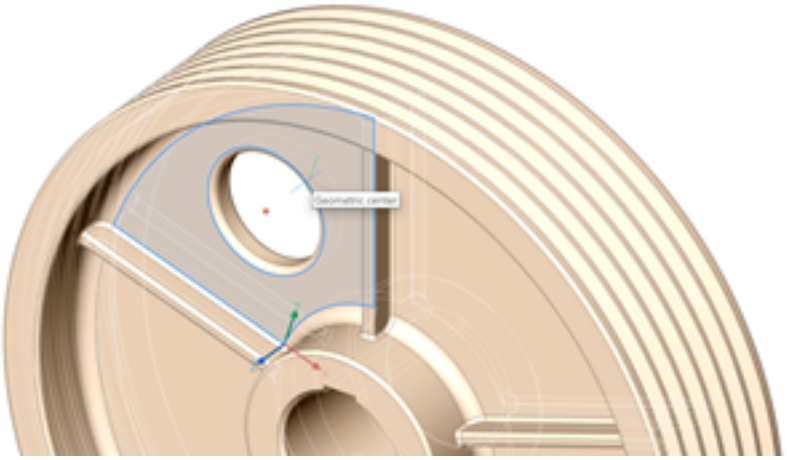


Figure 1.738

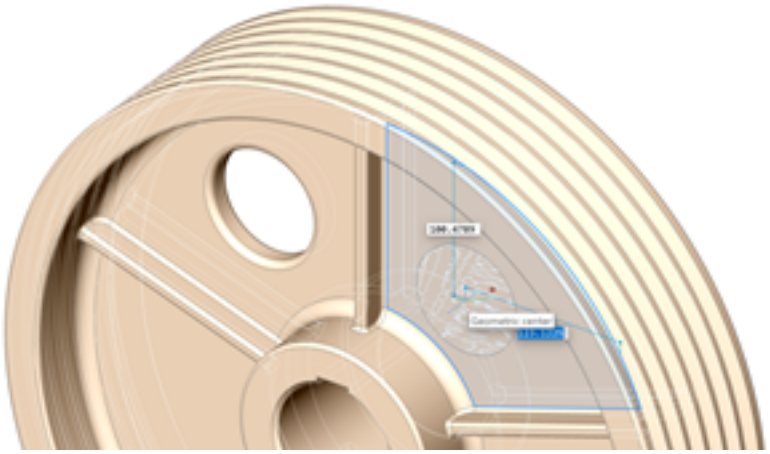


Figure 1.739

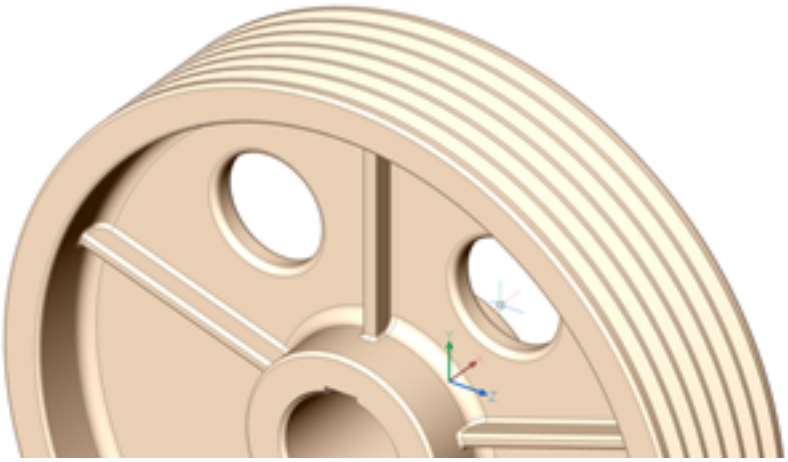


Figure 1.740

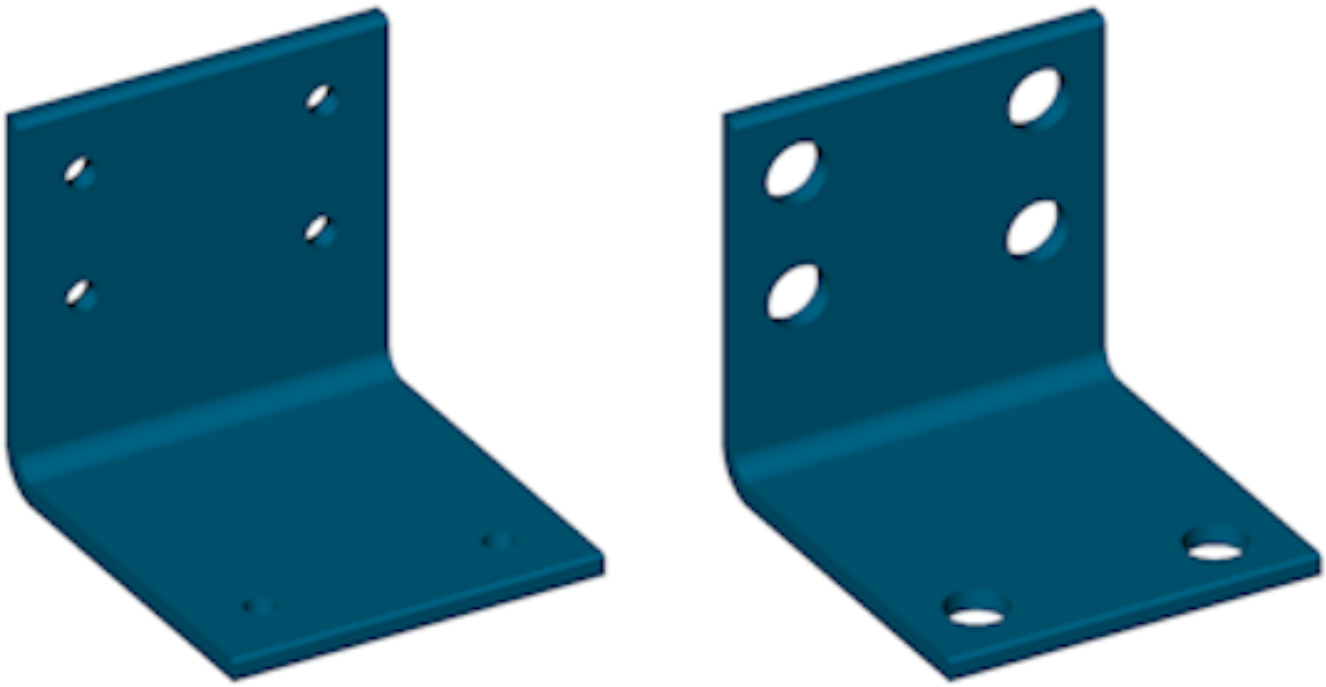


Figure 1.741



1.18.3 Design intent recognition

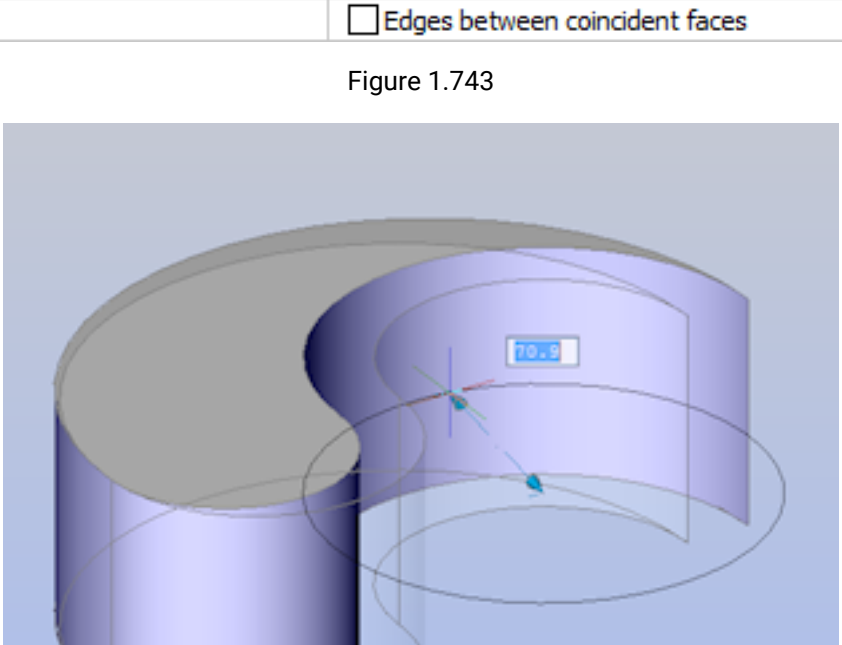
Direct modeling operations allow to change geometry of 3D solid models easily: when a single face of a solid is modified with a particular operation other faces are modified accordingly in order to keep design intent of the model and its topological and geometrical validity. For example, consider a model with a number of holes of the same radius. When the radius of a cylindrical face is modified with a push/pull operation or by changing the value of a radial constraint other faces that share the same diameter are recognized by BricsCAD and modified automatically to keep the radii synchronized. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEnti
Apart from the recognition of equal radii, BricsCAD automatically recognizes other geometrical relations between surfaces of a solid and preserves them during direct modeling operations. To define the design intent recognition, click the buttons on the **Design Intent** toolbar. A pressed button indicates the feature is selected. An unpressed button indicates the feature is not active. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/

- tangent surfaces (planes, cylinders and cones) ( )
- coincident planes ( )
- parallel planes ( )
- perpendicular planes ( )
- cylinders perpendicular to planes ( )
- coaxial surfaces (cylinders and cones) ( )
- surfaces of equal radius (cylinders and spheres) ( )
- vertices between 4 or more faces ( )
- edges between coincident faces ( )
- switch off/on all ( )

Figure 1.742

Automatic 3D geometry constraints recognition	0x0061 (97)
negative	☐ Switch off automatic 3D geometry constraints recognition
1	☑ Tangent surfaces
2	☐ Coincident planes
4	☐ Parallel planes
8	☐ Perpendicular planes
16	☐ Cylinders perpendicular to planes
32	☑ Coaxial surfaces
64	☑ Cylinders and spheres of equal radius
128	☐ Vertices between 4 or more faces
256	☐ Edges between coincident faces

Figure 1.743



The design intent options can also be set by editing the DMRECOGNIZE system variable in the Settings dialog (1.13.11). Click the **Options** tool button ( ) on the **3D Constraints** toolbar to set the **Automatic 3D geometry constraints recognition**. The state of tool buttons on the **Design Intent** toolbar (pressed or unpressed) is adjusted accordingly. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/ART/03

NOTE 3D Constraints (1.19.4) which are explicitly defined by the user are taken into account when solid model is being modified with direct modeling commands. These constraints have a higher priority than automatically recognized ones and allow the user to better control the behavior of solid model modifications.

The following examples show the difference in the behavior whether automatic 3D geometry constraints are recognized or not. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/
tangent surfaces recognized ( ) tangent surfaces not recognized ( ) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/
coincident planes recognized ( ) coincident planes not recognized ( ) (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/
parallel planes recognized ( ) parallel planes not recognized ( ) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/

Figure 1.744

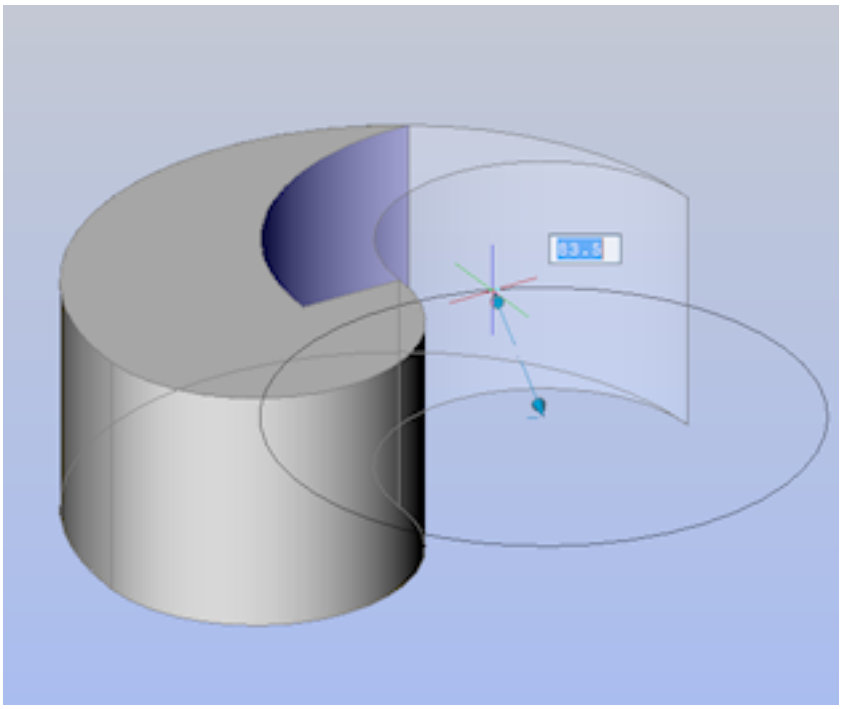


Figure 1.745

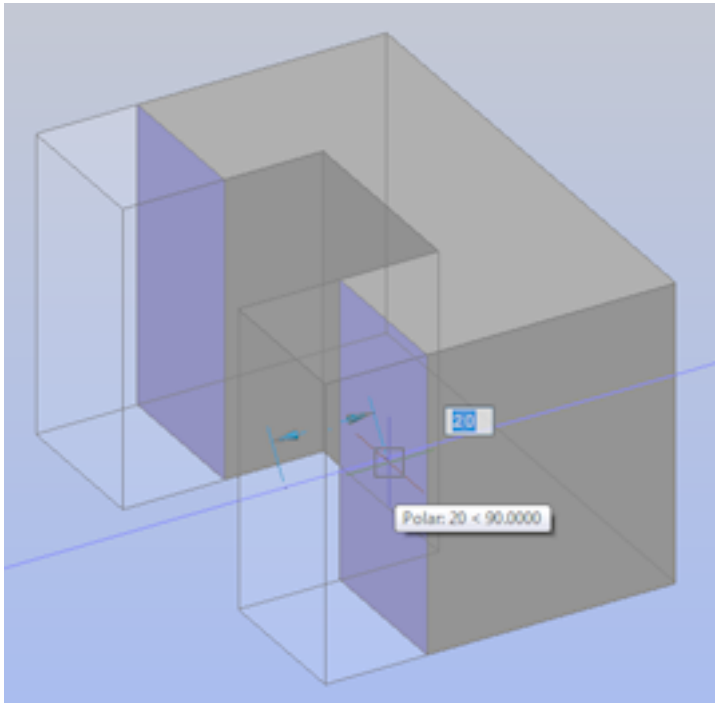


Figure 1.746

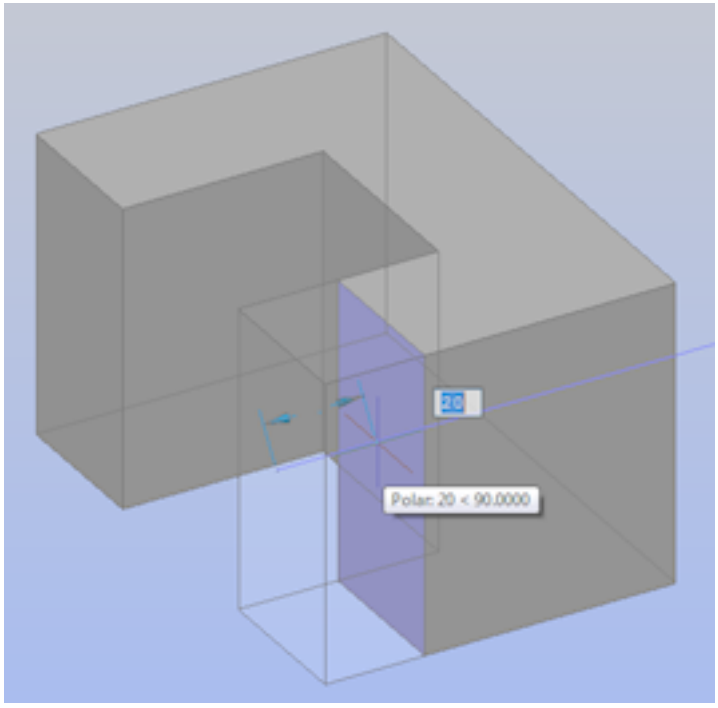


Figure 1.747

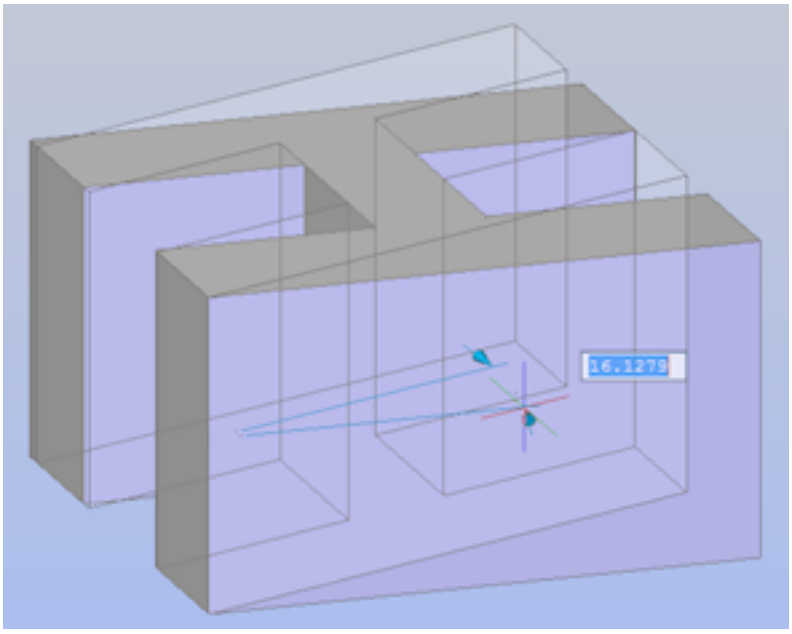


Figure 1.748

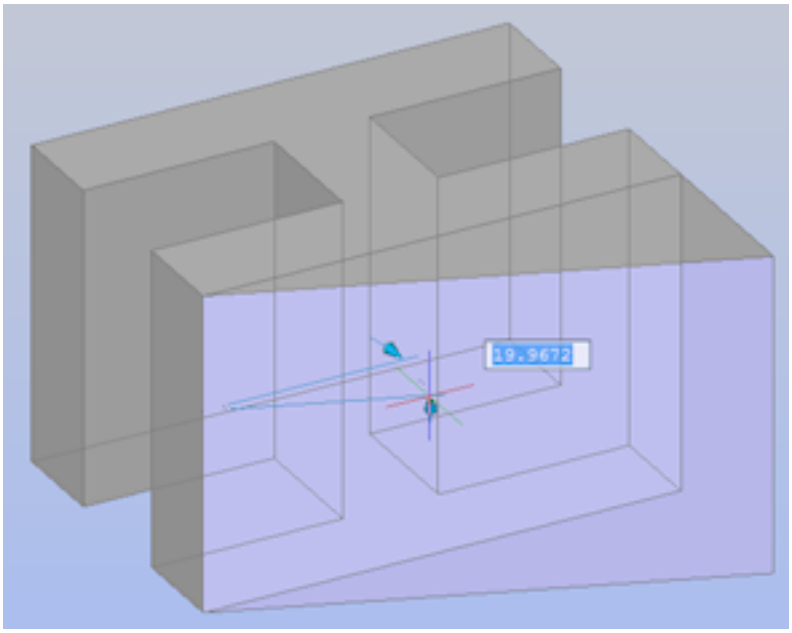


Figure 1.749

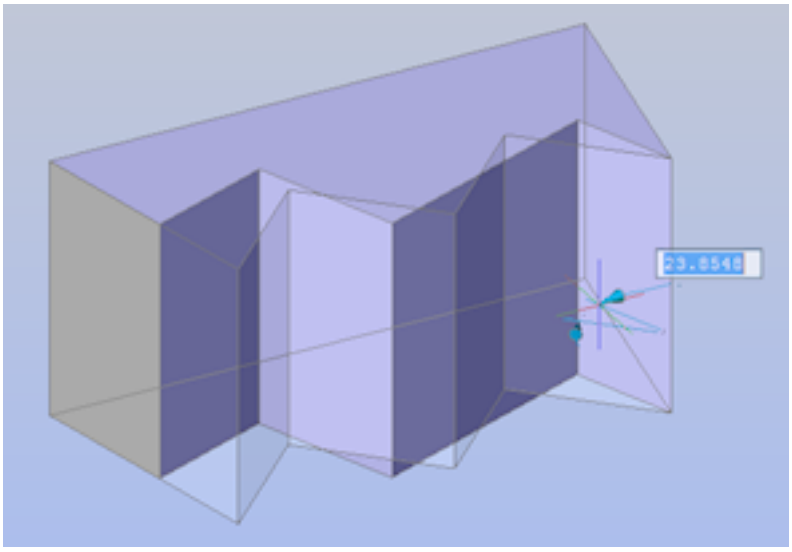


Figure 1.750

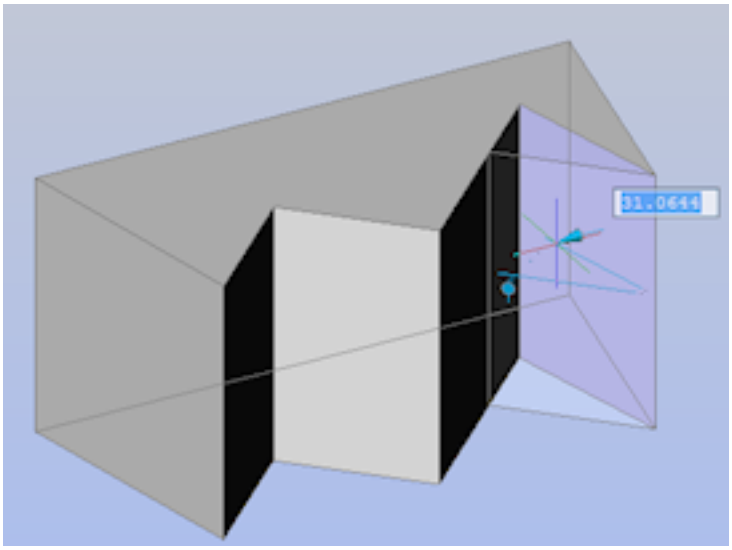


Figure 1.751

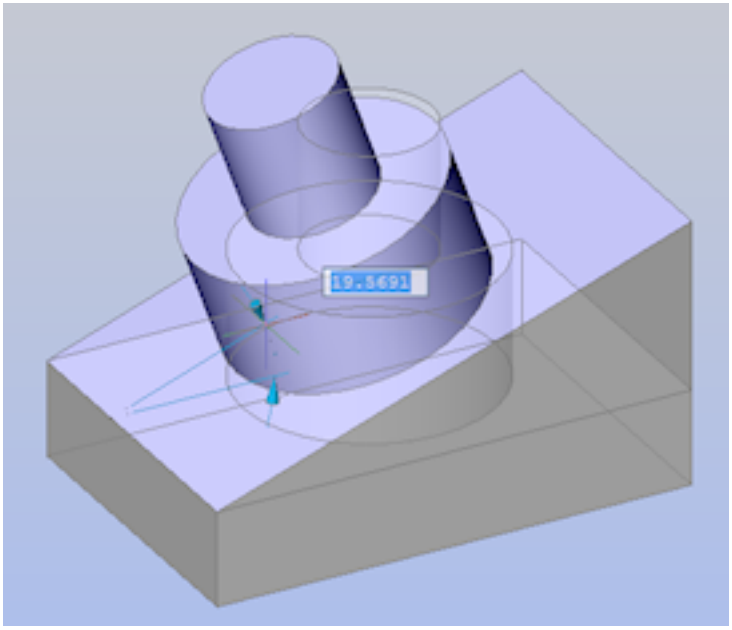


Figure 1.752

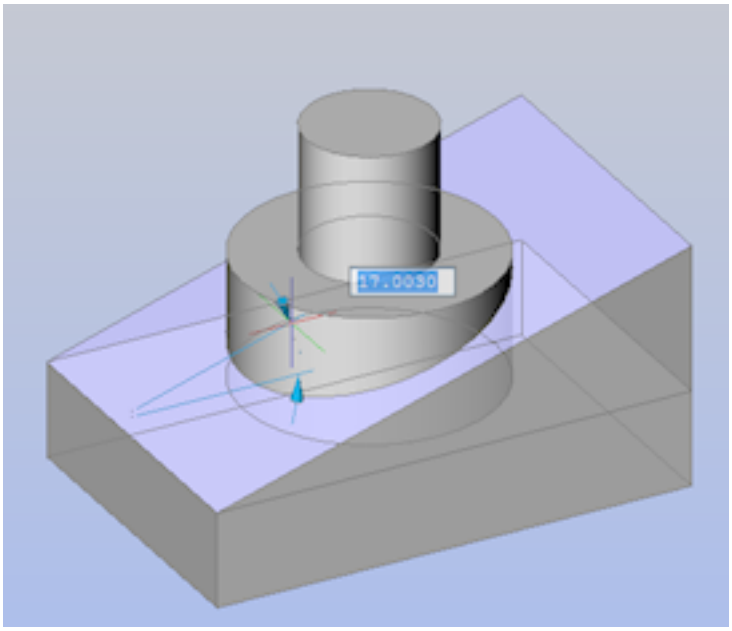


Figure 1.753

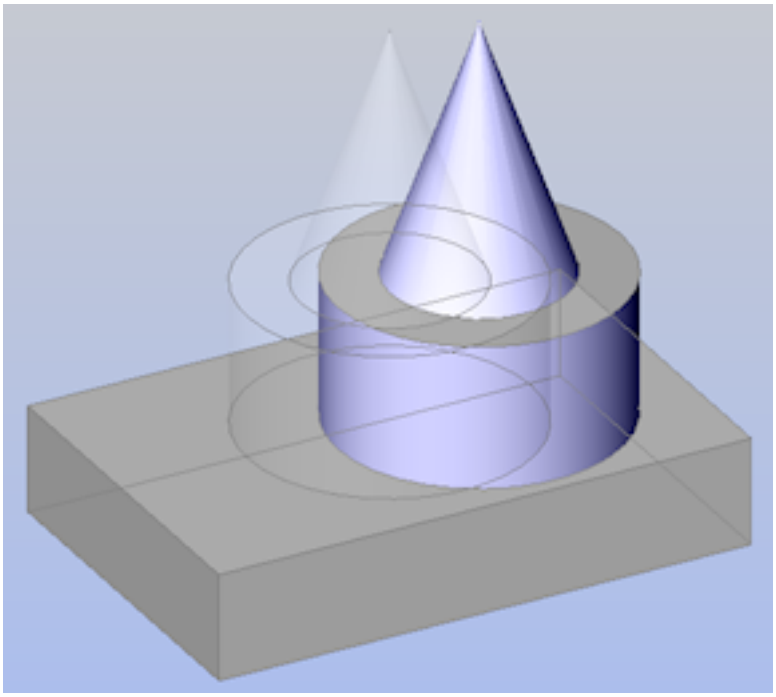


Figure 1.754

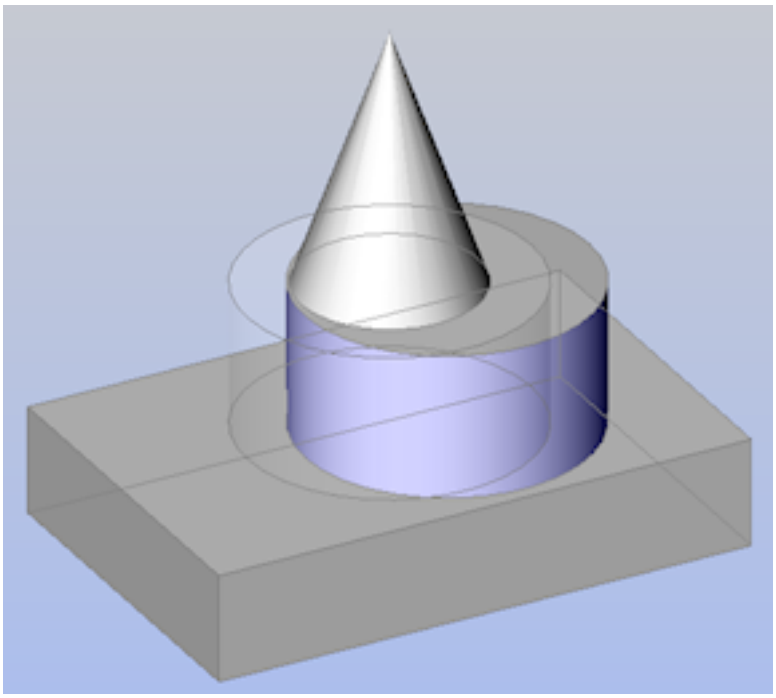


Figure 1.755

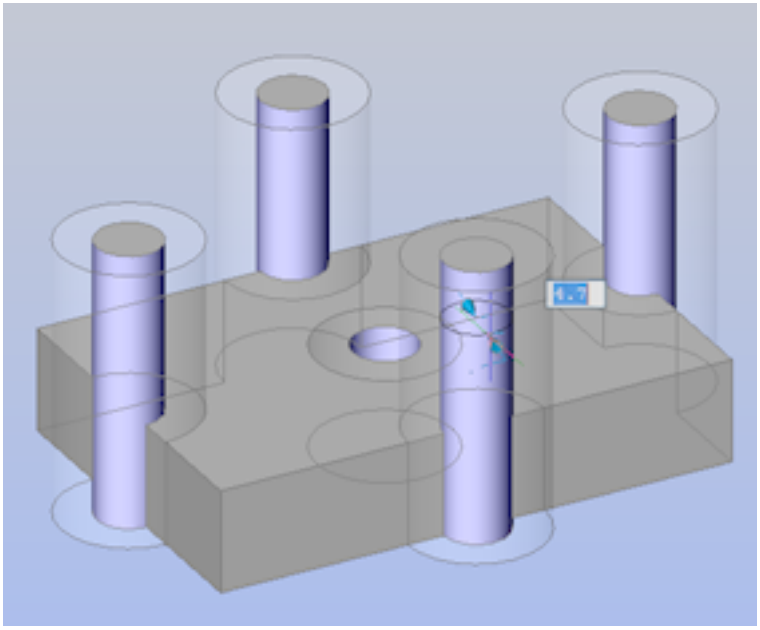
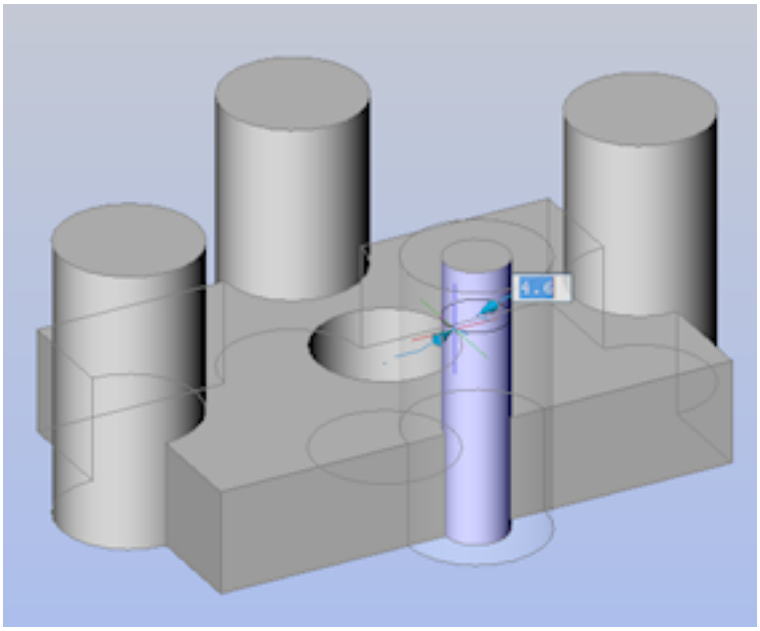


Figure 1.756



1.18.4 Working with sections

Commands: SECTION (6.27.11), SECTIONPLANE (6.27.12), LIVESECTION (6.20.34), SECTIONPLANESETTINGS (6.27.13), SECTIONPLANETOBLOCK (6.27.14). The **Section** command creates section planes of 3D solids; the results are region entities. The **SectionPlane** command creates a section entity that creates sections of 3D solids, surfaces, and meshes. The **LiveSection** command toggles the **Live Section** property of a section plane. The **SectionPlaneSettings** command defines the properties of a section plane entity in the Drawing Explorer (1.14.2) - **Section Planes** dialog. The **SectionPlaneToBlock** command saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

NOTE	The BimSection (6.7.27) command creates **BIM section** entities, which are SectionPlane entities with extended properties, such as type (plan, section, elevation)and scale.

To launch the Sectionplane command

Do one of the following:

- Click the **Section Plane** tool button ( ) on the **Sections** toolbar.
- Choose **Sections > Section Plane**in the **Model** menu.
- Type **sectionplane** in the command bar.

You are prompted: Specify start point or [select Face/Draw/Orthographic]:

Align a section plane to a face

1. LaunchSectionPlane (6.27.12) command.
2. Choose **Select Face** in the prompt menu or type **F** in the command bar. **You are prompted:** Select entity:
3. Select the face of a solid you want to align the section plane to.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e Solid	(\vref {fig:C:/Dev/Task- Select face	(\vref {fig:C:/Dev/Task- Section plane aligned to face

The section plane is aligned to the selected face. The **Clip Display** property of the section entity is switched on automatically.

4. (option) Edit the properties of the section plane.

To create a vertical section plane

1. Make sure the WCS is the current coordinate system.
2. LaunchSectionPlane (6.27.12) command.
3. Specify two points. The section plane is created through these points and perpendicular to the XY-plane of the current coordinate system. The part of the solid at the right hand side of the section plane is cut away. The **Clip Display** property of the section entity is switched on automatically.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help Specify two points	Vertical section plane

4. (option) Edit the properties of the section plane.

To create a horizontal section plane



1. Choose **Front** in the **Look From** widget.
2. Align the UCS (1.15.18) to the front view: Type **ucs** at the command prompt, choose the **View** option in the prompt menu or type **V** in the command bar. If UCSORTH0 (1.13.11) = ON, the UCS (1.15.18) is aligned with the view automatically.
3. Launch SectionPlane (6.27.12) command.
4. Specify two points. The section plane is created through these points and perpendicular to the XY-plane of the current coordinate system. The part of the solid at the right hand side of the section plane is cut away. The **Clip Display** property of the section entity is switched on automatically.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
Specify two points	The part of the solid at the right hand side of the section plane is cut away.	Horizontal section plane

5. (option) Edit the properties of the section plane.

To create a vertical jogged section plane

1. Make sure the WCS is the current coordinate system.
2. Launch SectionPlane (6.27.12) command.
3. Choose **Draw** in the prompt menu or type **D** in the command bar. **You are prompted:** Specify start point:
4. Specify the first point (1). **You are prompted:** Specify next point:
5. Specify the second point (2): **You are prompted:** Specify next point or ENTER to finish:
6. Specify the other points (3 - 6), the right click or press Enter to stop. **You are prompted:** Specify the direction of the section entity:
7. Click a point (7) to specify the direction of the section entity. The part of the solid opposite to the direction of the section entity is cut away. The **Clip Display** property of the section entity is switched on automatically.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
Specify points	The part of the solid opposite to the direction of the section entity is cut away.	Vertical jogged section plane

8. (option) Edit the properties of the section plane.

To create a horizontal jogged section plane



1. Choose **Front** in the **Look From** toolbar.

2. Align the UCS (1.15.18) to the front view: Type **ucs** at the command prompt, choose the **View** option in the prompt menu or type **V** in the command bar. If UCSORTH0 (1.13.11) = ON, the UCS (1.15.18) is aligned with the view automatically.
3. Launch SectionPlane (6.27.12)command.
4. Choose **Draw** in the prompt menu or type **D** in the command bar. **You are prompted:** Specify start point:
5. Specify the first point (1). **You are prompted:** Specify next point:
6. Specify the second point (2). **You are prompted:** Specify next point or ENTER to finish:
7. Specify the other points (3 - 6) , the right click or press Enter to stop. **You are prompted:** Specify the direction of the section entity:
8. Click a point (7) to specify the direction of the section entity. The part of the solid opposite to the direction of the section entity is cut away. The **Clip Display** property of the section entity is switched on automatically. The section plane gets the **Boundary** state.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
Specify points	The part of the solid opposite to the direction of the section entity is cut away.	Horizontal jogged section plane

9. (option) Edit the properties of the section plane.

To create an orthographic section plane

1. Make sure the WCS is the current coordinate system.
2. Launch SectionPlane (6.27.12) command.
3. Choose **Orthographic** in the prompt menu or type **O** in the command bar. **You are prompted:** Align section to [Front/bAck/Top/Bottom/Left/Right] <Top>:
4. Pick the orthographic section orientation of your choice in the prompt menu or type the corresponding option in the command bar. Orthographic section planes are created through the center of the solid, parallel to the XY-plane (Top and Bottom), YZ-plane (Left and Right) or XZ-plane (Front and Back) of the current coordinate system. The **Clip Display** property of the section entity is switched on automatically.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
Top	Front	Right

5. (option) Edit the properties of the section plane.

To open the Drawing Explorer - Section Planes dialog

Do one of the following:

- Choose **Drawing Explorer > Section Planes** in the **Tools** menu.
- Choose **Sections > Section Plane Settings...** in the **Model** menu.
- Type **sectionplanesettings** in the command bar, then press Enter.

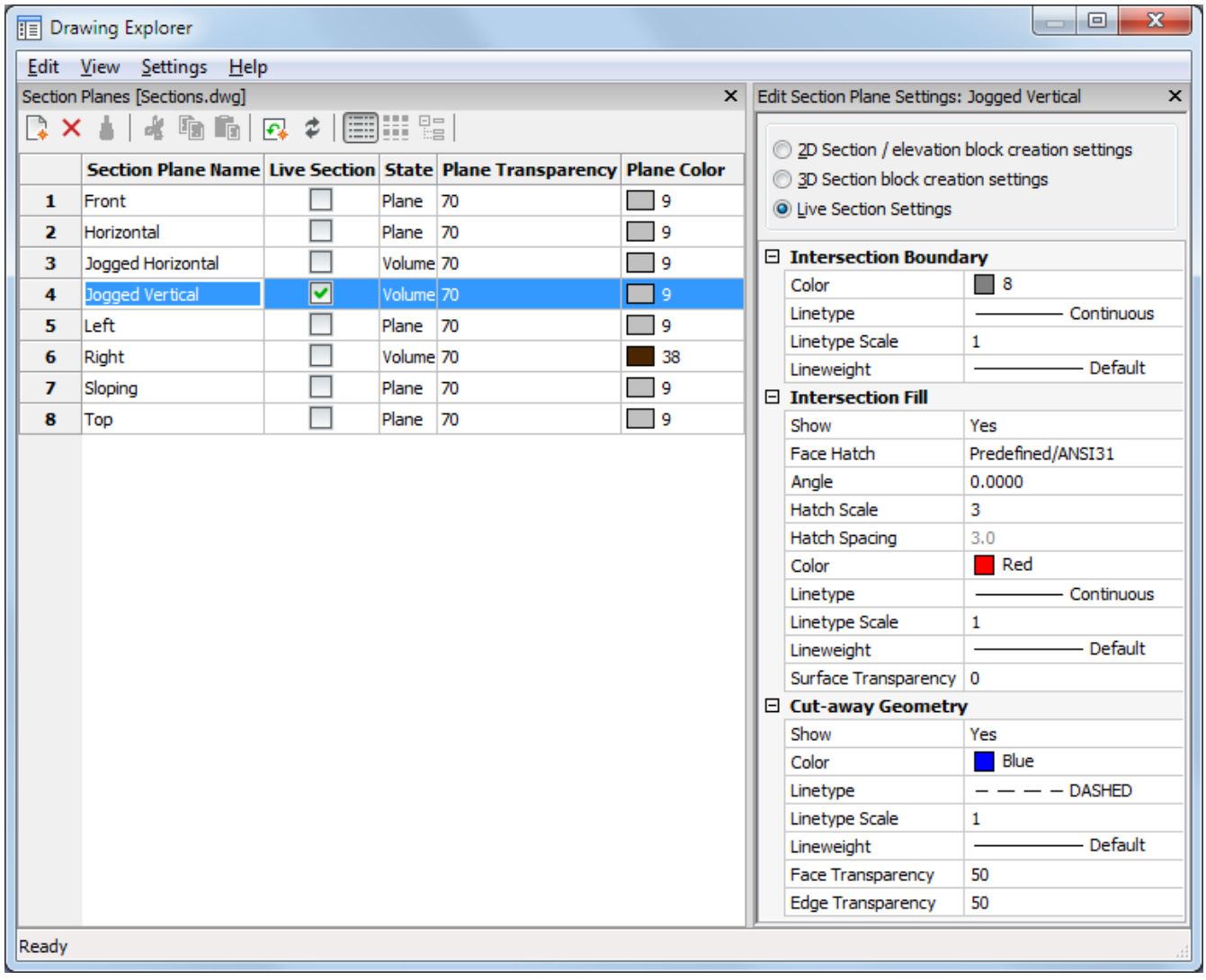
Editing the properties of a section plane

Double click the section entity in the model to toggle its **Clip Display** property.

To edit the properties of a section plane in the Drawing Explorer

1. Open the Drawing Explorer (1.14.2) - **Section Planes** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

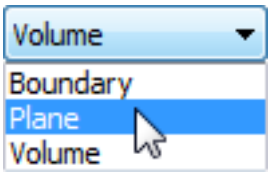
Figure 1.757



	Section Plane Name	Live Section	State	Plane Transparency	Plane Color
1	Front	☐	Plane	70	9
2	Horizontal	☐	Plane	70	9
3	Jogged Horizontal	☐	Volume	70	9
4	Jogged Vertical	☑	Volume	70	9
5	Left	☐	Plane	70	9
6	Right	☐	Volume	70	38
7	Sloping	☐	Plane	70	9
8	Top	☐	Plane	70	9

Intersection Boundary	
Color	8
Linetype	Continuous
Linetype Scale	1
Lineweight	Default
Intersection Fill	
Show	Yes
Face Hatch	Predefined/ANSI31
Angle	0.0000
Hatch Scale	3
Hatch Spacing	3.0
Color	Red
Linetype	Continuous
Linetype Scale	1
Lineweight	Default
Surface Transparency	0
Cut-away Geometry	
Show	Yes
Color	Blue
Linetype	DASHED
Linetype Scale	1
Lineweight	Default
Face Transparency	50
Edge Transparency	50

2. (option) Click the **Section Plane Name** field, then click again to type a new name.
3. (option) Click the **Live Section** check box of a section plane to toggle the **Live Section** property of the section plane.
4. If necessary, click the **Regen** tool button () in the **Details** toolbar to update the display.
5. (option) Click the **State** field of a section plane to choose a different state for the section plane.



6. (option) click the **Plane Transparency** field of a section plane to change the transparency of the section plane entity in the drawing. 0 = Opaque, 100 = Transparent (invisible).
7. (option) Click the **Plane Color** field, then choose a new color in the Select Color (1.17.4) dialog.

To edit the properties of a section plane in the properties bar: Select the section entity in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/S

- **Name**: allows to rename the section entity.
- **State**: click, then select **Plane**, **Boundary** or **Volume**.
- **Live Section**: click, then select **Yes** or **No**.
- **Clip display**: click, then select **Yes** or **No**.
- **Use lids**: click, then select **Yes** or **No**.
- **Plane transparency**: allows to define the transparency of the section plane.
- **Plane color**: allows to define the color of the section plane.

Figure 1.758

Section Entity	
Name	Plan
State	Plane
Live section	No
Clip display	No
Use lids	No
Plane transparency	70
Plane color	9

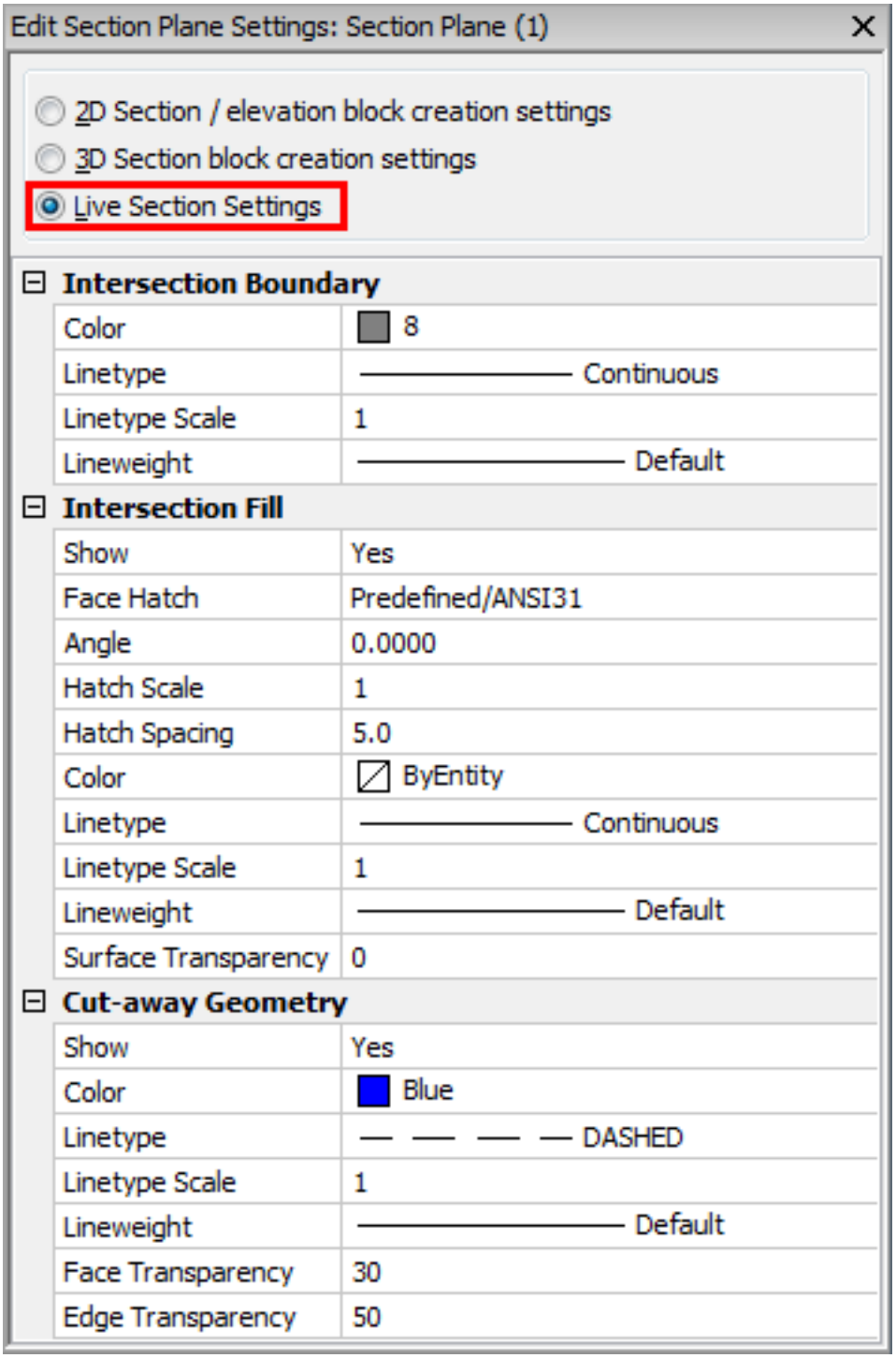
Difference between Live Section and Clip Display

In both statuses all drawing content is sectioned and clipped-away parts are no longer displayed. **Live** status: entities in a drawing are temporarily modified, e.g. clipped solids are replaced by temporary solids with a different shape. **Clip Display** status: the display of entities is clipped. As in the AutoCAD **3DCLIP** command, entities are not replaced by temporary versions. Different from the **Live** status, the **Clip Display** status allows to continue using all modeling and editing operations on all drawing content. While the **3DCLIP** command allows to clip the display by a **Front** and a **Back** plane, it is not very useful because these planes must be reset whenever the view direction changes. Using a Section entity to control which part of a model is displayed, makes it easy to store and repeat display states. Besides the **Plane** state, which typically clips a model in two halves, also the **Boundary** and **Volume** state of section entity can be used. As opposed to the **Live** status, the Clip Display status can be set for multiple section entities simultaneously. Clipped solids can optionally be closed by setting the **Use Lids** option. Although it has been made possible to snap to the geometry of these lids, they are not actually added to the drawing. If enabled, the hatch pattern defined in the **2dSection Intersection Fill** settings of the section entity displays on top of the lids. To set the **Live Section** property of a section plane:

- Use the **LiveSection** command.
- Set the**Live Section** field in theProperties bar
- Use the **Drawing Explorer - Section Planes** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
Cut-away Geometry ON	Cut-away Geometry OFF

Figure 1.759


Edit Section Plane Settings: Section Plane (1)
2D Section / elevation block creation settings
3D Section block creation settings
Live Section Settings
Intersection Boundary
Color 8
Linetype Continuous
Linetype Scale 1
Lineweight Default
Intersection Fill
Show Yes
Face Hatch Predefined/ANSI31
Angle 0.0000
Hatch Scale 1
Hatch Spacing 5.0
Color ByEntity
Linetype Continuous
Linetype Scale 1
Lineweight Default
Surface Transparency 0
Cut-away Geometry
Show Yes
Color Blue
Linetype DASHED
Linetype Scale 1
Lineweight Default
Face Transparency 30
Edge Transparency 50

To modify the size and position of a section plane

1. Select the section plane entity in the drawing. Depending on the current state of the section plane entity a number of grips display.
2. (option) Switch on the **Clip Display** property of the section plane to see the result of the modifications dynamically.
3. (option) To grip edit a **Plane** state section plane entity: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS
   Four grips display.
   - 1 ( ) = origin point: moves the position of the section plane in the XY-plane.
   - 2 ( ) = through point: redefines the through point of the section plane. The section plane is rotated around the origin point (1).
   - 3 ( ) = midpoint: moves the section plane parallel in a direction perpendicular to the section plane.
   - 4 ( ): click to flip the view direction of the section plane.
4. (option) To grip edit a **Boundary** state section plane entity: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
   Ten grips display.
   - 1 ( ) = origin point: moves the position of the section plane in the XY-plane.
   - 2 ( ) = through point: redefines the through point of the section plane. The section plane boundary is stretched and rotated around the origin point (1).
   - 3 ( ): stretches the section plane boundary parallel.
   - 4 ( ): modifies the shape of the section plane boundary.
   - 5 ( ): click to flip the view direction of the section plane.
5. (option) To grip edit a **Volume** state section plane entity: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
   Twelve grips display.
   - 1 ( ) = origin point: moves the position of the section plane in the XY-plane.
   - 2 ( ) = through point: redefines the through point of the section plane. The section plane volume is stretched and rotated around the origin point (1).
   - 3 ( ) :stretches the section plane volume parallel.
   - 4 ( ): modifies the shape of the section plane volume.
   - 5 ( ): modifies the height of the section plane volume.
   - Polar Tracking (1.15.16) must be on to move grips 5. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/er
   - 6 ( ): click to flip the view direction of the section plane.

Figure 1.760

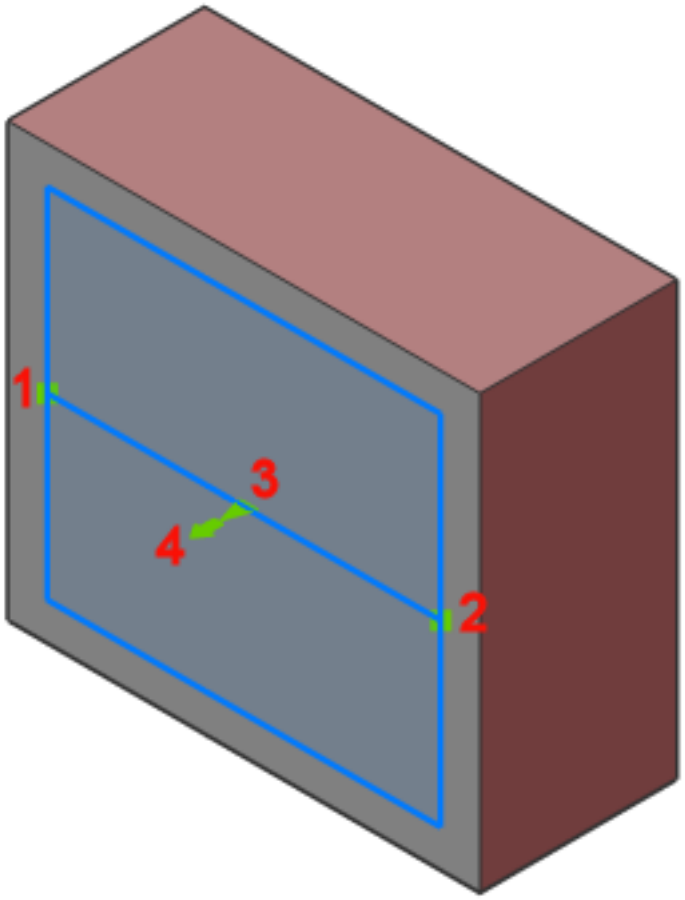


Figure 1.761

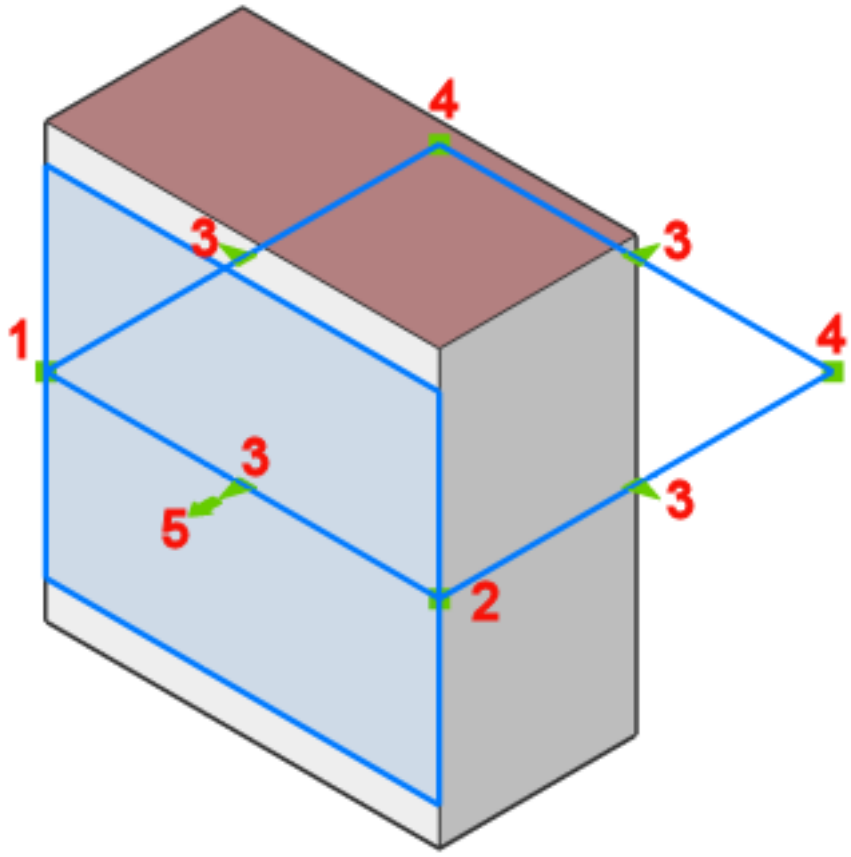


Figure 1.762

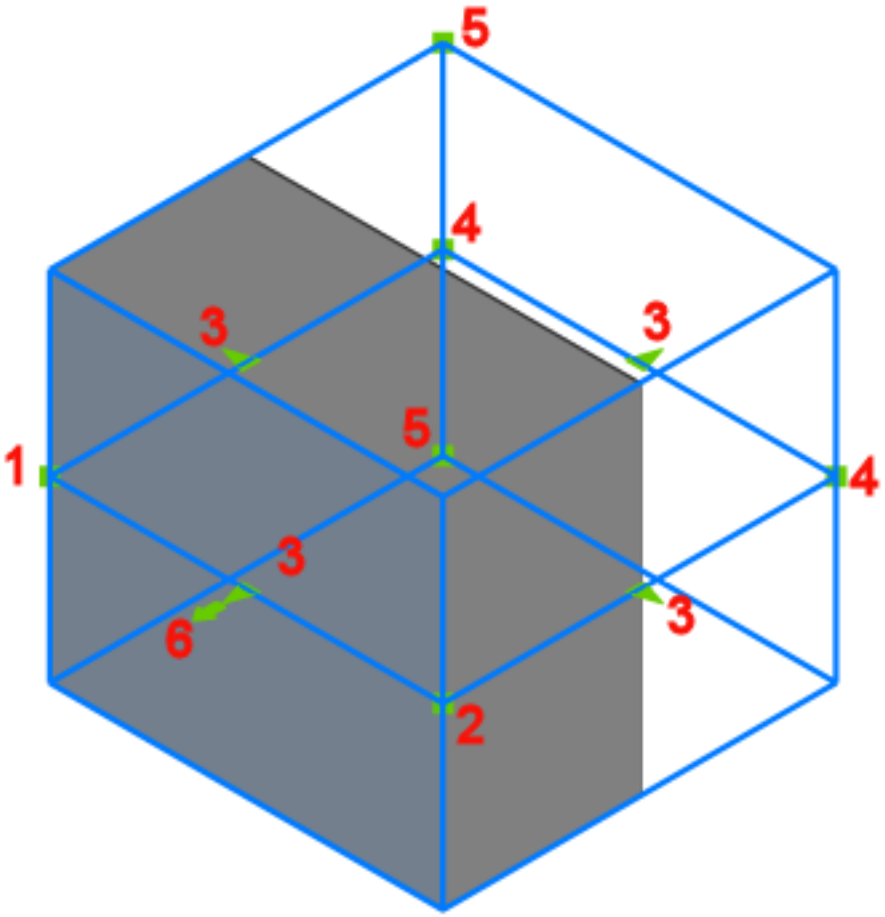
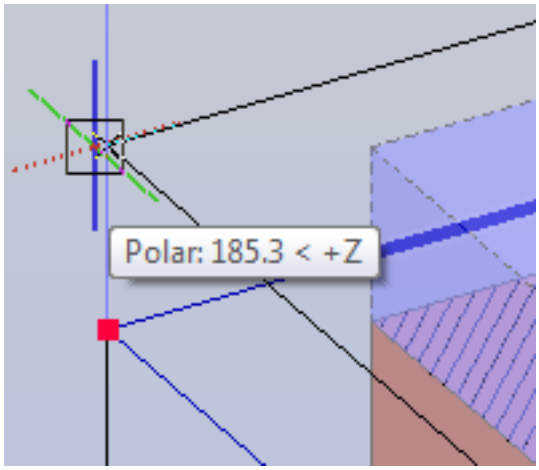


Figure 1.763



To save a section

SectionPlaneToBlock (6.27.14) command allows you to:

- Insert a section in the drawing as a 2D (1.18.4) section / elevation block or a 3D cutaway (1.18.4) block.
- Replace a section block (1.18.4) (e.g. when the section entity has been modified).
- Save (1.18.4) a section to a new drawing.

To launch SectionPlaneToBlock command

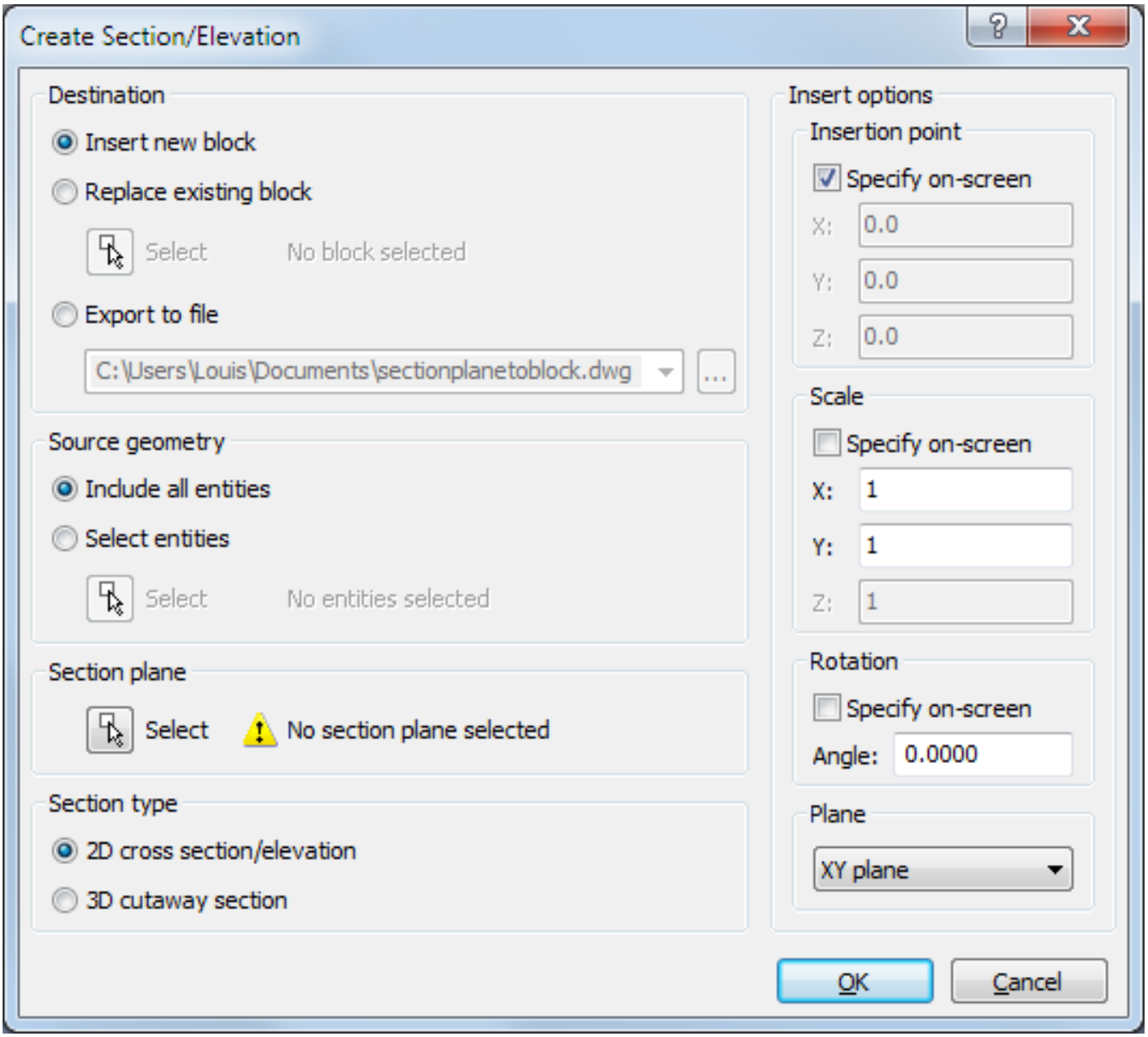
Do one of the following:

- Click the **Section Plane to Block** tool button ( ) on the **Sections** toolbar.
- Choose **Section Plane to Block** in the **Model > Sections** menu.
- Type **sectionplanetoblock** in the command bar, then press Enter

The **Create Section/Elevation** dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Usr
To insert a section as a 2D block

1. (option) Adjust the 2D Section / Elevation (6.27.13) settings of the section plane entity.
2. Launch SectionPlaneToBlock (6.27.14) command.
3. Set the **Destination** to **Insert new block**.
4. Under **Source Geometry**, do one of the following:
   - Choose **Include all entities** to include all 3D solids in the drawing.
   - Choose **Select entities**, then click the **Select** button ( ) to select 3D solids.
5. Under **Section plane**, click the **Select** button ( ), then select a section plane entity in the drawing.
6. Set the **Section type** to **2D cross section/elevation**.
7. Adjust the **Insert** options (see Insert a block (1.22.6)).
8. Click the **OK** button. The **Create Section/Elevation** dialog closes. The block is attached to the cursor. Follow the instructions in the command bar to place the block in the drawing.

Figure 1.764


Create Section/Elevation
Destination
Insert new block
Replace existing block
Select
No block selected
Export to file
C:\Users\Louis\Documents\sectionplanetoblock.dwg
Source geometry
Include all entities
Select entities
Select
No entities selected
Section plane
Select
No section plane selected
Section type
2D cross section/elevation
3D cutaway section
Insert options
Insertion point
Specify on-screen
X: 0.0
Y: 0.0
Z: 0.0
Scale
Specify on-screen
X: 1
Y: 1
Z: 1
Rotation
Specify on-screen
Angle: 0.0000
Plane
XY plane
OK
Cancel

To insert a section as a 3D block

1. (option) Adjust the 3D cutaway section (6.27.13) settings of the section plane entity.
2. Launch SectionPlaneToBlock (6.27.14) command.
3. Set the **Destination** to **Insert new block**.
4. Under **Source Geometry**, do one of the following:
   - Choose **Include all entities** to include all 3D solids in the drawing.
   - Choose **Select entities**, then click the **Select** button ( ) to select 3D solids.
5. Under **Section plane**, click the **Select** button ( ), then select a section plane entity in the drawing.
6. Set the **Section type** to **3D cutaway section**.
7. Adjust the **Insert** options (see Insert a block (1.22.6)).
8. Click the **OK** button. The **Create Section/Elevation** dialog closes. The block is attached to the cursor. Follow the instructions in the command bar to place the block in the drawing.

To replace an existing block

1. Launch SectionPlaneToBlock (6.27.14) command.
2. Set the **Destination** to **Replace existing block**.
3. Click the **Select** button ( ), then select the block to be replaced in the drawing.
4. Under **Source Geometry**, do one of the following:
   - Choose **Include all entities** to include all 3D solids in the drawing.
   - Choose **Select entities**, then click the **Select** button ( ) to select 3D solids.
5. Under **Section plane**, click the **Select** button ( ), then select a section plane entity in the drawing.
6. Under **Section type**, do one of the following:
   - Choose **2D cross section/elevation** , to replace the block by a 2D section / elevation block.
   - Choose **3D cutaway section,** to replace the block by a 3D cutaway section block.
7. Click the **OK** button. The selected block is replaced.

To export a section to a file

1. Launch SectionPlaneToBlock (6.27.14) command.
2. Set the **Destination** to **Export to file**.
3. Click the **Browse** button ( ), then do the following in the **Select export file** dialog box:
   - Select a folder.
   - Type a name in the **File name** field.
   - Click the **Save** button.
4. Under **Source Geometry**, do one of the following:
   - Choose **Include all entities** to include all 3D solids in the drawing.
   - Choose **Select entities**, then click the **Select** button ( ) to select 3D solids.
5. Under **Section plane**, click the **Select** button ( ), then select a section plane entity in the drawing.
6. Under **Section type**, do one of the following:
   - Choose **2D cross section/elevation** , to replace the block by a 2D section / elevation block.
   - Choose **3D cutaway section** . to replace the block by a 3D cutaway section block.
7. Click the **OK** button. A section is saved to a drawing.

Figure 1.765

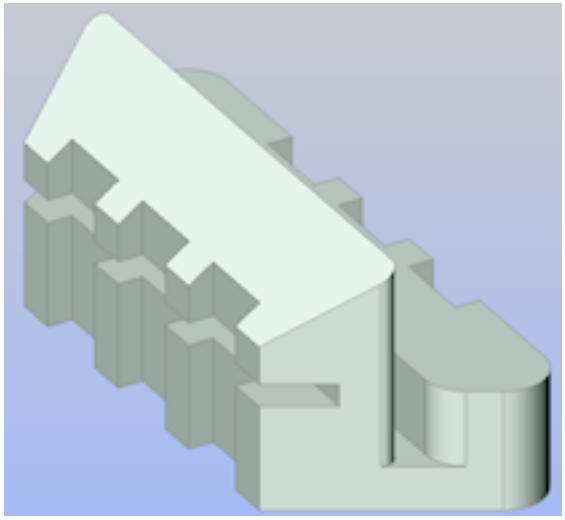


Figure 1.766

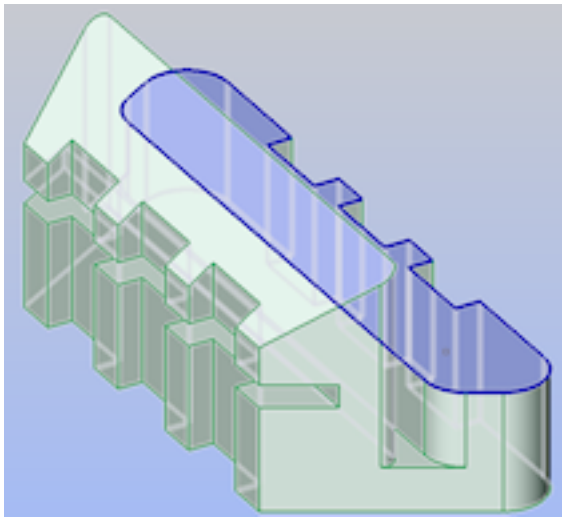


Figure 1.767

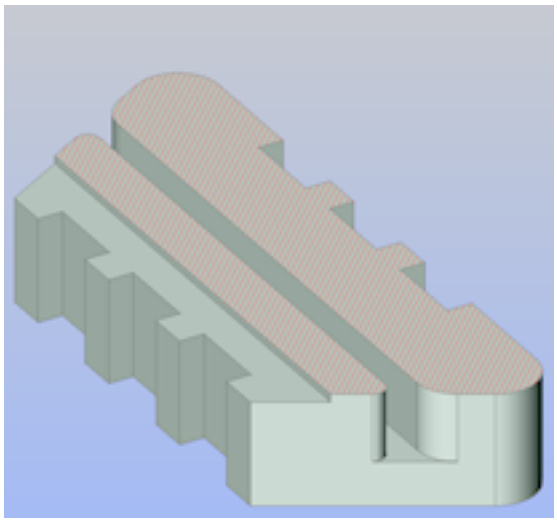


Figure 1.768

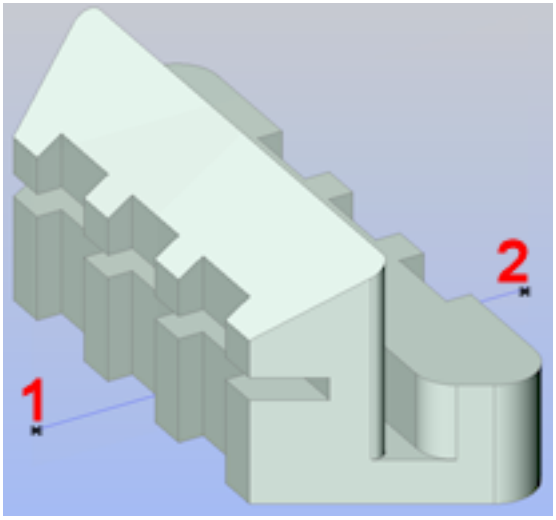


Figure 1.769

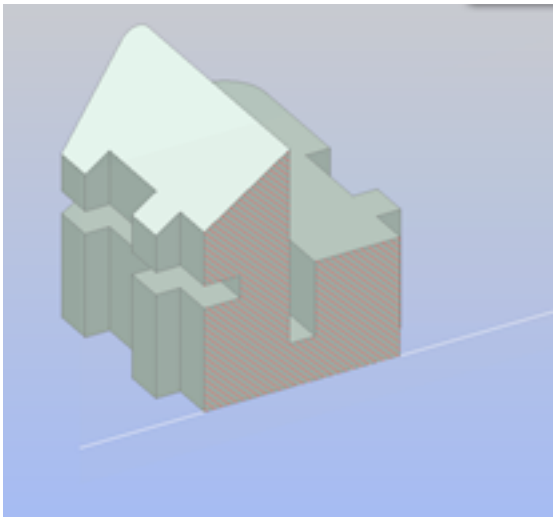


Figure 1.770

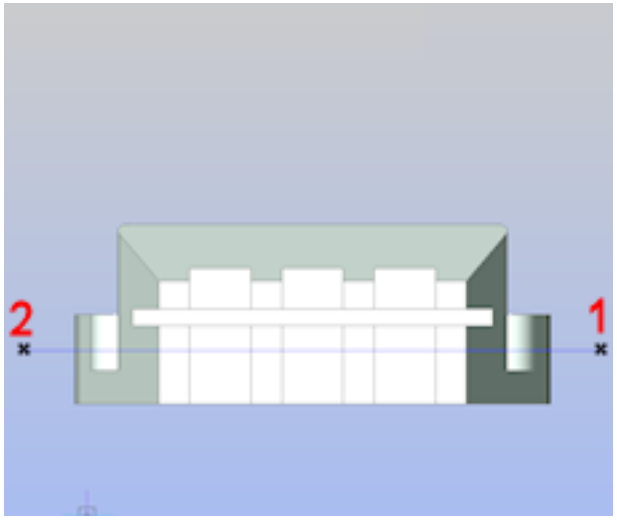


Figure 1.771

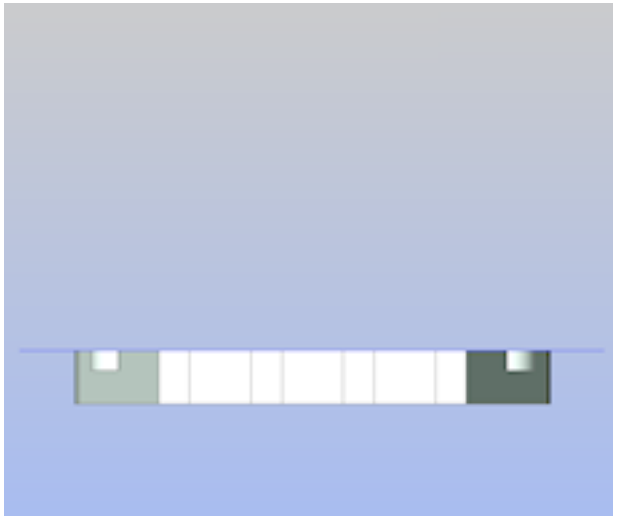


Figure 1.772

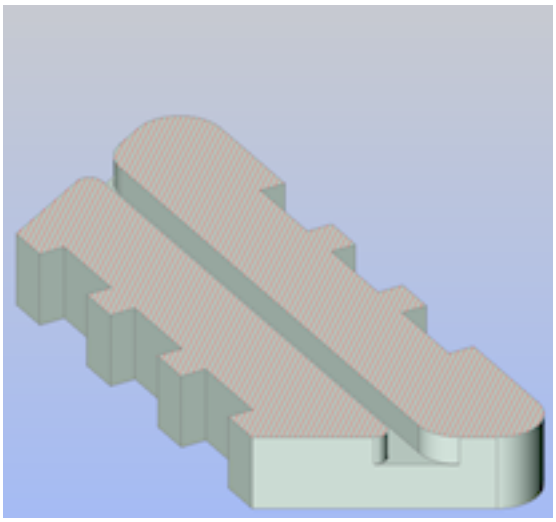


Figure 1.773

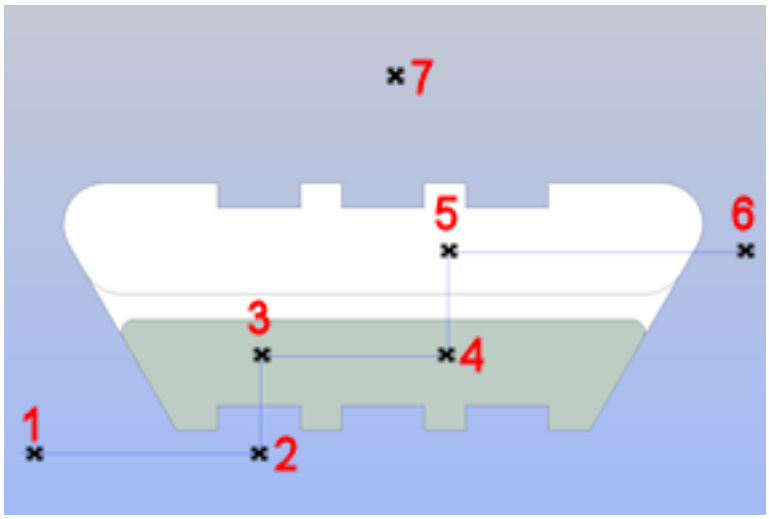


Figure 1.774

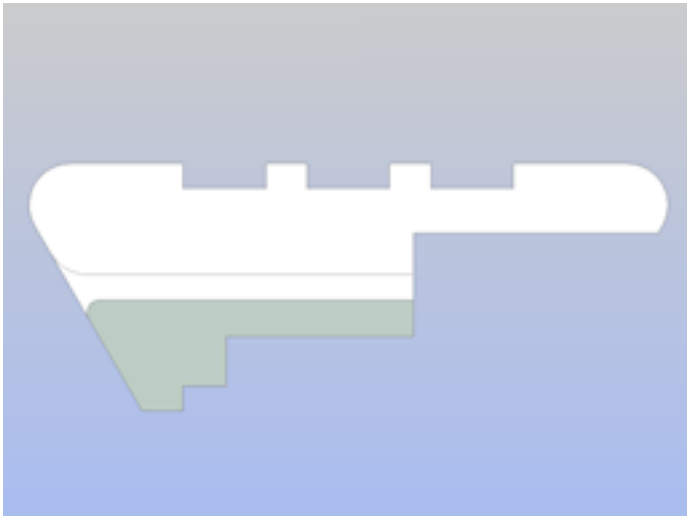


Figure 1.775

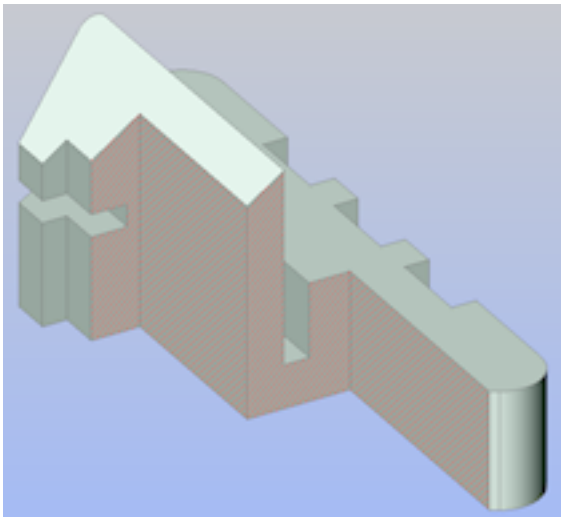


Figure 1.776

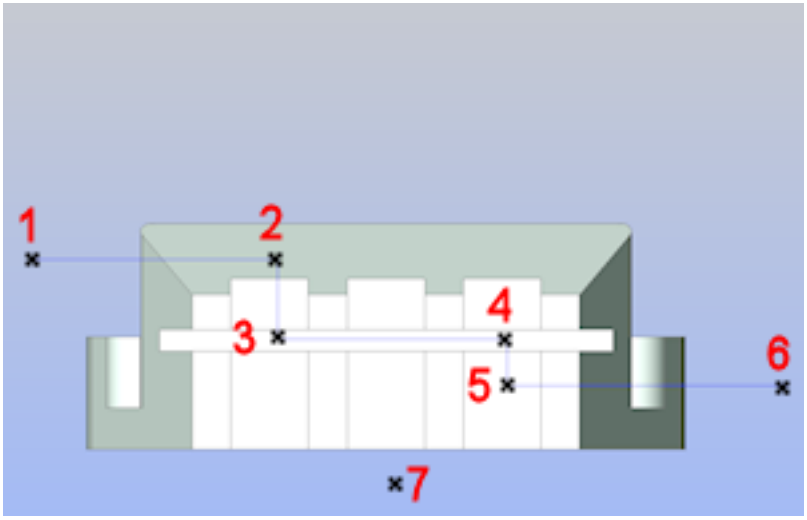


Figure 1.777



Figure 1.778

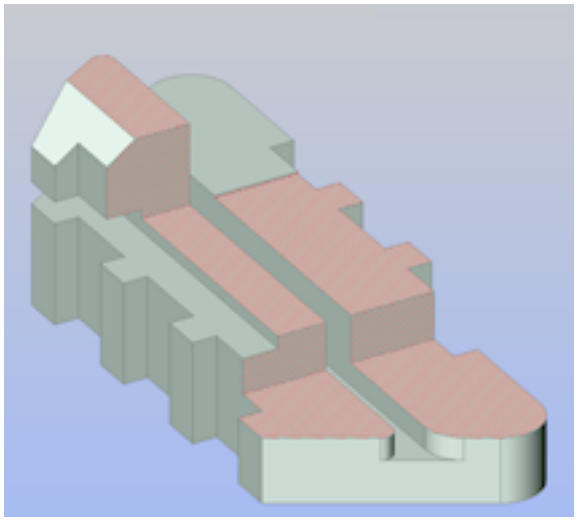


Figure 1.779

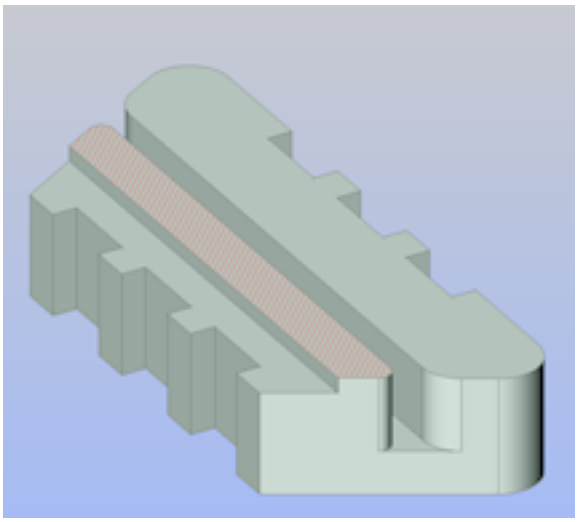


Figure 1.780

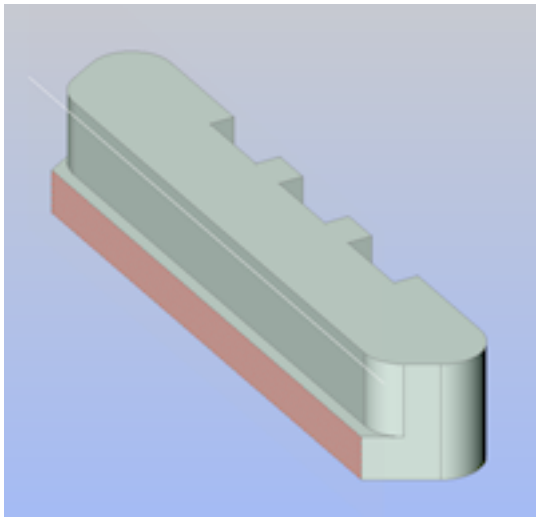


Figure 1.781

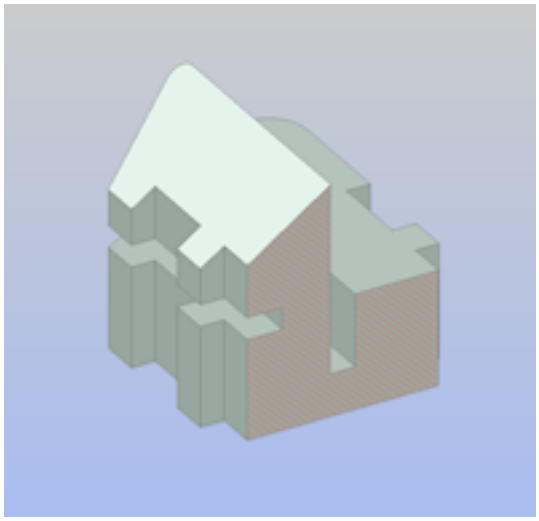


Figure 1.782

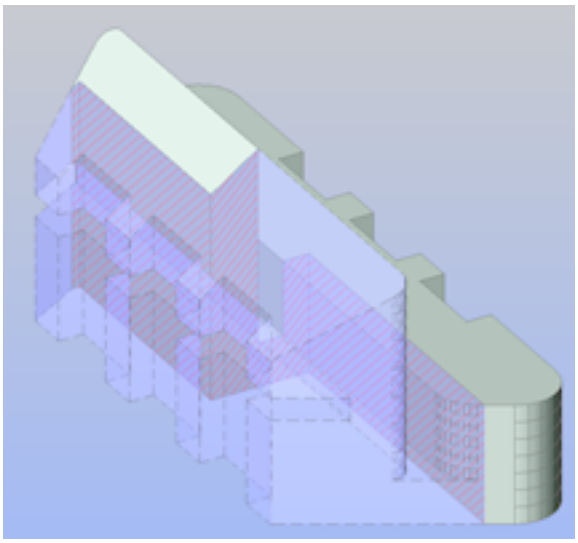


Figure 1.783

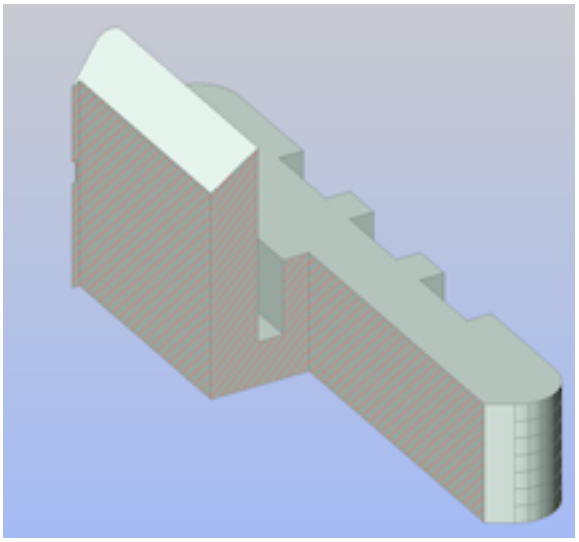
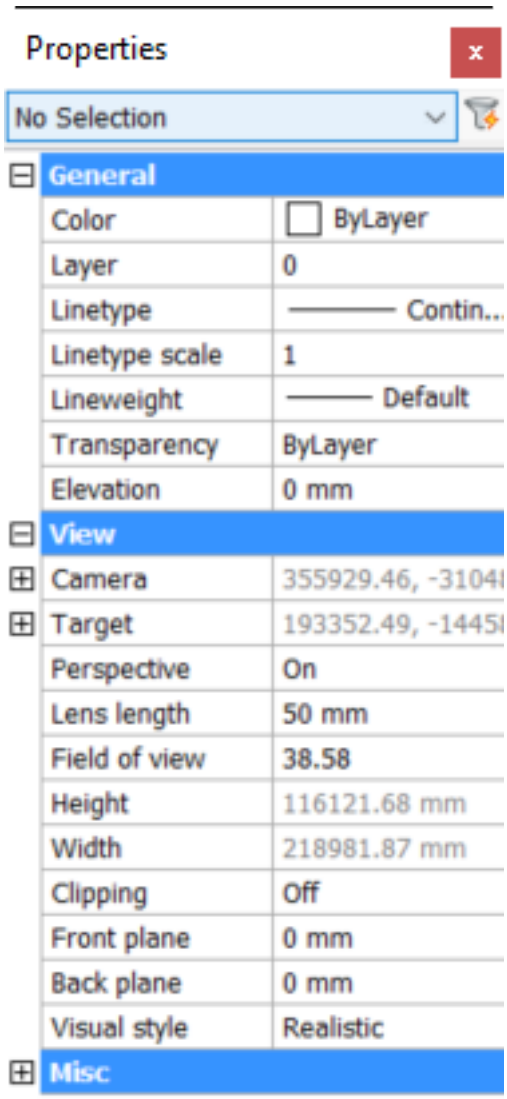


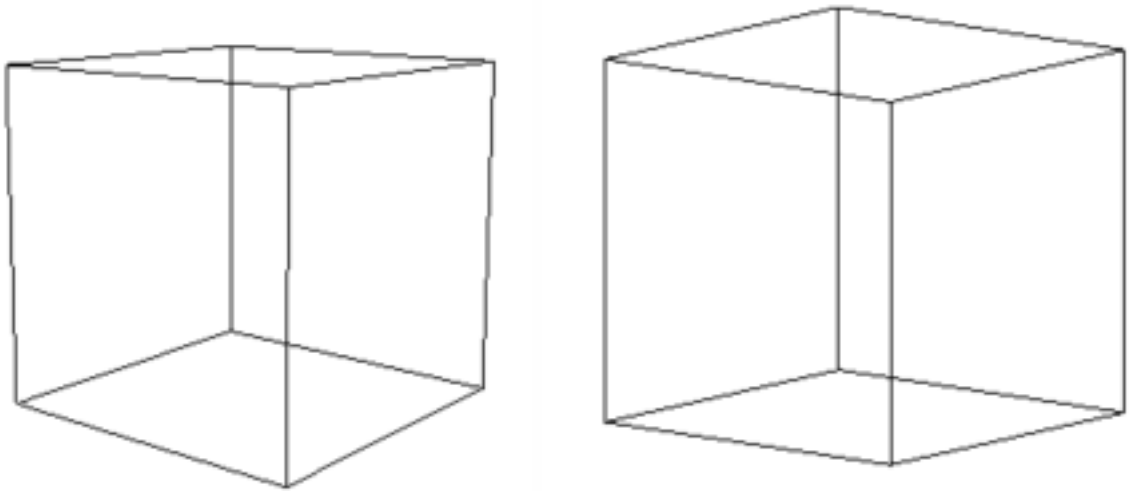
Figure 1.784



1.18.5 Navigate in 3D

As you navigate around and through your 3D model, the location of the model in space remain constant. It's your current view (viewpoint), of the model that's changing. You can see the current view properties in the Properties panel when no objects are selected. The Camera (X,Y,Z) represents your location as you view the model. Pretend you're viewing it through a camera lens. The Target (X,Y,Z) is where the camera lens is pointed. As you navigate around the model, the Camera and Target coordinates update accordingly. Those coordinates are read-only in the Properties Panel. There are other View properties that you can edit. Perspective is the most important controls to be aware of for navigation purposes. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/HHBrics 3D-Navigation-03.png}) You can set Perspective to On or Off for the current view. When set to On, the view displays in visual perspective mode (left). When set to Off, it displays in parallel perspective mode (right). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/HHBricsCAD-3D-Navigation-03-1.png})

Figure 1.785



Mouse and Keyboard Navigation Options

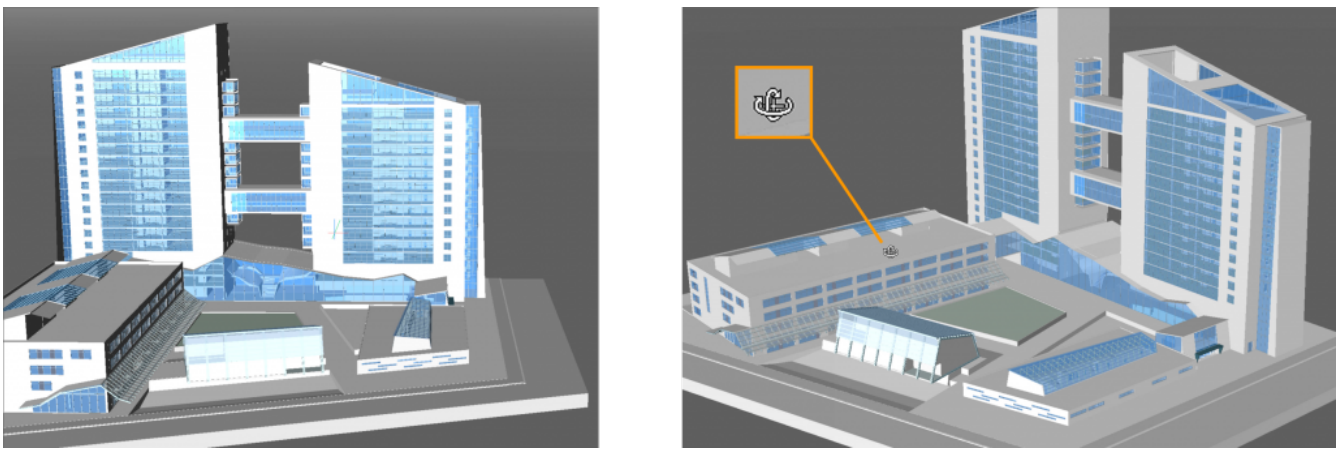
One of the easiest and most common methods for navigating around a 3D model is with the mouse and keyboard. Simply press the Shift key and middle mouse button as you move the mouse to orbit the model. The cursor changes to an orbit icon indicating you're in orbit mode. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3D-Navigation-02-1024x338.png}) There are many other mouse and keyboard navigation methods that you're probably familiar with from your 2D CAD experience. Most of those methods are also useful for 3D modeling. Below are a few of my favorites.

- Zoom in/out – Roll the mouse wheel. You can zoom all the way into a model and walk around inside it when in perspective mode (1.16.11). In isometric mode you can only zoom in very close, but not go inside.
- Zoom Extents – Double-click on the mouse wheel.
- Pan – Press the mouse wheel or middle mouse button.

Refer to the Help system for a comprehensive list of view manipulation controls using the mouse and keyboard (1.16.7).

Tutorial: How to setup the 3DConnexion 3D mouse in BricsCAD Click here (1.6.34) to watch

Figure 1.786



LookFrom Navigation Tool

The LookFrom tool is displayed in the upper right corner of the Drawing area. It's comparable to the ViewCube in AutoCAD. Click on different places on the LookFrom tool to display the view from standard viewpoints. By default, it includes Top, Left, Right, Front, Back, Top Back Left, Top Back Right, Top Front Left, and Top Front Right. As you pass the cursor over a viewpoint, the relevant icon appears at the center and a tooltip displays below. You can press the Ctrl key to access similar options for Bottom viewpoints. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
3D-Navigation-04.png}) A right-click menu offers access to additional controls and settings. To learn more refer to BricsCAD Online Help (1.16.8). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/HI
3D-Navigation-06.png})

Figure 1.787

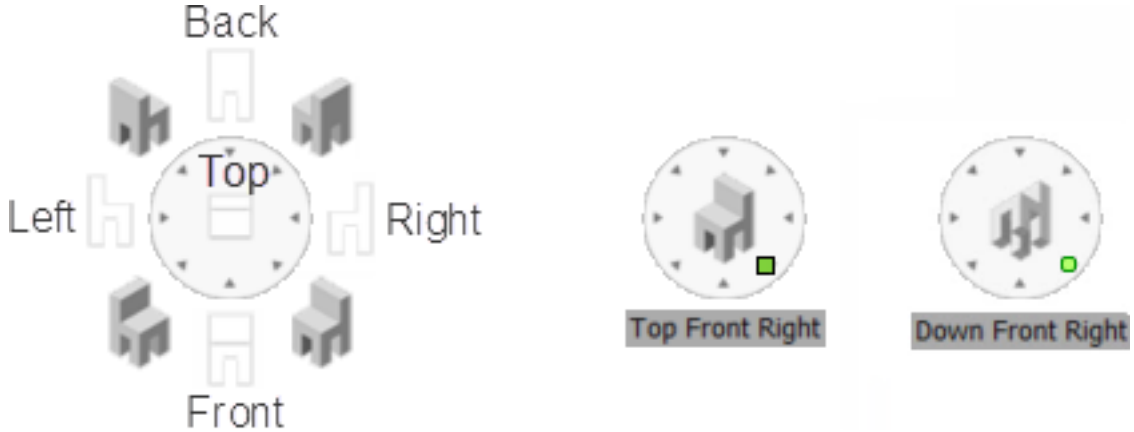
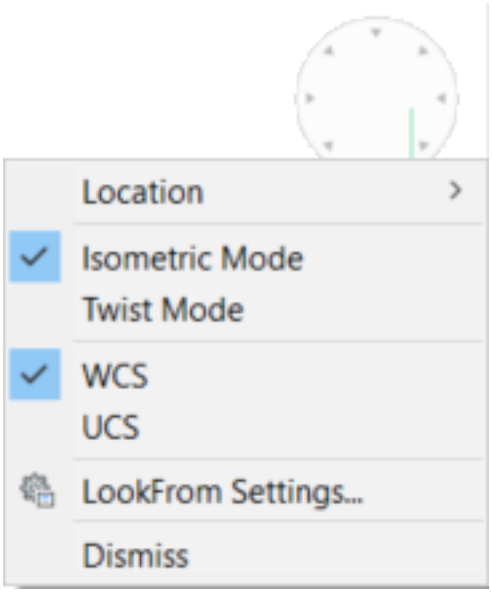


Figure 1.788


Location
Isometric Mode
Twist Mode
WCS
UCS
LookFrom Settings...
Dismiss

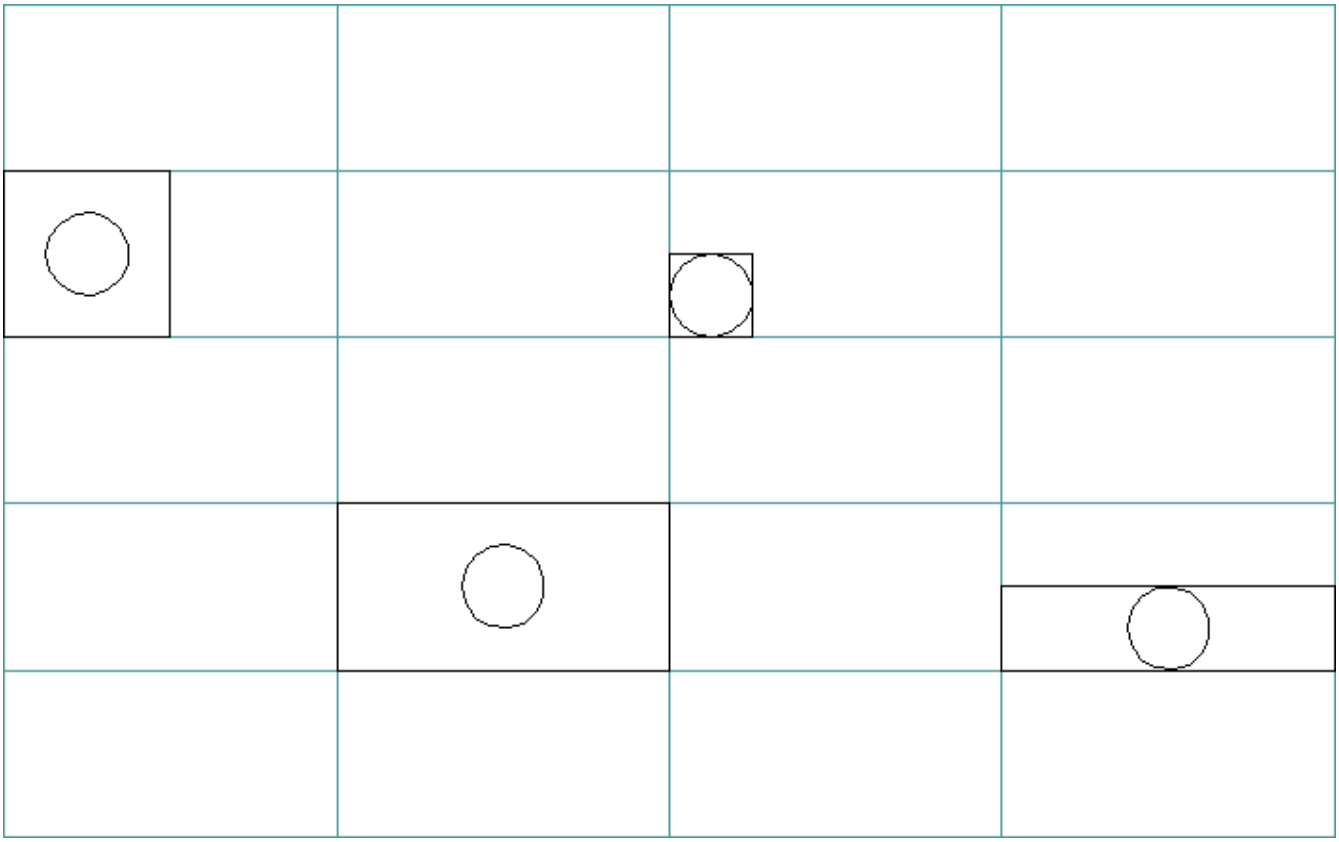
1.18.6 Dynamic blocks vs. 2D Parametric Blocks

If you come from an AutoCAD® design environment, you may be familiar with Dynamic Blocks. You may also be familiar with 2D parametric constraints. They are two different functionalities that somewhat overlap. If you insert AutoCAD dynamic blocks into your BricsCAD drawing, you can, in most cases, edit those block **instances**. You cannot, however, create or edit dynamic block **definitions** in BricsCAD. Instead of offering dynamic block functionality, BricsCAD focuses entirely on parametrics. It allows you to create both 2D and 3D block definitions using the same set of tools and workflow. If you've never worked with parametric constraints, they may sound scary. But, they're quite intuitive and logical. And, incredibly powerful!

To define a parametric 2D light fixture block

If you create architectural lighting plans, you probably have a library of blocks like these: 2' x 2', 1' x 1', 2' x 4', 1' x 4' (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture01.png}) Using traditional blocks, each symbol requires a separate block definition. And, if you want to change your design to use a different size, you must delete the existing block instances and insert new ones. Defining a single parametric block to accommodate all the potential sizes, is a great solution!

Figure 1.789

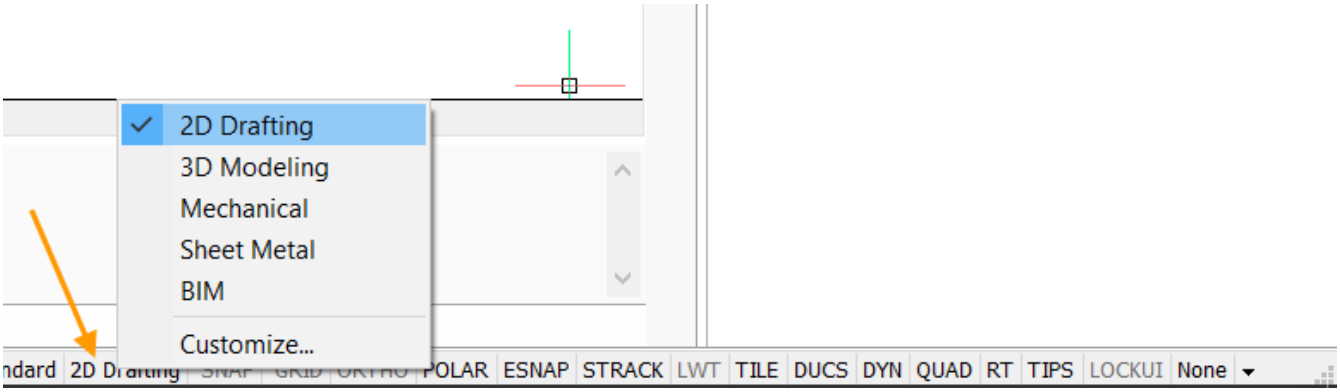


Set the workspace To ensure all the necessary tools are easily accessible, it's helpful to set the 2D drafting workspace. For this exercise, I'll use the Quad and other default UI elements. You can, of course, use the Command line as well.

1. Right-click on the current workspace in the status bar.
2. Select 2D Drafting.
3. Start a new drawing.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture02.png})

Figure 1.790



Draw geometry Draw the basic geometry for the block you want to define. The exact size doesn't matter but it's helpful to draw it close to the scale you want. If you already have the geometry as a block, open the block as its own drawing (DWG).

1. Right-click in the drawing area to display the Quad.
   - From the Draw tab, choose Rectangle. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCA
     2D-Parametric-Block-LightFixture03.png})
   - Enter 0,0 for the first corner of the rectangle. This will be the insertion point of the block.
   - Enter 48
   - Press the Tab key
   - Enter 24
2. Right-click in the drawing area to display the Quad.
   - From the Draw tab, choose Circle. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3D
     2D-Parametric-Block-LightFixture04.png})
   - Pick a point inside the rectangle to place the center of the circle.
   - Enter 6 for the radius.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture05.png})

Figure 1.791

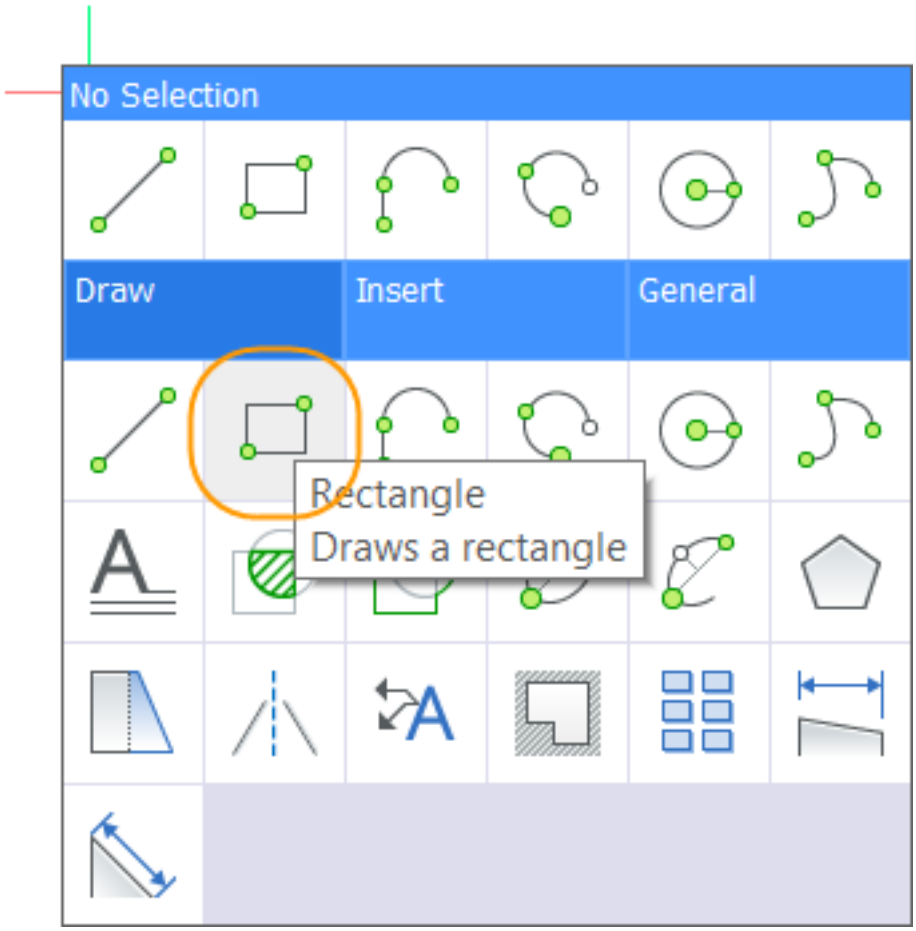

No Selection
Draw
Insert
General
Rectangle
Draws a rectangle

Figure 1.792

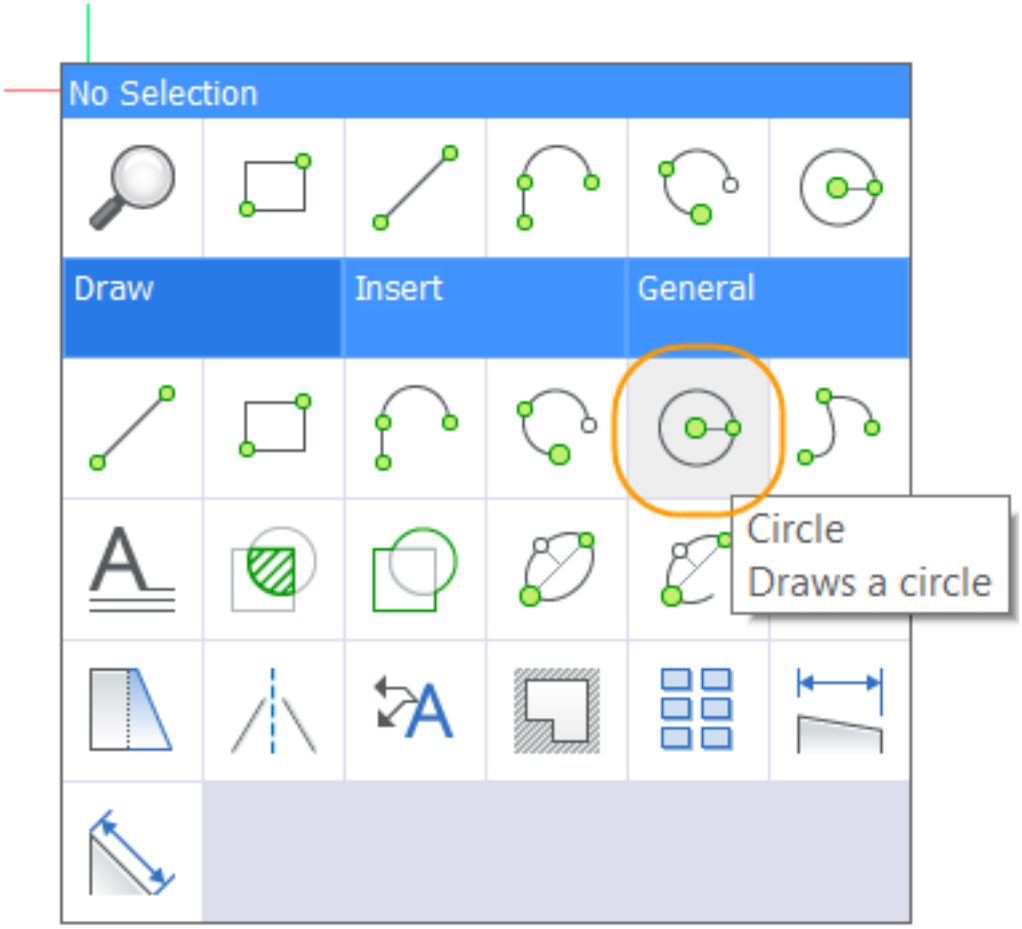
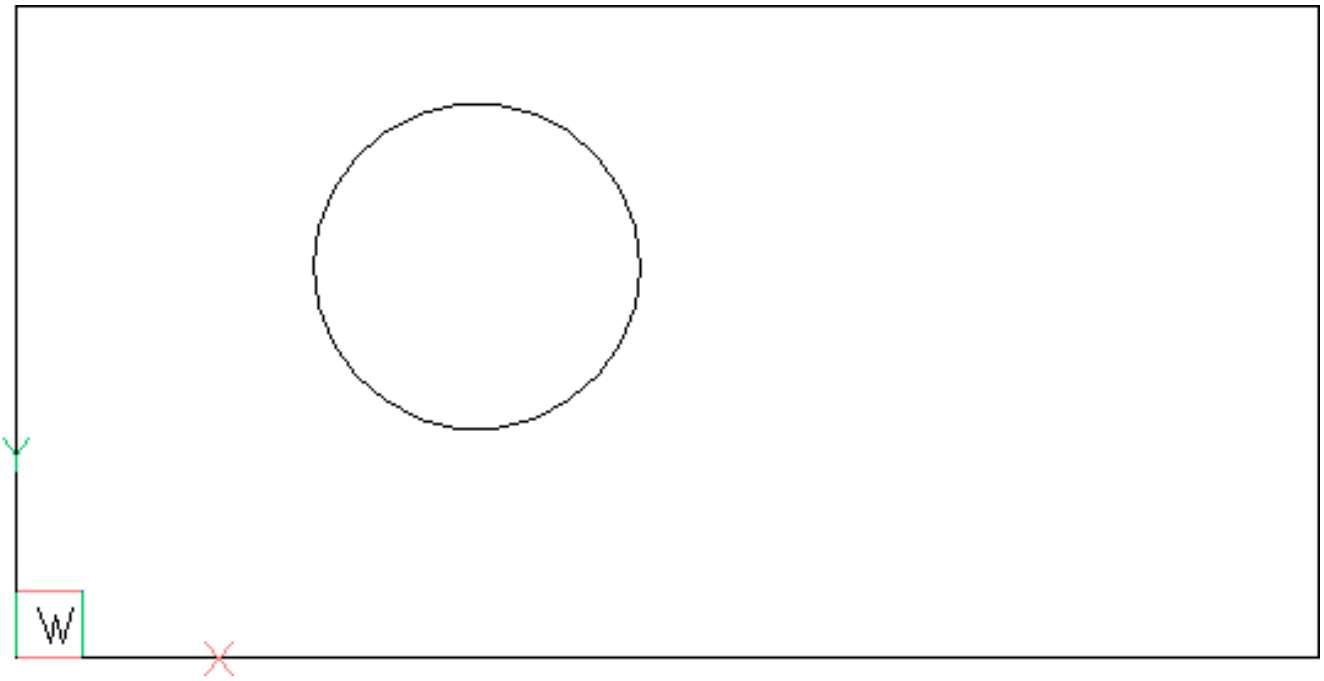


Figure 1.793



Add geometric constraints to the rectangle A rectangle is nothing more than a closed polyline. If you try to edit it, it may lose its rectangular shape. To ensure a rectangle maintains its shape, you must add geometric constraints. They establish non-numerical relationships between the various parts of geometry.

1. From the Parametric menu, choose Fix and select the left endpoint of the bottom line of the rectangle. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture06.png}) Applying a Fix constraint at the insertion point of the block ensures that point will remain fixed even though geometry may move and resize around it.
2. From the Parametric menu, choose Perpendicular and select the two lines at the lower left corner of the rectangle to ensure they always remain perpendicular to each other.
3. From the Parametric menu, choose Parallel and select the two horizontal lines to ensure they always remain parallel to each other.
4. From the Parametric menu, choose Parallel and select the two vertical lines to ensure they always remain parallel to each other.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture07.png})

Figure 1.794

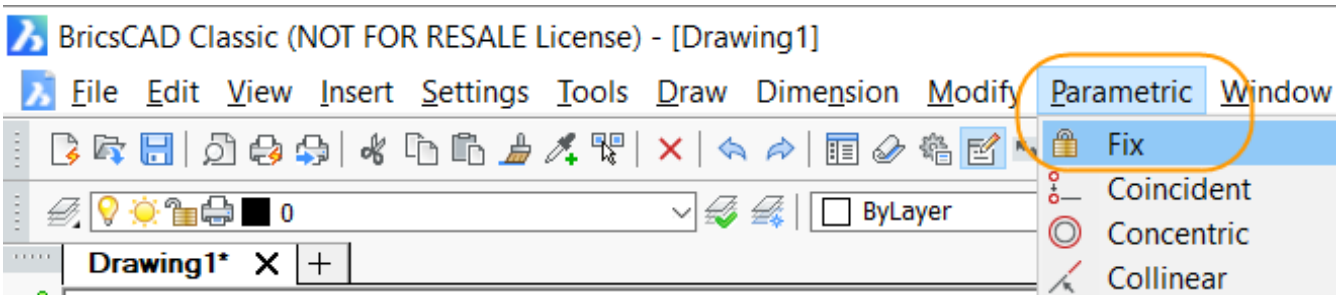
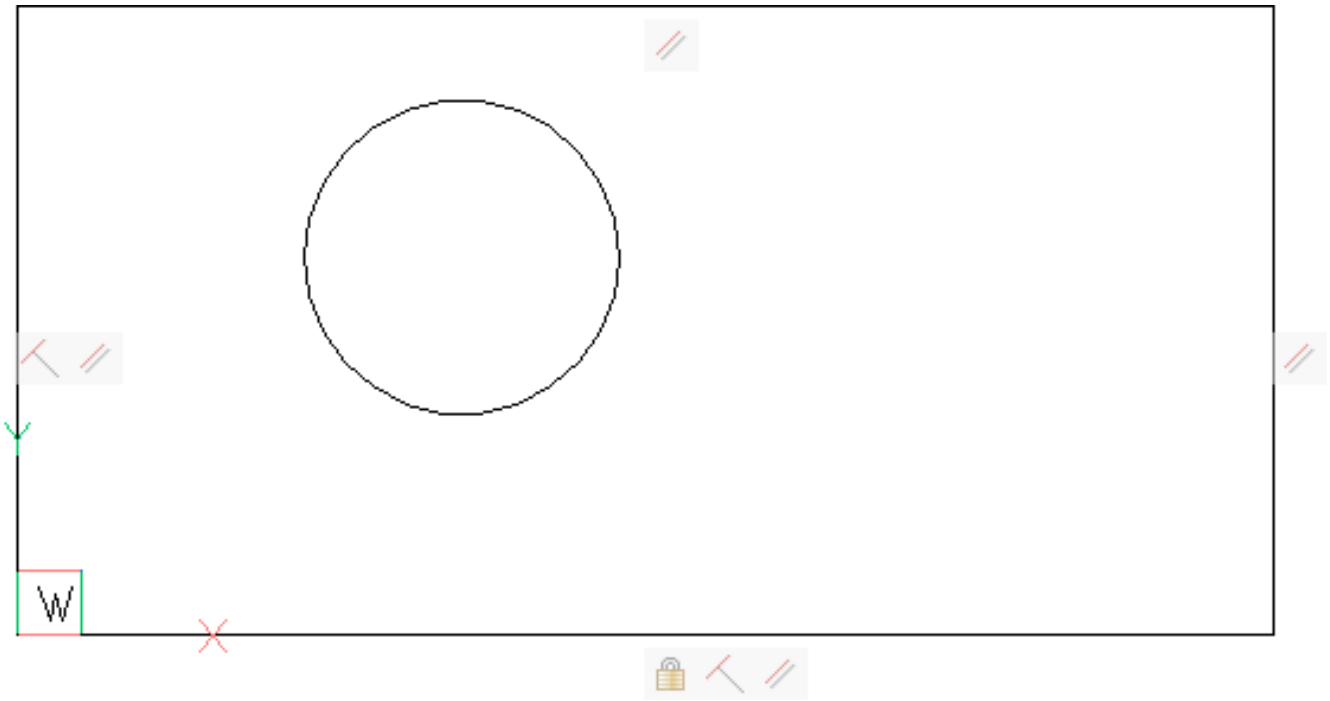


Figure 1.795



Add dimensional constraints to the rectangle and circle Dimensional constraints drive the size of geometric elements and the distances or angles between them.

1. From the Parametric menu, choose Linear and press enter to accept the default <Entity> option. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture08.png})

   - Select the left vertical line.
   - Place the dimension to the left of the rectangle.
   - Press Enter to accept the default value.

2. Press Enter twice to repeat the previous command and <Entity> option.

   - Select the bottom horizontal line.
   - Place the dimension below the rectangle.
   - Press Enter to accept the default value.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture09.png})

3. Press Enter to repeat the previous command.

   - Select the bottom endpoint of the left vertical line and the center of the circle.
   - Place the dimension to the left of the rectangle.
   - Press Enter to accept the default value.

4. Press Enter to repeat the previous command.

   - Select the left endpoint of the bottom horizontal line and the center of the circle.
   - Place the dimension below the rectangle.
   - Press Enter to accept the default value.

After adding the dimensional constraints, your drawing should look like the following image. The dimension values between the circle and rectangle will vary but you'll fix that next. You'll also change the default names (d1, d2, d3, etc) to something more meaningful. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeli
2D-Parametric-Block-LightFixture10.png})

Figure 1.796

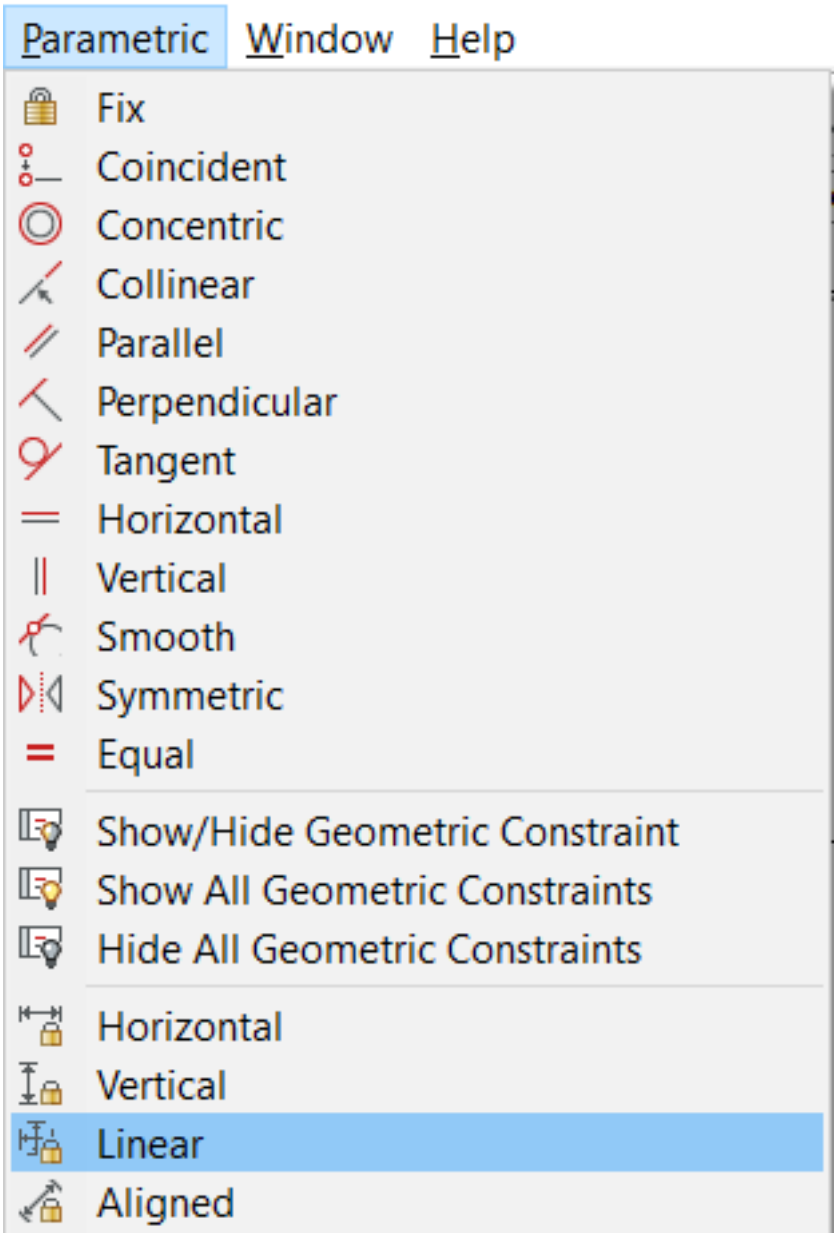

Parametric
Window
Help
Fix
Coincident
Concentric
Collinear
Parallel
Perpendicular
Tangent
Horizontal
Vertical
Smooth
Symmetric
Equal
Show/Hide Geometric Constraint
Show All Geometric Constraints
Hide All Geometric Constraints
Horizontal
Vertical
Linear
Aligned

Figure 1.797

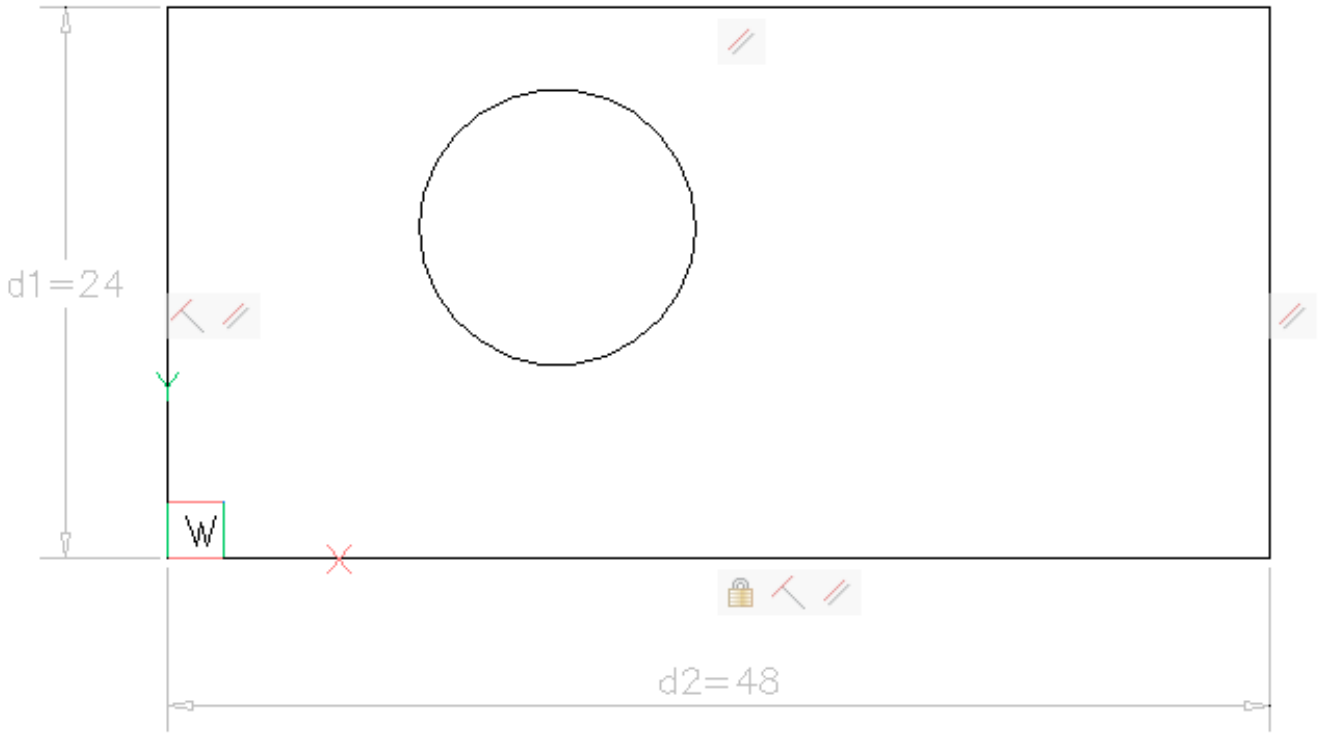
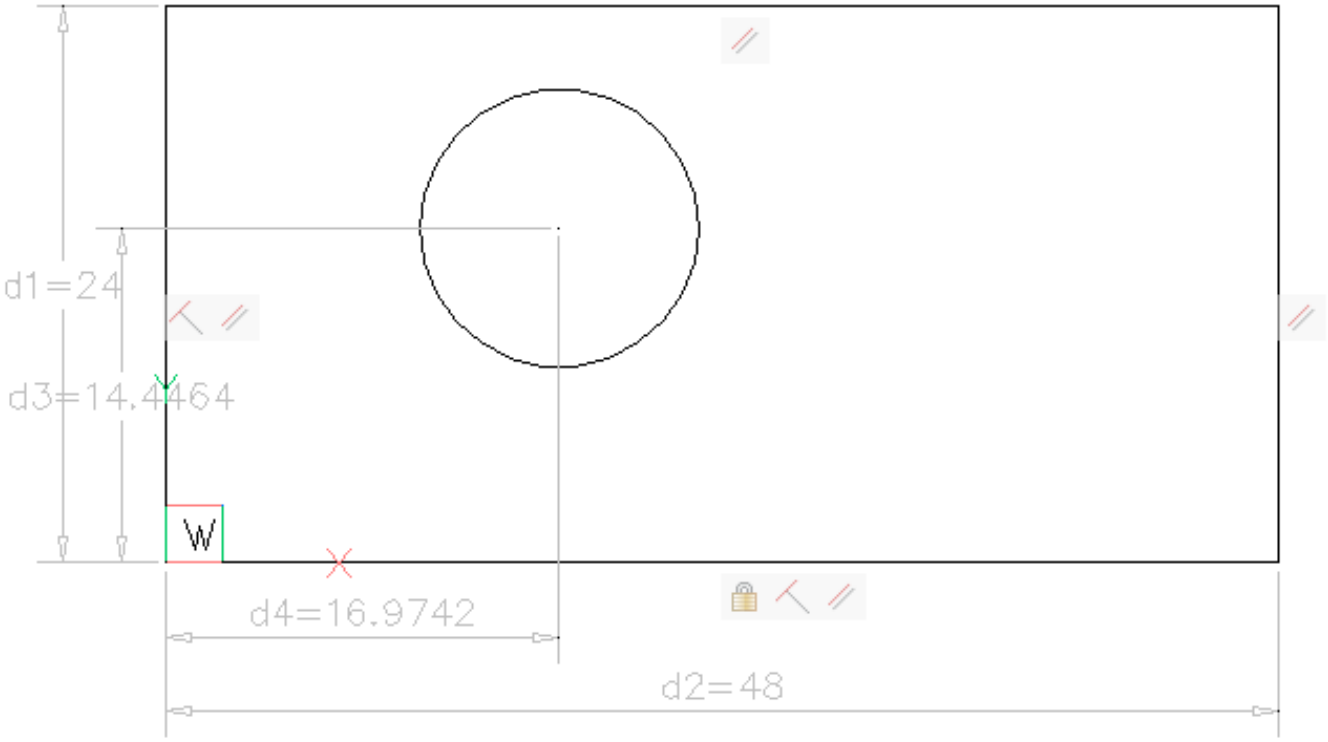


Figure 1.798

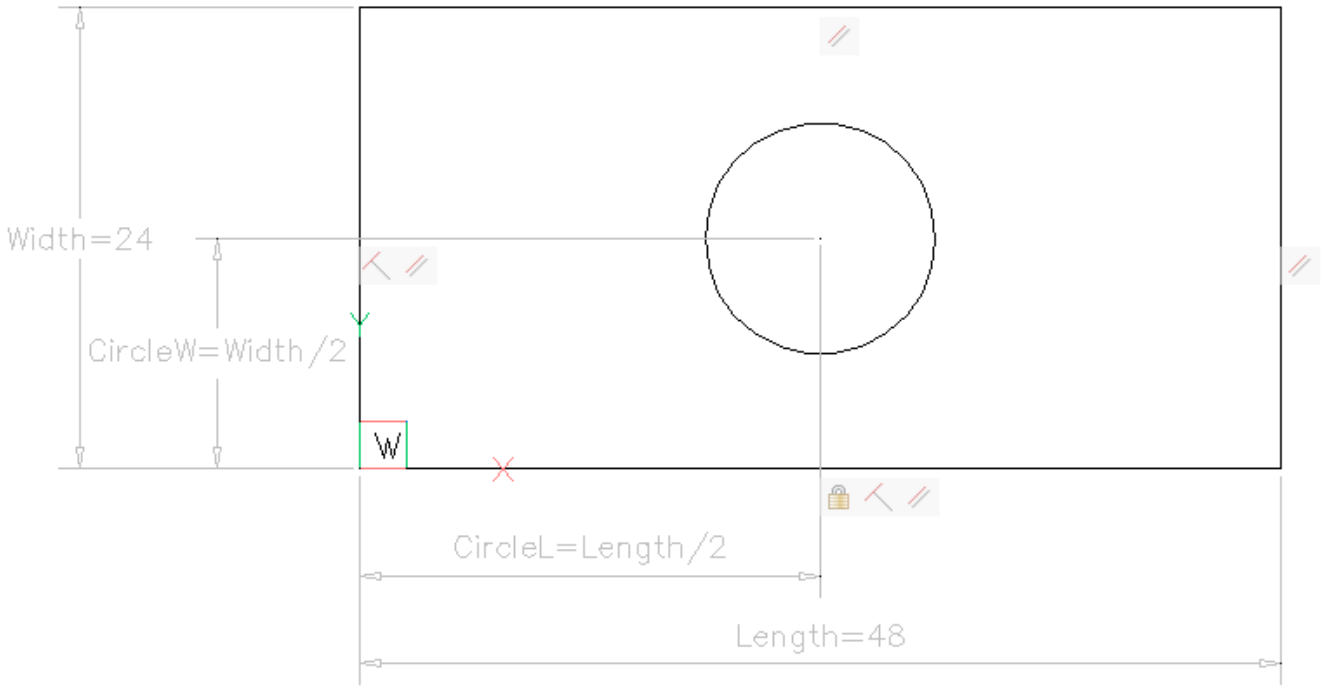


Edit dimensional constraint values You accepted the default value for each dimensional constraint as you created it, but you could have entered different values. Either way, you can change the dimension values later. You can also enter expressions as values to create dependencies on other dimensions. In this example, you always want the circle to remain at the center of rectangle. Therefore, the distance from the lower left corner of the rectangle to the center of the circle should be half the width and length of the rectangle.

1. Select the vertical dimension representing the width of the rectangle and, in the Properties panel, change its name to Width.
2. Select the horizontal dimension representing the length of the rectangle and, in the Properties panel, change its name to Length.
3. Select the vertical dimension representing the distance to the center of the circle. Change its name to CircleW and its expression to Width/2.
4. Select the horizonal dimension representing the distance to the center of the circle. Change its name to CircleL and its expression to Length/2.

As you change the expressions for the circle dimensions, you'll see the circle move to the center of the rectangle. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture11.png})

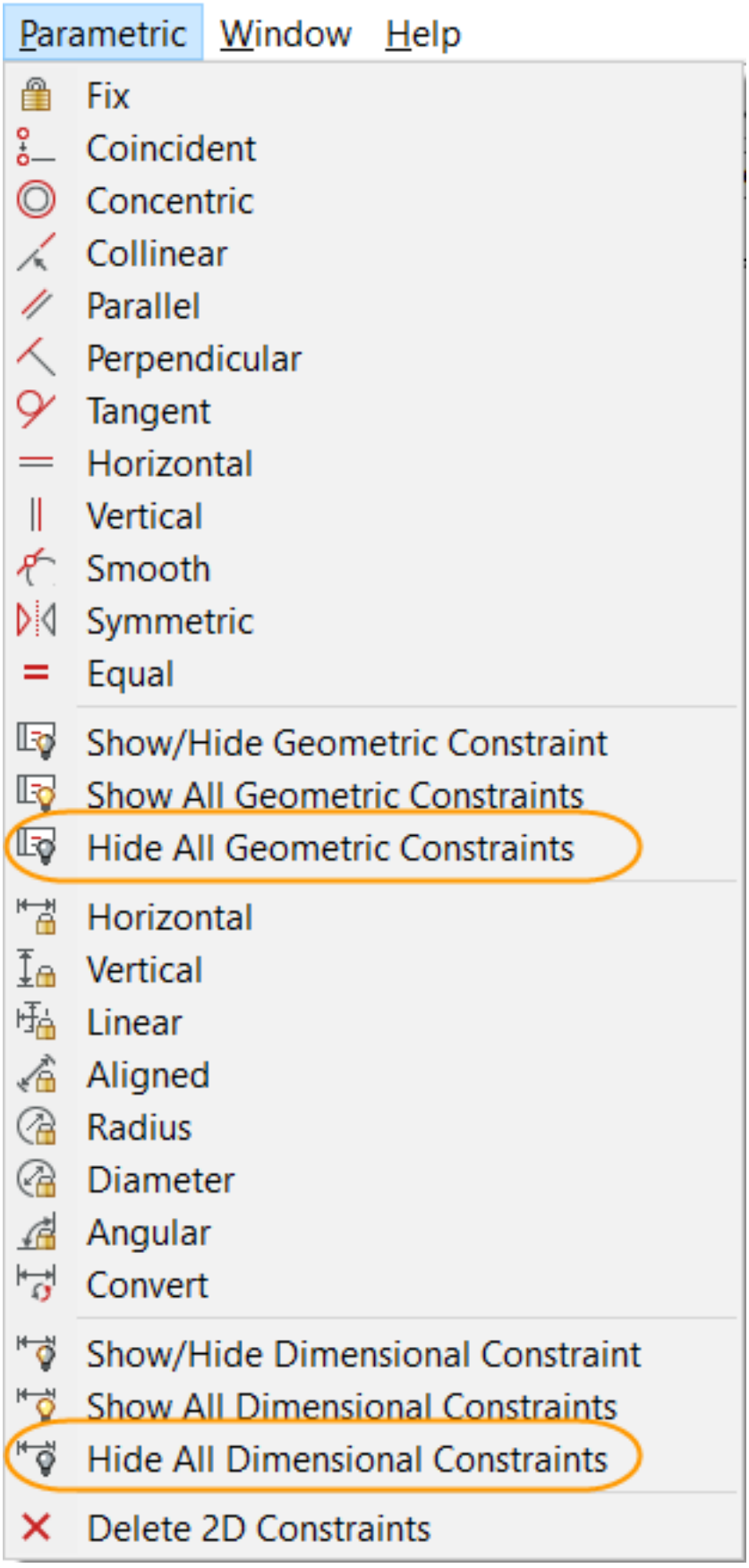
Figure 1.799



Save the parametric drawing To take advantage of parametric blocks, each block must be saved as its own drawing (DWG) file. And if you don't want the constraints to show on the inserted block, you must hide them when you save the block definition (DWG).

1. From the Parametric menu, choose Hide All Geometric Constraints.
2. From the Parametric menu, choose Hide All Dimensional Constraints. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys
2D-Parametric-Block-LightFixture12.png})
3. From the File menu, choose SaveAs and enter 2D Recessed Troffer for the file name.

Figure 1.800

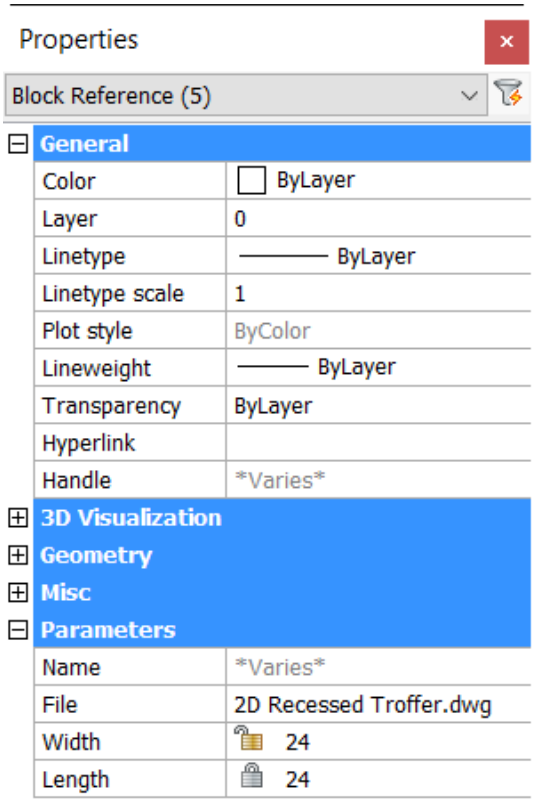

Parametric
Window
Help
Fix
Coincident
Concentric
Collinear
Parallel
Perpendicular
Tangent
Horizontal
Vertical
Smooth
Symmetric
Equal
Show/Hide Geometric Constraint
Show All Geometric Constraints
Hide All Geometric Constraints
Horizontal
Vertical
Linear
Aligned
Radius
Diameter
Angular
Convert
Show/Hide Dimensional Constraint
Show All Dimensional Constraints
Hide All Dimensional Constraints
Delete 2D Constraints

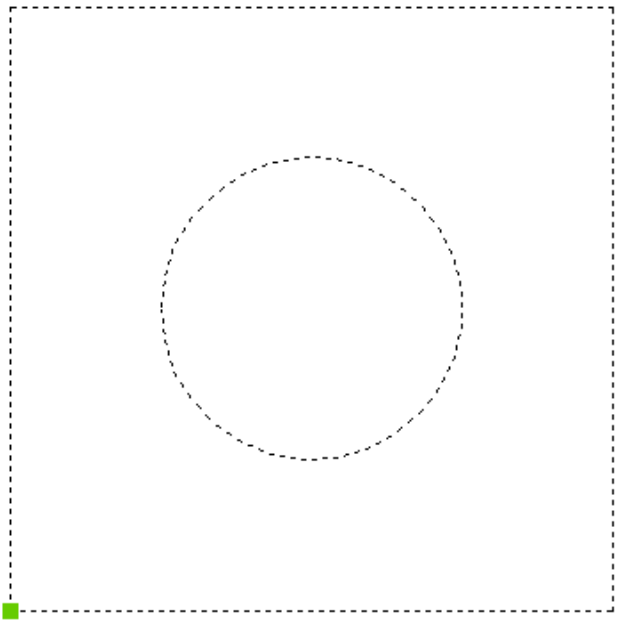
To insert a parametric drawing as a block

All the tools you used to define the 2D parametric drawing are available in every edition of BricsCAD including BricsCAD Classic. However, to insert the drawing as a parametric block, you must have the Platinum edition.

1. At the Command line, enter RUNASLEVEL and choose the BricsCAD option to ensure it's set to Platinum. If it wasn't, you must close and relaunch BricsCAD after changing RUNASLEVEL to Platinum.
2. Open the drawing in which you want to insert the 2D Recessed Troffer.
3. At the Command line, enter BMINSERT and select **2D Recessed Troffer.dwg**.
   - Choose the Multiple option.
   - Place several instances of the block in your current drawing.
4. Select any of the inserted blocks and open the Properties panel.
5. Modify the Width or Length parameters. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3
   2D-Parametric-Block-LightFixture13.png})

Figure 1.801


Properties
Block Reference (5)
General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Plot style
ByColor
Lineweight
ByLayer
Transparency
ByLayer
Hyperlink
Handle
Varies
3D Visualization
Geometry
Misc
Parameters
Name
Varies
File
2D Recessed Troffer.dwg
Width
24
Length
24



1.18.7 Creating a design table to drive Parametric Blocks parameters

Create a design table

First, you must create the design table as a CSV (comma separated) file. You can use a spreadsheet app or a simple text editor. Below are the steps to use Wordpad, but you can use any app that allows you to save it as plain text with a .CSV extension.

1. Open WordPad.
2. Enter the parameter names in the top row, separated by commas. I added a parameter called “Size”. It represents the names of the different variations. For example, I might call them Small, Medium, Large, and XLarge. But, I named them according to their size in feet, as is common here in the United States. The Width and Length refer to the dimension constraints that will drive the size (in inches).
3. Add the sizes you want to include with their correspondingWidth and Length values. I’ve only added four sizes, but you can have a long list with any number of size combinations. The image below shows what it looks like in Excel (left) and in WordPad (right). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DMo 2D-Parametric-Block-LightFixture14.png})
4. Save the file as CSV format. It’s plain text separated by commas with a .CSV file extension.

Figure 1.802

	A	B	C
1	Size	Width	Length
2	1x1	12	12
3	2x2	24	24
4	2x4	24	48
5	1x4	12	48

```
Size,Width,Length
1x1,12,12
2x2,24,24
2x4,24,48
1x4,12,48
```

Add the design table to the parametric block

Next, you must associate the design table with the parametric block drawing.

1. In BricsCAD, open **2D Recessed Troffer.dwg**. This is the block definition drawing, not a drawing with the block inserted.
2. Right-click on any UI element and turn on the Mechanical Browser (3.2.2) panel. In the Mechanical Browse (3.2.2), you will see the list of parameters and their values. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e 2D-Parametric-Block-LightFixture16.png})
3. At the Command line, enter DESIGNTABLE (6.10.26) and press enter to accept the default option <From file>
4. Select the .csv file you created. Notice the Size parameter is added to the Mechanical Browser (3.2.2) with the Width and Length parameters grouped under it.
5. Check the Value property for each of the main parameters: Size, Width, and Length. You can select the default values to use when you insert the block. For example, if you most often use 2'x4' fixtures in your designs, set that size as the default. The corresponding Width and Length values will follow. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture17.png})
6. Check the Exposed property for each of the main parameters: Size, Width, and Length. They should all be set to Auto. I'll explain these settings after we insert the block into a drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture18.png})
7. Save the block drawing.

Figure 1.803

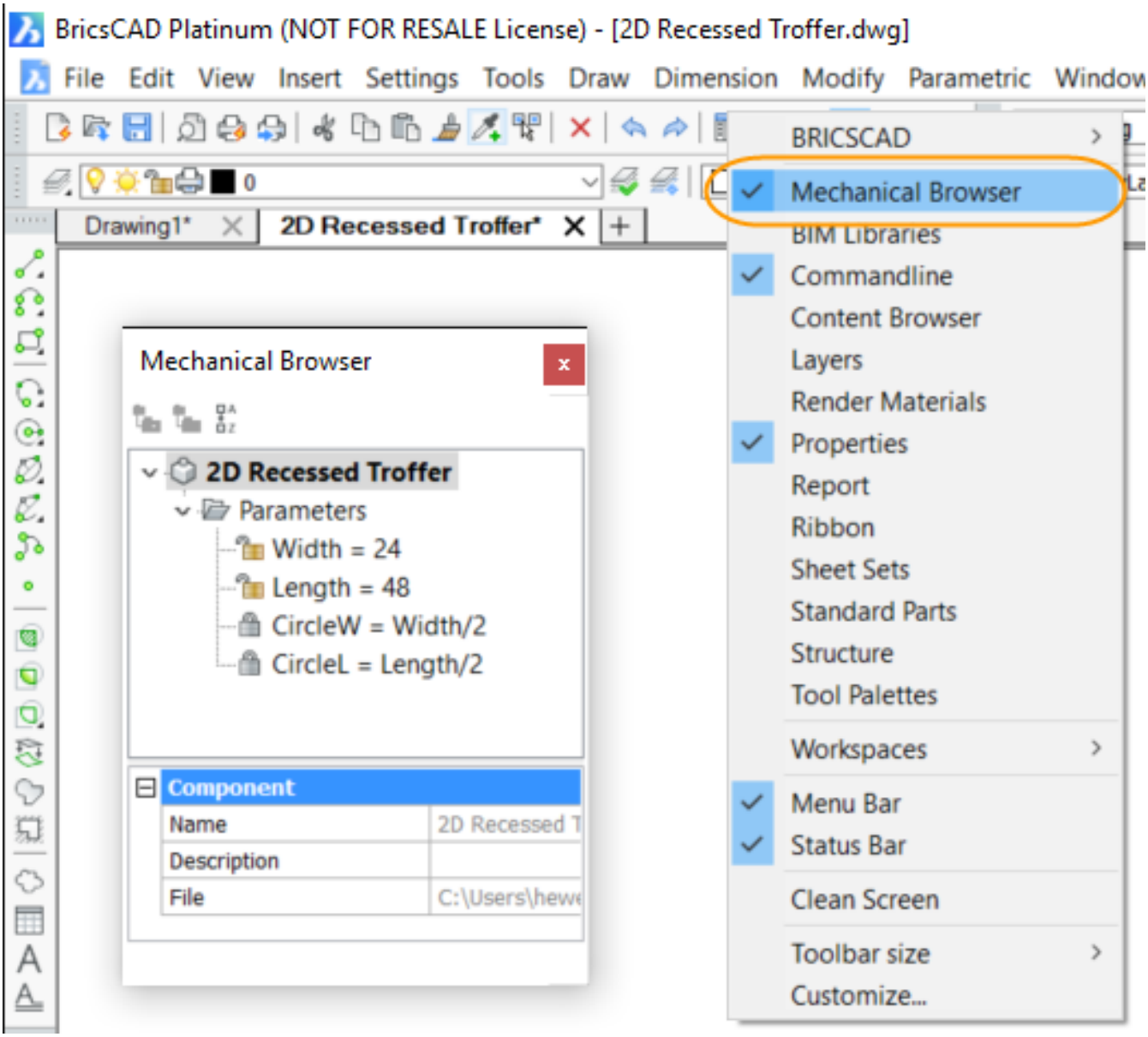


Figure 1.804

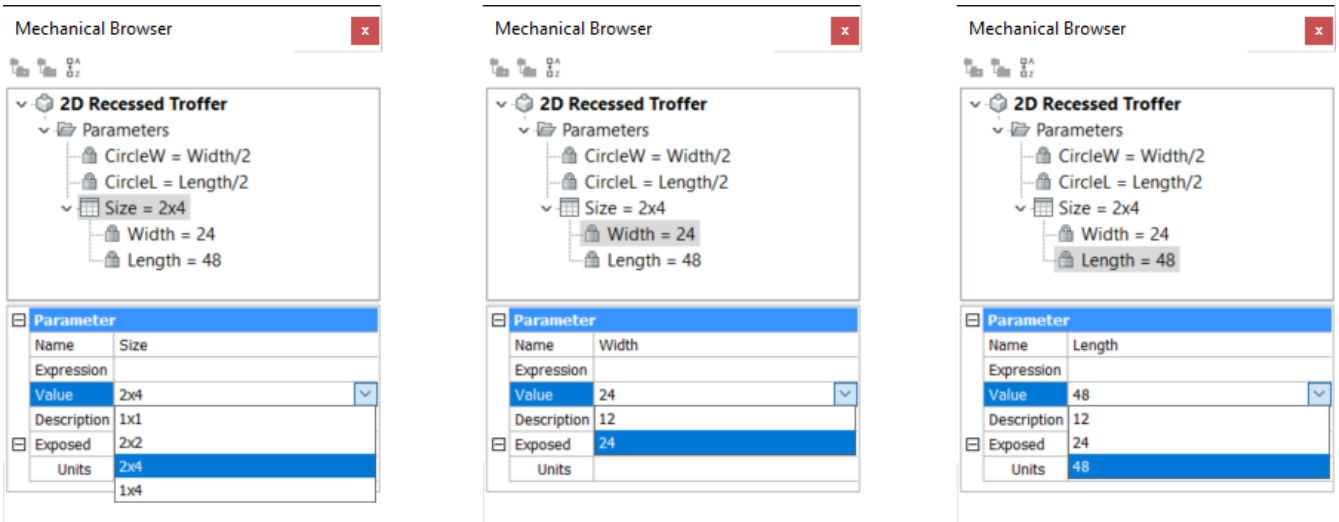
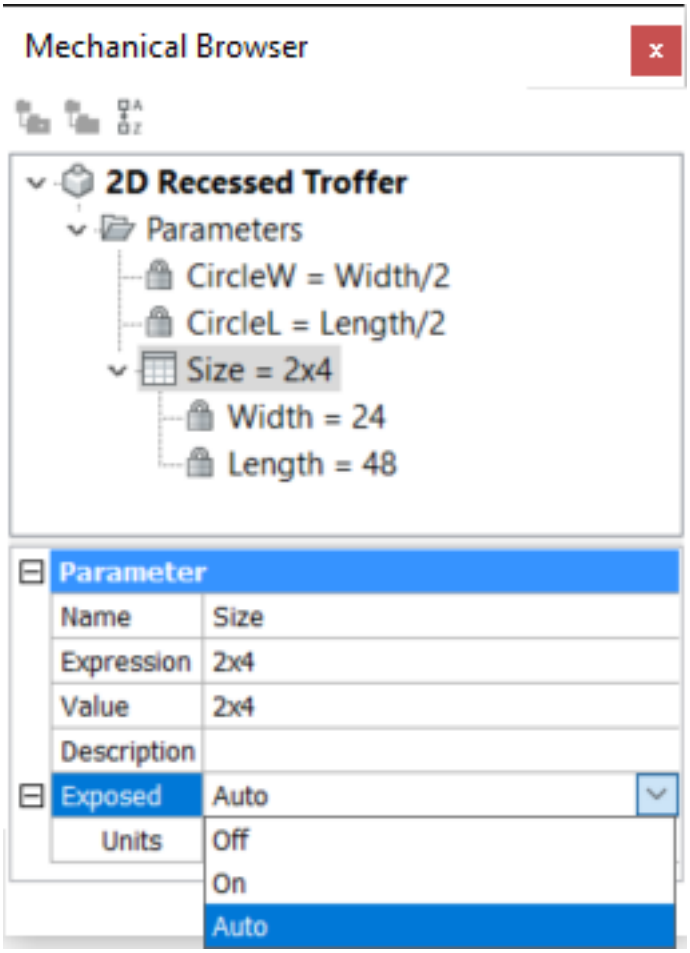


Figure 1.805

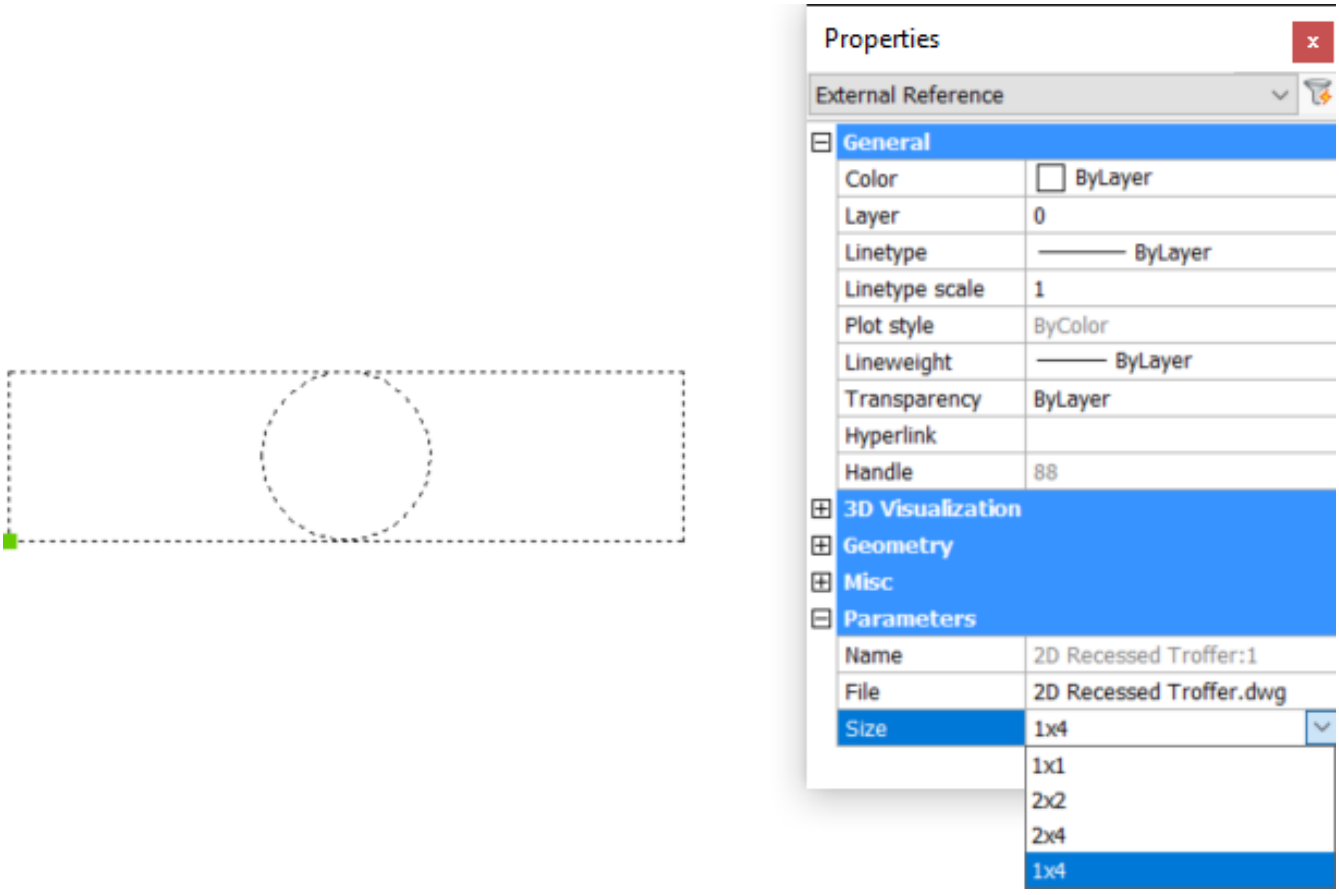

Mechanical Browser
2D Recessed Troffer
Parameters
CircleW = Width/2
CircleL = Length/2
Size = 2x4
Width = 24
Length = 48
Parameter
Name
Size
Expression
2x4
Value
2x4
Description
Exposed
Auto
Units
Off
On
Auto

Modify the parametric block instance

Then, you can insert the parametric block into a drawing and test it out1

1. Open the drawing in which you want to insert the light fixture. If a block definition of the same name already exists in the current drawing, you may have to rename or erase/purge the old definition.
2. At the Command line, enter BMINSERT and select **2D Recessed Troffer.dwg** to insert it as a block in your current drawing.
3. Select the inserted block.
4. In the Properties panel, select a different size under the Parameters pane. Now you can only select from the sized in the design table. This helps ensure accuracy and compliance with standards. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture19.png})

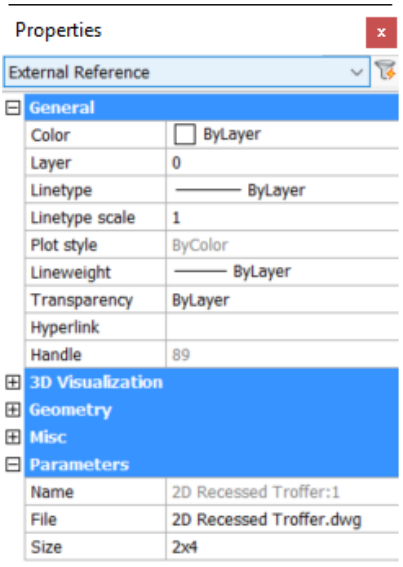
Figure 1.806



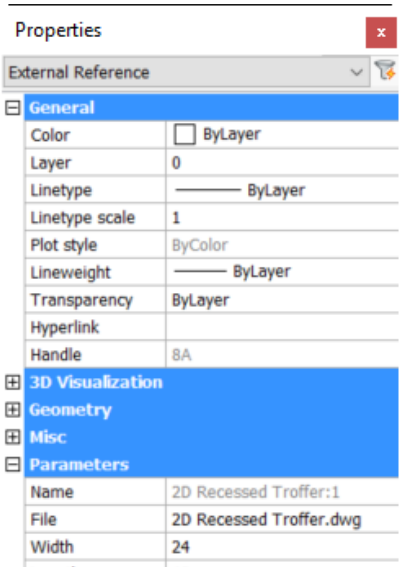
Exposing parameters

When you add the design table to the block, the Exposed property for all three parameters is set to Auto by default. In our example, that means the Size is exposed for modifying the block insertion in the Properties panel but the dependent parameters, Width and Length, are not. You can explicitly set the Exposed property in the block definition to On or Off depending which parameters you want available for modifying the block insertion. Here are a few examples. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/3DModeling/TIFY-2D-Parametric-Block-LightFixture20.png})

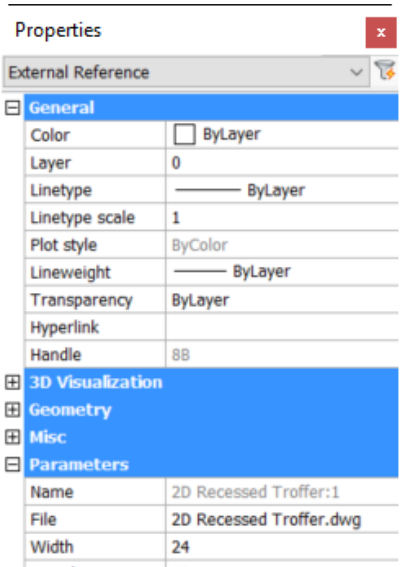
Figure 1.807



Width = Auto or Off
Length = Auto or Off
Size = Auto or On



Width = On
Length = On
Size = Off



Width = On
Length = On
Size = Auto or On

1.18.8 Create Site

Command: Site (6.27.41) **Overview:**In BricsCAD, topographical surfaces are created using the **Create Site** tool.

- Toposurface can be created from a selection set of points and polylines.
- Importing points file, such as survey points data obtained from a surveyor. Use these points to create a topographical surface.
- Points placed in the drawing area set by elevation and can be used for creating topographical surfaces.
- The linear entities can be added as breaklines or points to the Site.

Tutorial: How to create a Site Click here (2.3.17) to watch

About

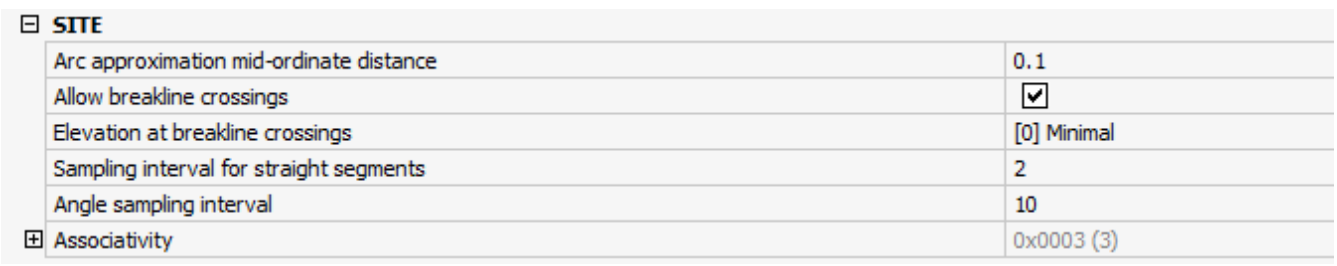
Creating a topographical surface is a process that needs to befollowed tomap a particular area in detail. On BIM models, these topographic surfaces are used to represent the site.The topographical surface can be generated based on drawing entities, data imported from a comma delimited file format (CSV) or text file format (TXT), and points placed at different elevations.  Keyboard: **site** Menu:**BIM | Create Site** Toolbar: **BIM Site Design** Quad Tab: **Model | Create Site** Ribbon Tab | Panel:**Data | Site | Create Site Setting the site preferences**

1. Open the Settings dialog.
2. Under **BIM** options, expand the **Site** settings category.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/Sitesettings.png})

Arc approximation mid-ordinate distance	Mid-ordinate distance is the maximal distance between arc and chord (straight) segment which is used for arc approximation. Defines how many points along the curve you want to add. When you have polylines with curves, this parameter allows you to tessellate the arcs in the polyline. The default value of this parameter is variable in Bricsys predefined templates. For instance, the distance value is set to **0.01** in **BIM-m**, **1** in **BIM-cm**, **10** in **BIM-mm**, and **0.4** in **BIM imperial** templates.
Allow breakline crossings	Defines if crossing breaklines are allowed. By default, it is set to ”On”. If breakline crossings are allowed, intersections between breakline segments are calculated and added as points to the TIN surface.
Elevation at breakline crossings	Defines how elevation at crossing breaklines should be determined.There are three options (e.g., minimal, maximal, and average) for defining breakline elevation at the intersection.
Sampling interval for straight segments	Defines the length of sampling interval which is used for sampling of straight segments (e.g. contour lines).
Angle sampling interval	Defines angle sampling interval in decimal degrees, which is used for grading rounding at convex vertices.

Figure 1.808



Associativity	Enable/disable grading and surface associativities. By default, the system variable is set to 3. • 0: Neither **Tin Surface** nor **Grading** associativity is enabled. • 1: Enables **Tin Surface** associativity. • 2: Enables **Grading** associativity. • 3: Both **Tin Surface** and **Grading** associativities are enabled.

Table 1: The site settings

Procedure: Creating a Toposurface from a selection set of points and polylines

1. Open the drawing file that contains a set of points (1) and polylines (2).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/CreateSitesetofpointpolyli

2. Launch the SITE command in the command line

Orselect Create Site tool in the Model tab from the Quad. **You are prompted:Select entities to create site or [Import from file/Place points]:**

3. Select the entities in the drawing area to create a toposurface.

Optionalprompt menu can be used to select all entities (3) in the drawing area. (\vref {fig:C:/Dev/Task-2448/images/staticbrics
Command line depicts how many entities are included in the set, e.g.Entities in set: 2044.

4. Press Enter.

The toposurface (4) is created based on selected points and polylines. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/

Figure 1.809

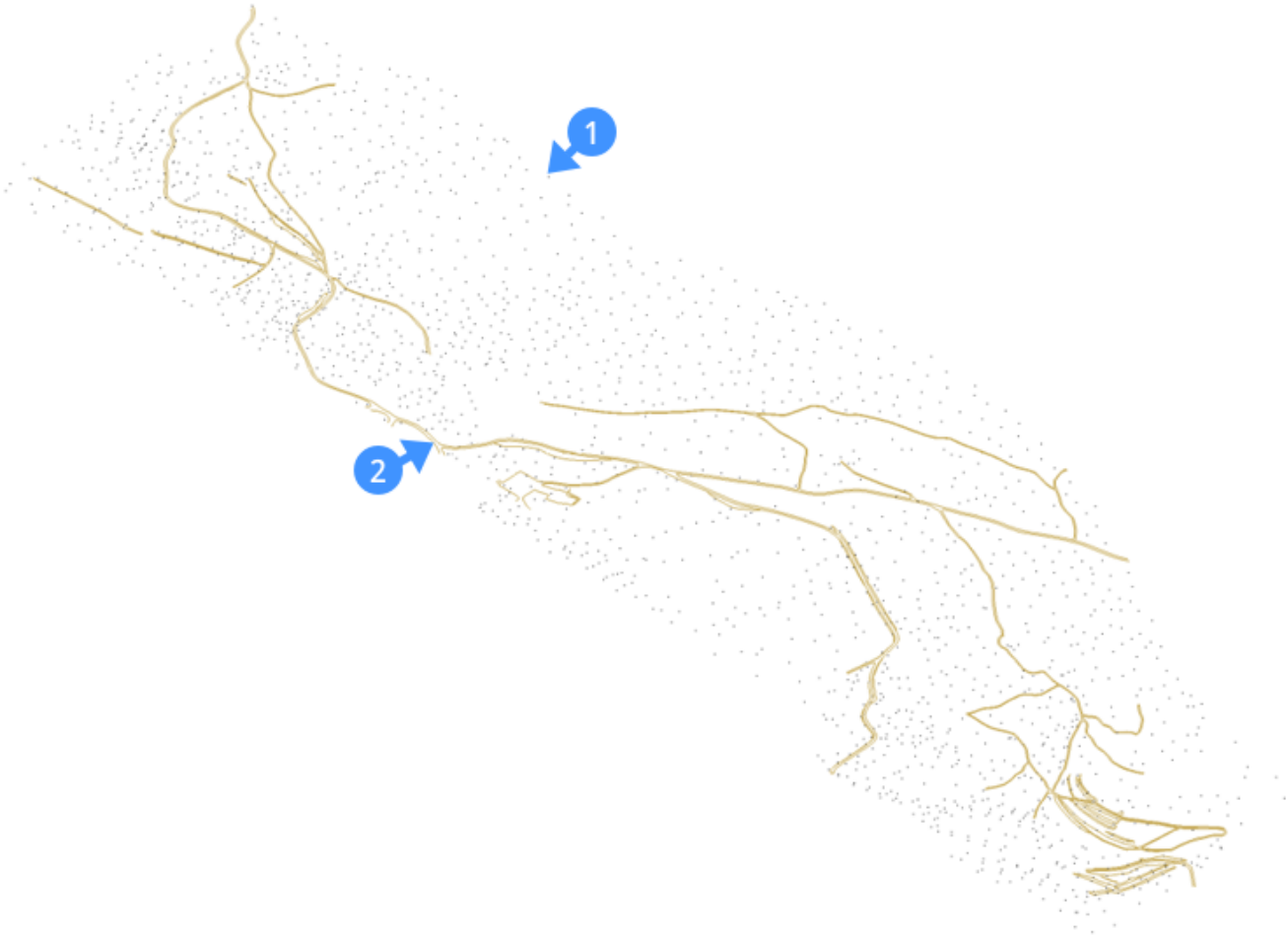


Figure 1.810

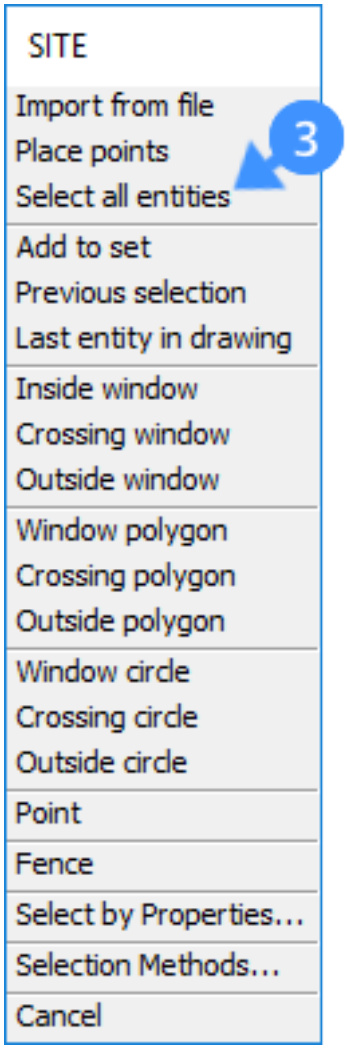
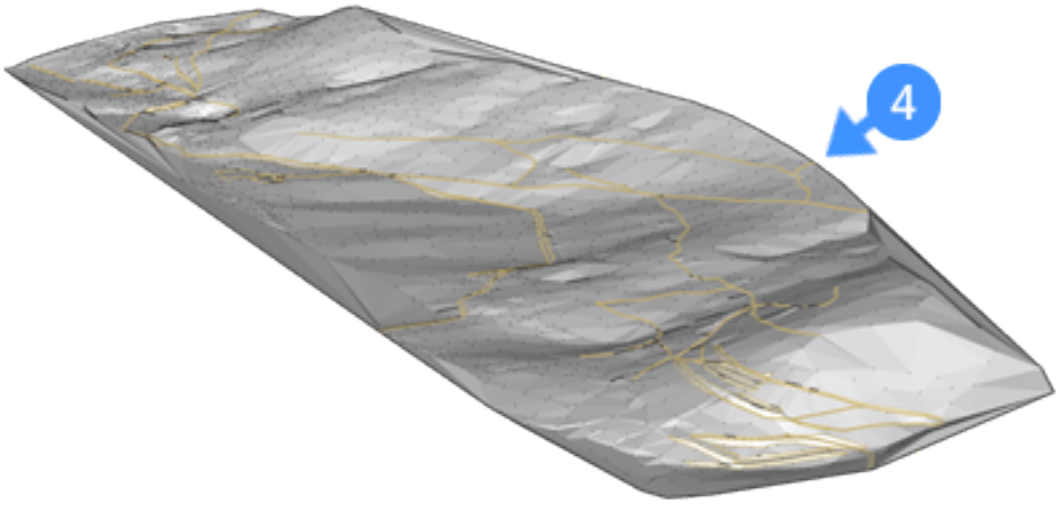


Figure 1.811



Procedure: Creating a Toposurface from an imported points file:

1. Launch the SITE command in the command line.

Orselect Create Site toolin the Model tab from the Quad. **You are prompted:Select entities to create site or [Import from file/Place points]**

2. Select the**Import from file** (5) option in the prompt menu.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SitePromptMenuImportfro
Ortype "I" in the command line. Open points file (6) dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/

3. Choose the point data file you want to open.

4. Click Open to import the points.

The toposurface (7) is imported from an existing survey points file. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
Note: If the toposurface is not shown in the current viewport, use the **Zoom Extents** option to display it to the extents of entities.

Figure 1.812

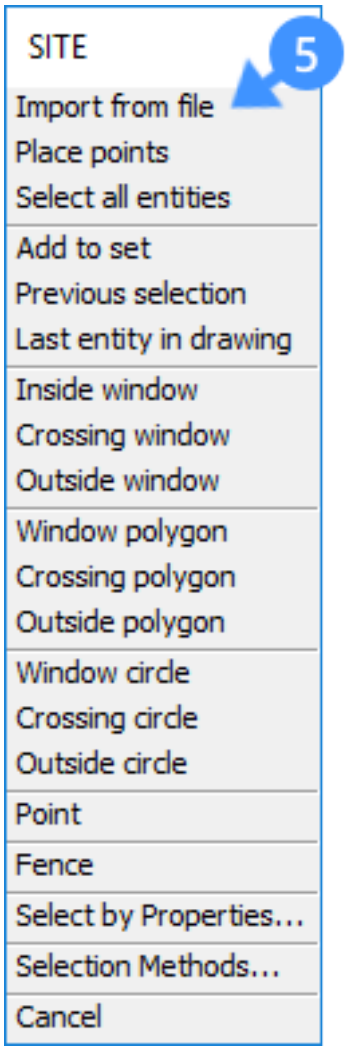

SITE
5
Import from file
Place points
Select all entities
Add to set
Previous selection
Last entity in drawing
Inside window
Crossing window
Outside window
Window polygon
Crossing polygon
Outside polygon
Window circle
Crossing circle
Outside circle
Point
Fence
Select by Properties...
Selection Methods...
Cancel

Figure 1.813

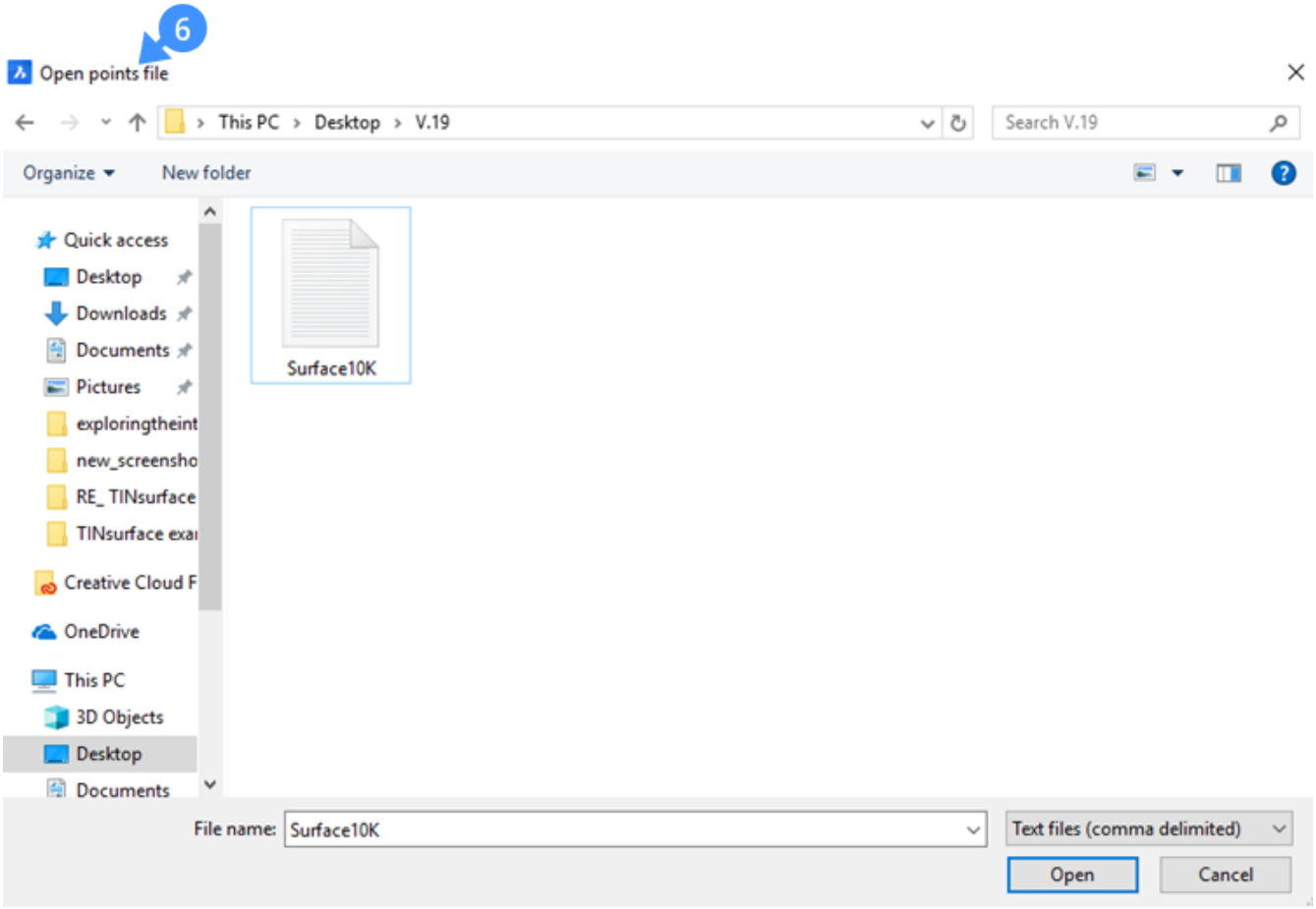
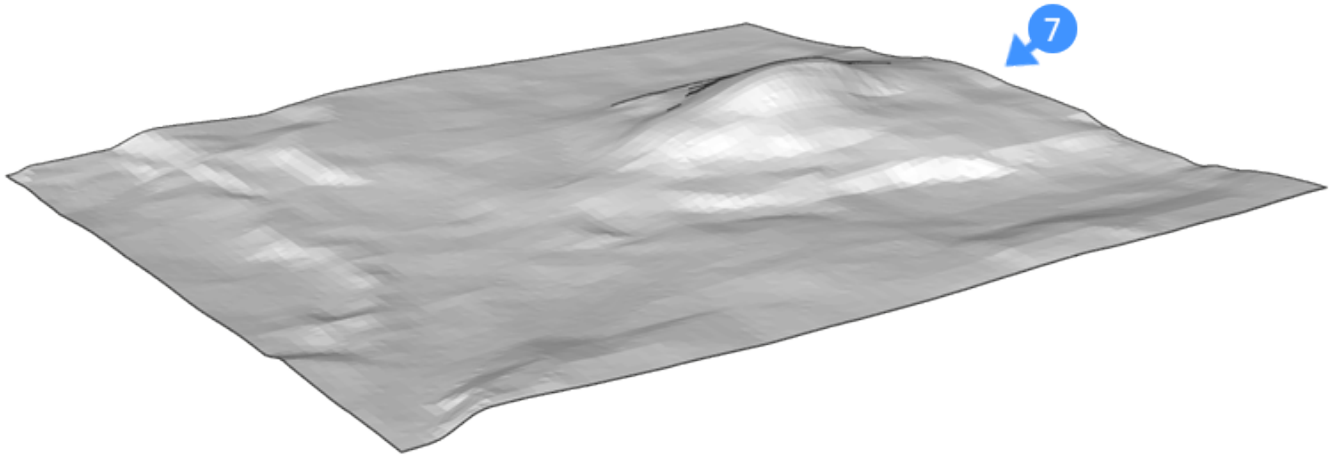


Figure 1.814



Procedure: Creating a Toposurface by placing points:

1. Open a new drawing.

Note: The new drawing starts with the default template as defined by the BASEFILE system variable.

2. Launch the SITE command in the command line.

Orselect Create Site tool in the Model tab from the Quad. **You are prompted:Select entities to create site or [Import from file/Place points]**

3. Select the**Place points**(8) option in the prompt menu.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SitePromptmenuPlacepoi
Ortype ”P” in the command line. **You are prompted:Pick point**

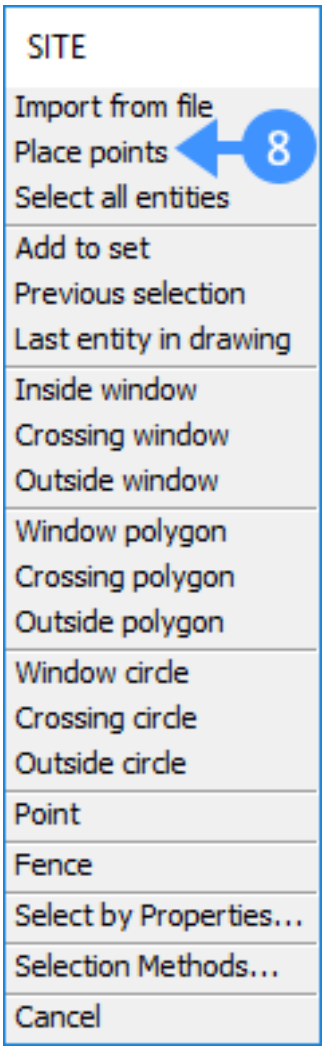
4. Pick a point where you want to start creating the toposurface.

You are prompted:Enter elevation <0.000>

5. Specify the elevation value of each those points, and press Enter to accept.

Note: At least three points are needed to create a TIN surface.

Figure 1.815


SITE
Import from file
Place points
8
Select all entities
Add to set
Previous selection
Last entity in drawing
Inside window
Crossing window
Outside window
Window polygon
Crossing polygon
Outside polygon
Window circle
Crossing circle
Outside circle
Point
Fence
Select by Properties...
Selection Methods...
Cancel

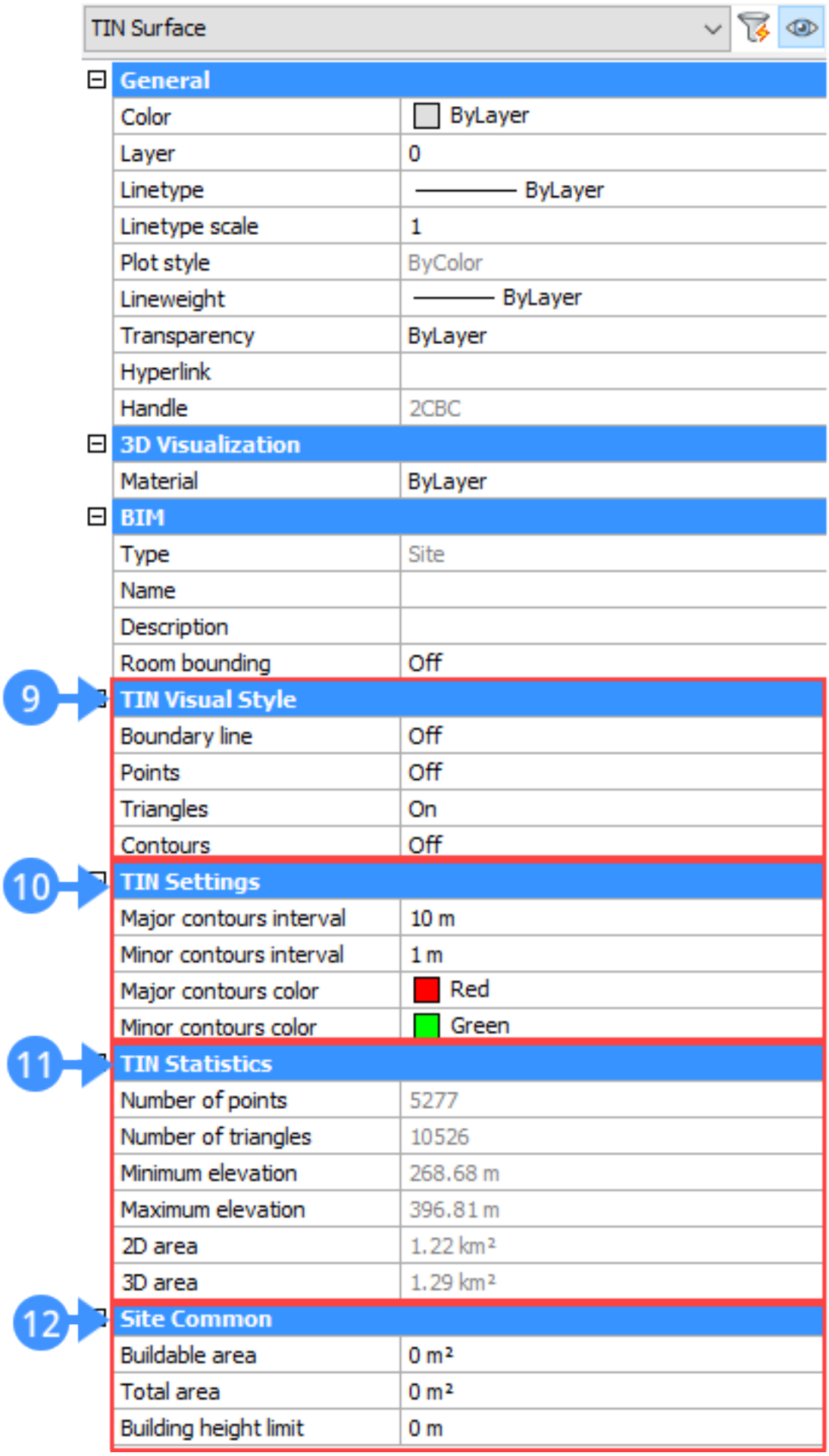
Displaying the properties of a TIN surface:

1. Select the TIN surface. The properties of the TIN Surface are displayed in the **Properties** panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/TINSurfaceProperties.png})

TIN Visual Style (9)	Controls the visibility of the following properties: • Boundary line: Represents the toposurface boundary lines. • Points: Represents the toposurface points. • Triangles: Represents how the elevation points are connected to each other. It follows the Delaunay triangulation method where breaklines are not included in densification. • Contours: Represents the joining of points of equal elevation within a line. To turn one of the properties on/off, click their field in the Properties bar, then press the arrow button and choose On/Off in the context menu.
TIN Settings (10)	Controls the contour lines settings. • Major contours interval: Sets the major contour interval. • Minor contours interval: Sets the minor contours interval. • Major contours color: Defines the major contours color. • Minor contours color:Defines the minor contours color.
TIN Statistics (11)	Demonstrates the statistical information about the TIN Surface. • Number of points: Demonstrates the number of points that define the TIN surface. • Number of triangles: Demonstrates the number of non-overlapping triangles in the TIN surface. • Miniumum elevation: Demonstrates the minimum elevation value found in the TIN surface. • Maximum elevation: Demonstrates the maximum elevation value found in the TIN surface. • 2D area: Demonstrates the 2D area that is calculated from the flat TIN surface. • 3D area: Demonstrates the 3D area that is calculated from the TIN surface.
Site Common (12)	Demonstrates the IFC common site properties. • Buildable area: The area of site utilization expressed as a maximum value according to local building codes. • Total area: Total planned area for the site. Used for programming the site space. • Building height limit: Allowed maximum height of buildings on this site according to local building codes. Source: http://www.buildingsmart-tech.org

Table 2:The properties of the TIN Surface in the properties panel.

Figure 1.816


TIN Surface
General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Plot style
ByColor
Lineweight
ByLayer
Transparency
ByLayer
Hyperlink
Handle
2CBC
3D Visualization
Material
ByLayer
BIM
Type
Site
Name
Description
Room bounding
Off
9
TIN Visual Style
Boundary line
Off
Points
Off
Triangles
On
Contours
Off
10
TIN Settings
Major contours interval
10 m
Minor contours interval
1 m
Major contours color
Red
Minor contours color
Green
11
TIN Statistics
Number of points
5277
Number of triangles
10526
Minimum elevation
268.68 m
Maximum elevation
396.81 m
2D area
1.22 km²
3D area
1.29 km²
12
Site Common
Buildable area
0 m²
Total area
0 m²
Building height limit
0 m

Adding linear entities as breaklines to the TIN surface:

1. Launch the SITE command in the command line.

Orselect Create Site tool in the Model tab from the Quad. **You are prompted:Select entities to create site or [Import from file/Place points]**

2. Select the linear entity in the TIN surface, press Enter.

You are prompted: Selection contains linear entities. Add linear entities to site as [Breaklines/Points] <Breaklines>:

3. Press Enter to add the linear entity as a breakline.

1.18.9 Grading

Command: Grading (6.15.24) **Overview:**In BricsCAD, graded surfaces are created using the**Create Grading**tool.

- Graded surfaces can be created from an existing surface.
- Defining a graded area can be accomplished using a polyline as an entity.
- Two projection methods namely 'Slope to surface' and 'Offset to slope' are supported. Slope to surface needs an input entity (e.g. polyline, line, arc, circle), surface, and slope to grade the surface. Offset to slope needs an input entity, offset, and slope.
- The graded surfaces are not editable with the SITEEDIT command.

Tutorial: How to create a Grading Click here (2.3.19) to watch

About

Grading is a set of processes to maintain a base level for construction works or site designing by defining a slope or adjusting the offset of a grading. For instance, creating; parking areas, recreation reserves or road and railway designs.

Procedure: Grading a surface to create pond

1. Open a drawing that has a topographical surface (1).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/TINSurfaceGrading.png})

2. Launch the Grading (6.15.24) command in the command line

Orselect Create Grading tool in the Model tab from the Quad. **You are prompted:Select entity:**

3. Select the entity (2) that defines the footprint area of your pond.

You are prompted: Select target surface or [slope/Offset]: Optional without selecting any target surface,click the slope/Offset option in the prompt menu or type 'O' in the command line to follow the offset-projection method. **Note:** The offset-slope projection method requires input entity, offset, and slope.

4. Select the target surface (3) to create a graded area.

You are prompted: Move mouse to change the slope of the grading:(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he

5. Define the slope of the grading by moving the mouse.

Note: Grading is attached to the input entity and it is connected with the target surface. Moving the mouse in/out dynamically increase/decrease the slope of the grading. If **Dynamic dimension (DYN)** is On, change the slope in the highlighted cell (4), and press Enter. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtec

6. Press Enter to accept the slope of the grading.

Figure 1.817

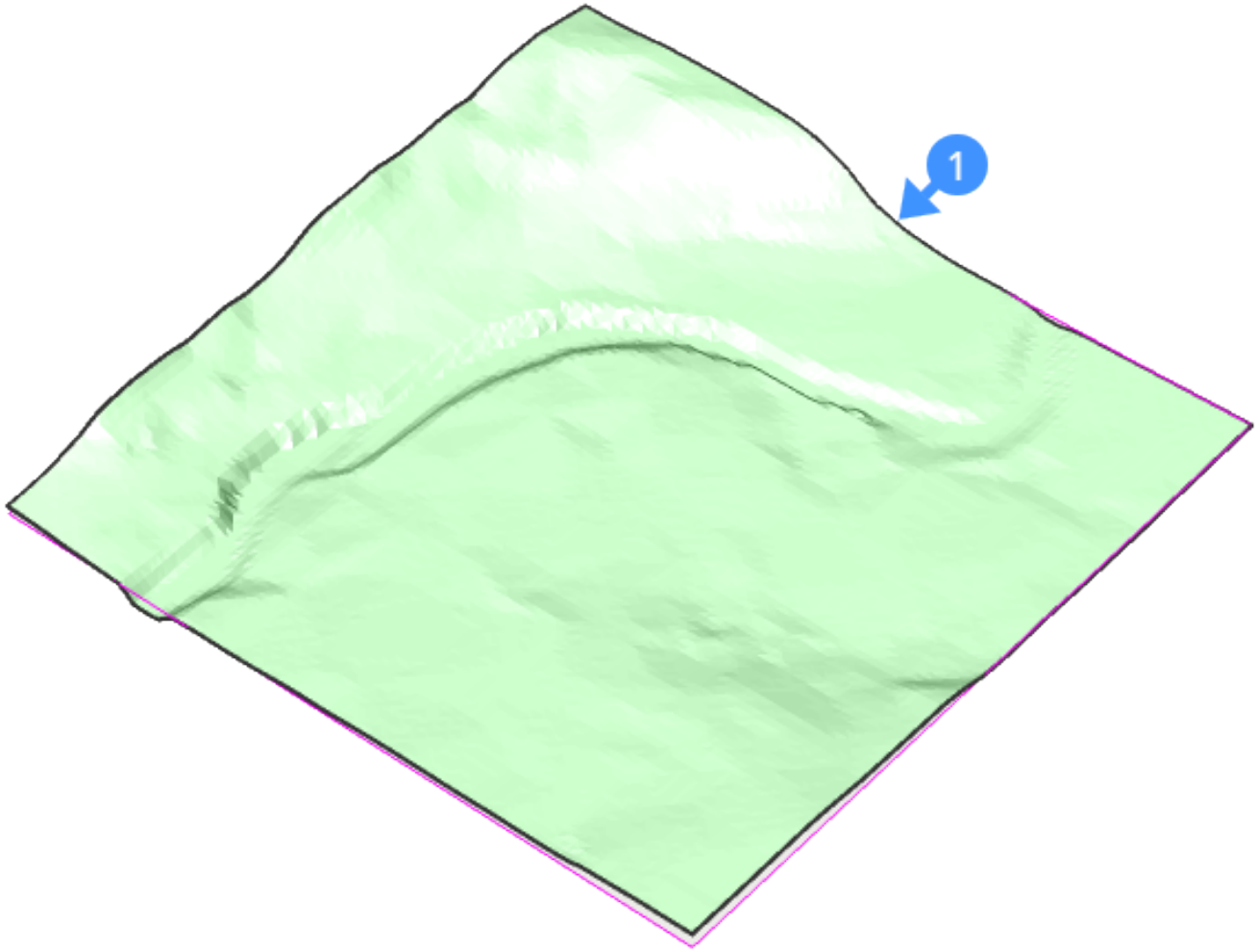


Figure 1.818

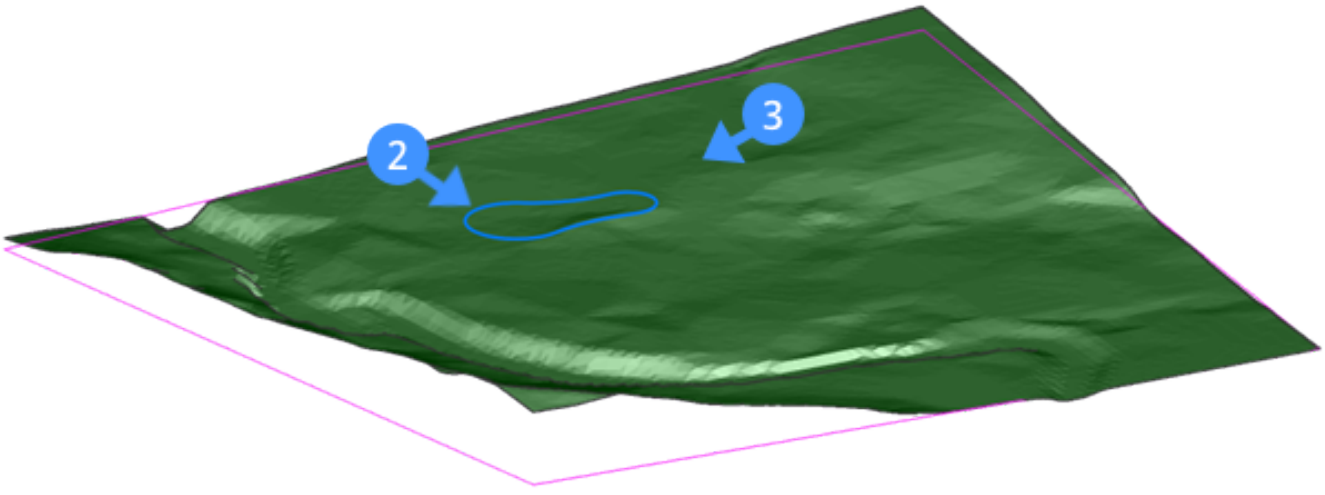
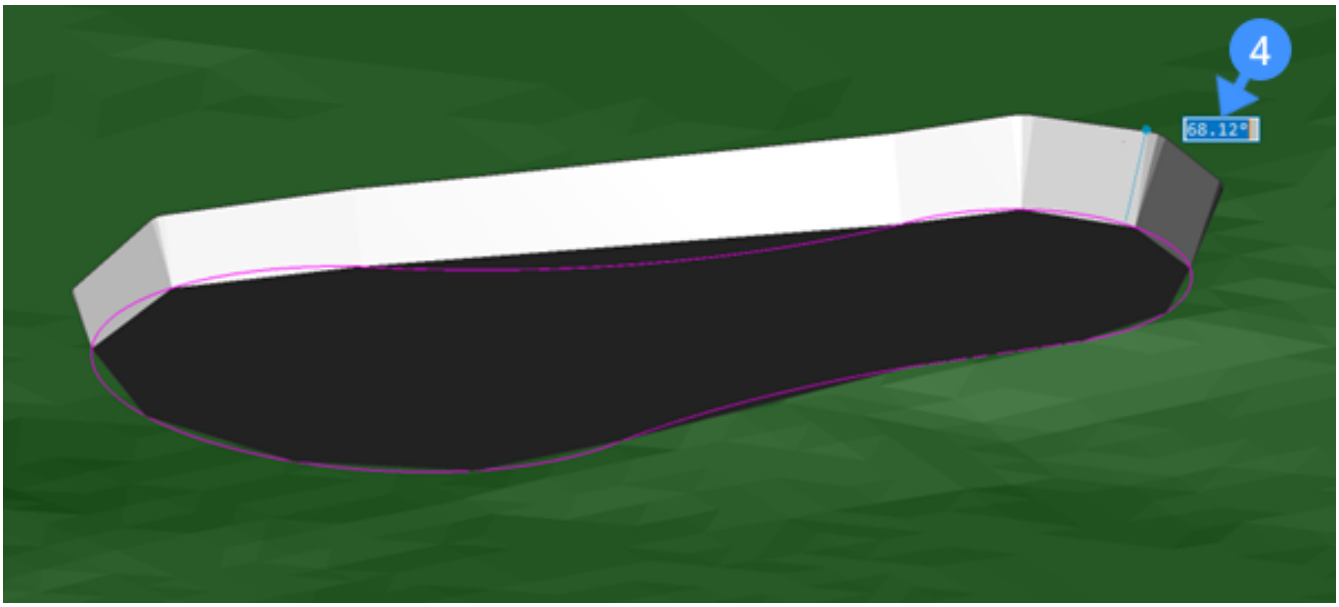


Figure 1.819



Displaying the properties of a TIN surface:

1. Select the graded surface.

The properties of this Surface is displayed in the**Properties**panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

Grading type	Demonstrates which projection method is used to grade the surface e.g. Slope to surface or Offset-slope.
Target surface handle	Displays the handle number of the target surface.
Cut slope	Displays the cut slope.
Fill slope	Displays the fill slope.
Side	Controls on which side of input entity grading is drawn. Left or right. To change the side of the grading, click its field,press the arrow button and choose the side in the context menu.
Infill	The internal area of entities is filled if this option is turned on. Valid for closed entities only (circle, closed polyline).

Figure 1.820

Grading	
Grading type	Slope to surface
Target surface handle	148
Cut slope	45
Fill slope	45
Side	Left
Infill	On
Boundary	Off
Fill volume	0 m³
Cut volume	21757.00 m³
Net volume	-21757.00 m³

Boundary	Controls the boundary of the grading. To turn the boundary on/off, click its field, and press the arrow button and choose On/Off in the context menu. When the grading boundary is turned on, that boundary is added as a 'hide' boundary to the target surface.
Fill volume	Displays the fill volume for a body/shape limited with target surface and graded surface.
Cut volume	Displays the cut volume for a body/shape limited with target surface and graded surface.
Net volume	Displays the net volume for a body/shape limited with target surface and graded surface. Note that only 'Slope to surface' grading type is valid.

The following images show when the grading boundary of the pond is off (5) and on (6). Based on the grading boundary, the net volume for the body of the pond is calculated and can be found in the properties panel (7). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/PondBoundaryoff.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/PondBoundaryon.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/NetvolumePond.png})

Figure 1.821

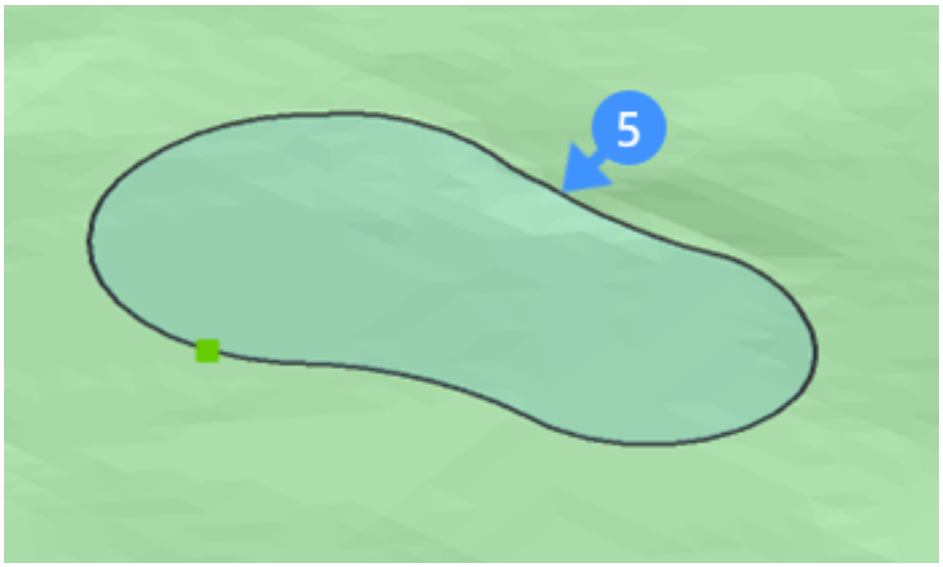


Figure 1.822



Figure 1.823

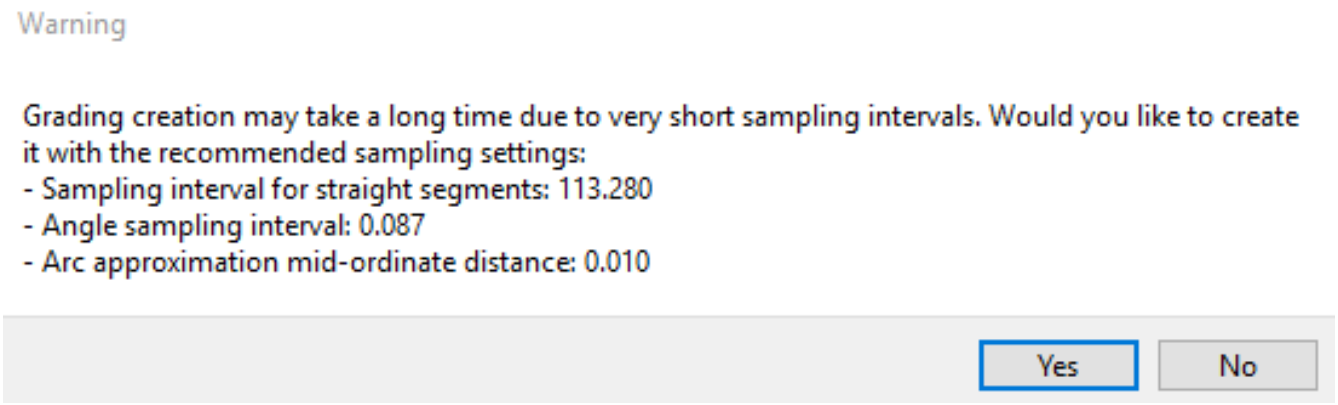
Grading	
Grading type	Slope to surface
Target surface handle	148
Cut slope	45
Fill slope	45
Side	Left
Infill	On
Boundary	Off
Fill volume	0 m³
Cut volume	21757.00 m³
Net volume	-21757.00 m³

7

Warning message while grading the surface:

If the grading process takes a long time to complete, BricsCAD displays a warning message. The warning message suggests the possible/ appropriate settings to grade the surface. The following image depicts the warning message when the 'Sampling interval for straight segments' parameter is too short, and the 'Arc tesselation mid-ordinate distance' parameter does not match with the drawing unit system to create the grading. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/GradingWarningMessage.png}) Press Yes to accept the suggested settings.

Figure 1.824



Selecting polylines with a large number of vertices to grade the surface

Grading jig (preview) for polylines with a large number of vertices takes a long time to grade the surface. In such cases, grading parameters, such as side, fill slope, and cut slope display in the command line to give you the ability to define these parameters manually.

1.18.10 Site Edit

Command: SiteEdit (6.27.43) **Overview:**In BricsCAD, points, breaklines, and boundaries on a triangulated irregular network (TIN) surface can be added or removed using the Site Edit command. The following entities represent features on a TIN surface.

- Points contain x, y coordinates, and z values. All points are used to make a connection with the two nearest neighbors to create triangles. The surface triangulation is basedon the Delaunay algorithm, which ensures that no point is inside the circumcircle of any triangle.
- Breaklines represent linear infrastructure features like curbs, retaining walls, etc. These lines also define the edges of triangles. Breaklines can be created from linear entities such as line, polyline, arc, circle.
- Boundaries are the features to define bounded 3D surfaces. The surface boundaries can be created with closed polylines. Defining outer boundaries in a large 3D surface improve the performance of a site plan.

Tutorial: How to edit a Site Click here (2.3.18) to watch

About

The **Site Edit** command adds or removes points, breaklines, boundaries from an existing TIN surface. **BIM Toolbar:**  **Quad:**  Keyboard: **siteedit** Menu:**BIM | Add to Site, Remove from Site** Toolbar: **Add to Site, Remove from Site** Quad Tab: **Model | Add boundaries, Add breaklines, Add points, Remove boundaries, Remove breaklines, and Remove points** Ribbon Tab | Panel:**Data | Site Design | Add to Site, Remove from Site**

Procedure: Adding points to a Toposurface

1. Open a TIN surface with surface points.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/TINSurfacewithPoints.png **Figure 1, the TIN surface with mass points (1)**

2. Select the TIN surface, and click the Add to Site toolin the BIM Toolbar.

Orlaunch the SITEEDIT command in the command line **You are prompted: Pick points to add to the site or [Add Points/Add Breaklines/Add bOundaries/Remove Points/Remove Breaklines/Remove bOundaries]:**

3. Click on the area to add points to the surface.

You are prompted: Enter elevation <*value>: *value= is calculated as an interpolated elevation for selected point inside triangle (2, 3,4). Note:"Enter elevation" displays in the command line when the Entity Snap Tracking (ESNAP) mode is off. If ESNAP is on you can check the elevation of a point in the cursor coordinate. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/TINSurfaceBeforeAddPoints.png})

Figure 1.825

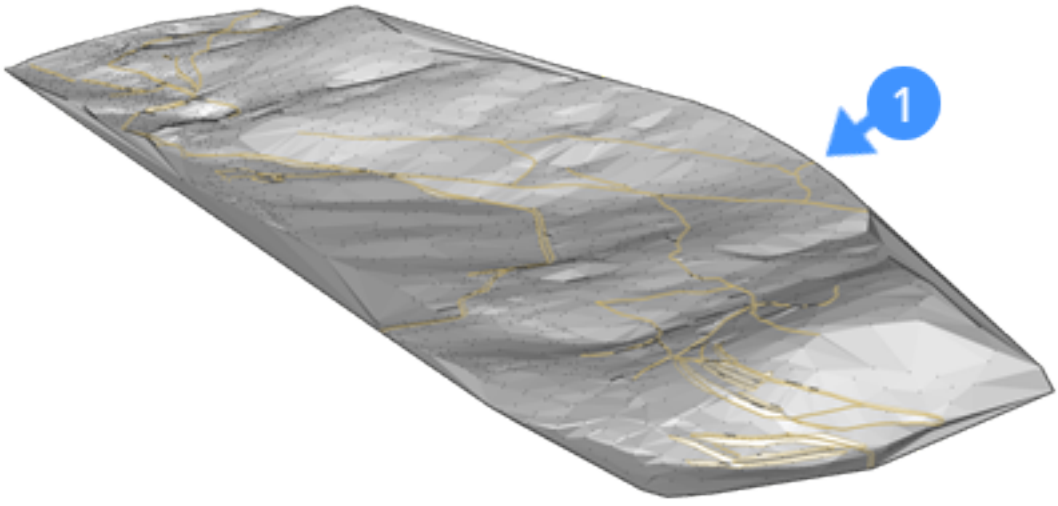
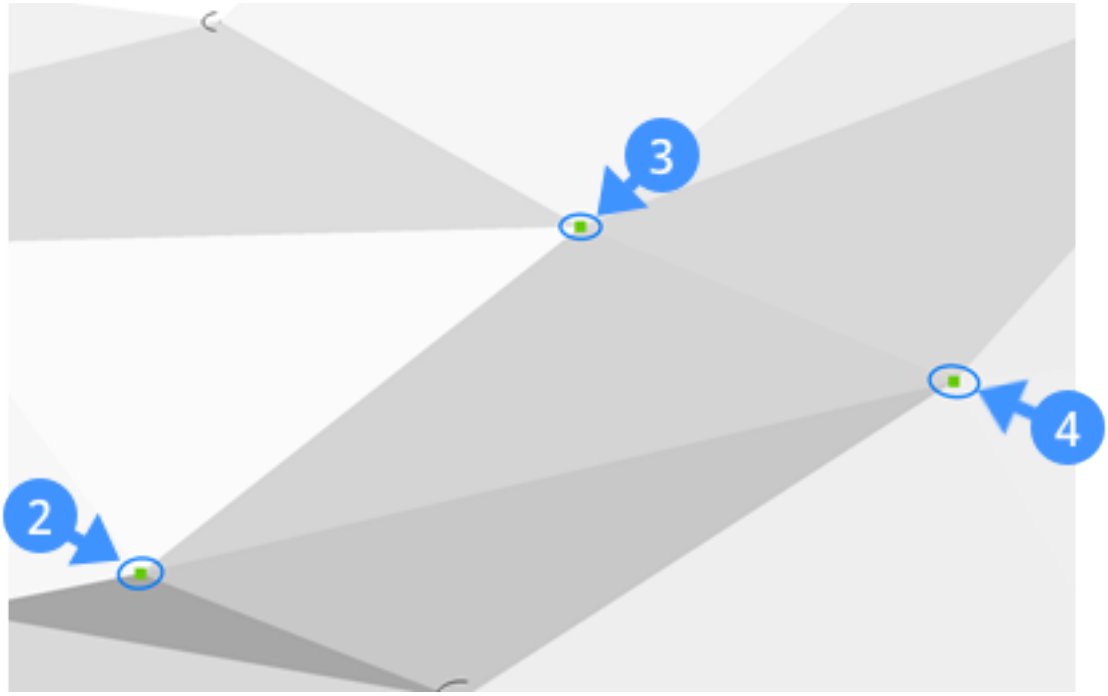


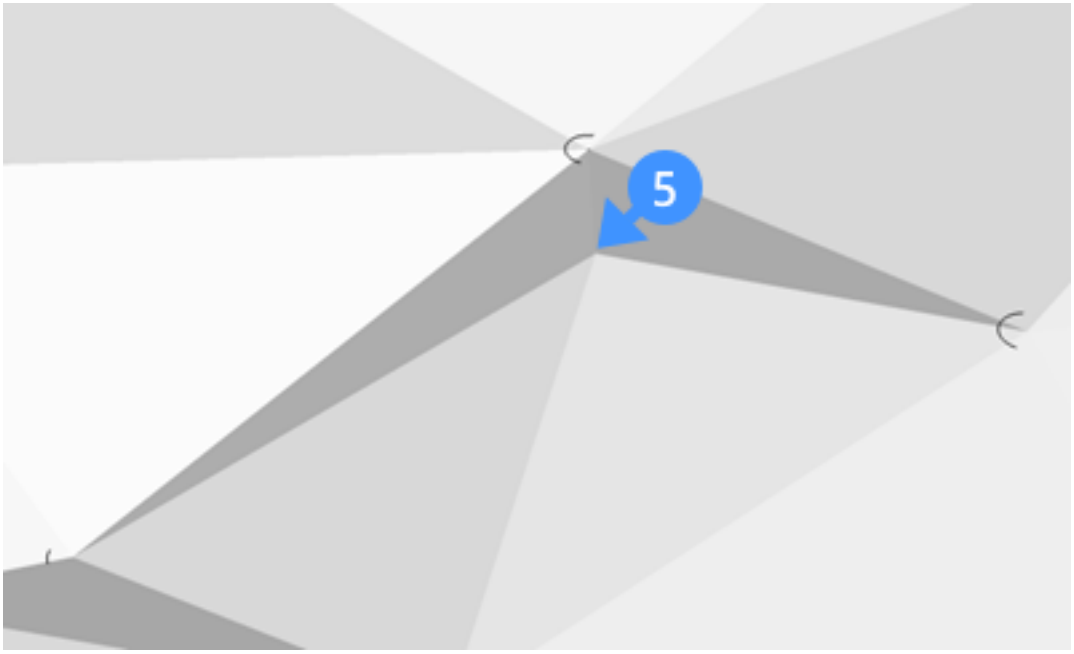
Figure 1.826



4. To specify the elevation of the newly added point, type an elevation value in the command line, and press Enter.

You are prompted:Pick point: Keep on adding points on the surface, or press Enter to exit the command. The new surface point (5) is added from entities, such as polyline vertices. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

Figure 1.827



Procedure: Adding breaklines to a Toposurface :

1. Open a TIN surface with breaklines.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/TINSurfacewithBreaklines.
Figure 3, the TIN surface with breaklines (6)

2. Select the TIN surface, and click the Add to Site toolin the BIM Toolbar.

Orlaunch the SITEEDIT command in the command line **You are prompted: Pick points to add to the site or [Add Points/Add Breaklines/Add bOundaries/Remove Points/Remove Breaklines/Remove bOundaries]:**

3. Select the Add Breaklines option or type AB in the command line.

**You are prompted:Select entities: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtec
Figure 4, the retaining wall breaklines before adding breaklines to the surface (7).**

4. Select the breaklines you want to modify the surface according to the edges of these linear features, and press Enter.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/TINSurfaceAfterAddBreakl
Figure 5, the retaining wall breaklines after adding boundaries to the surface (8). The TIN surface is modified according to the edge of retaining wall breaklines. Note: If the breakline is moved, the surface will be updated automatically.

Figure 1.828

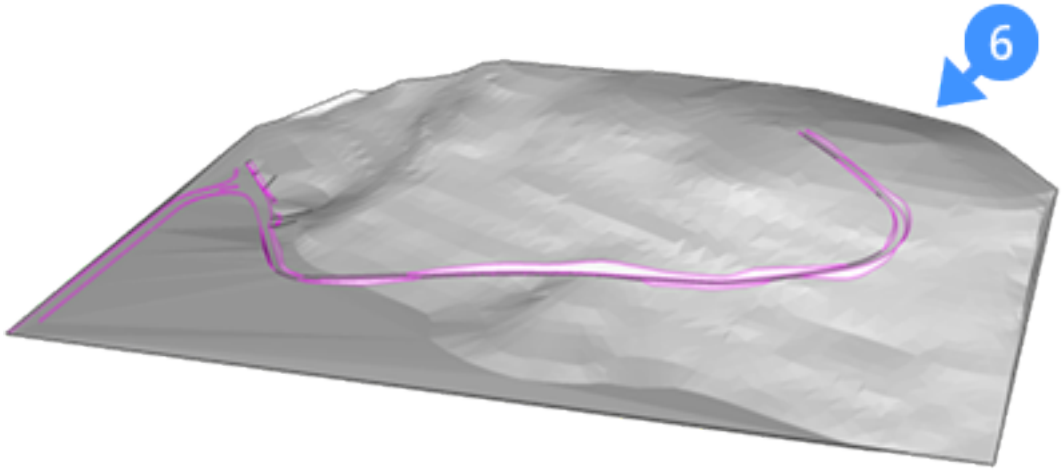


Figure 1.829

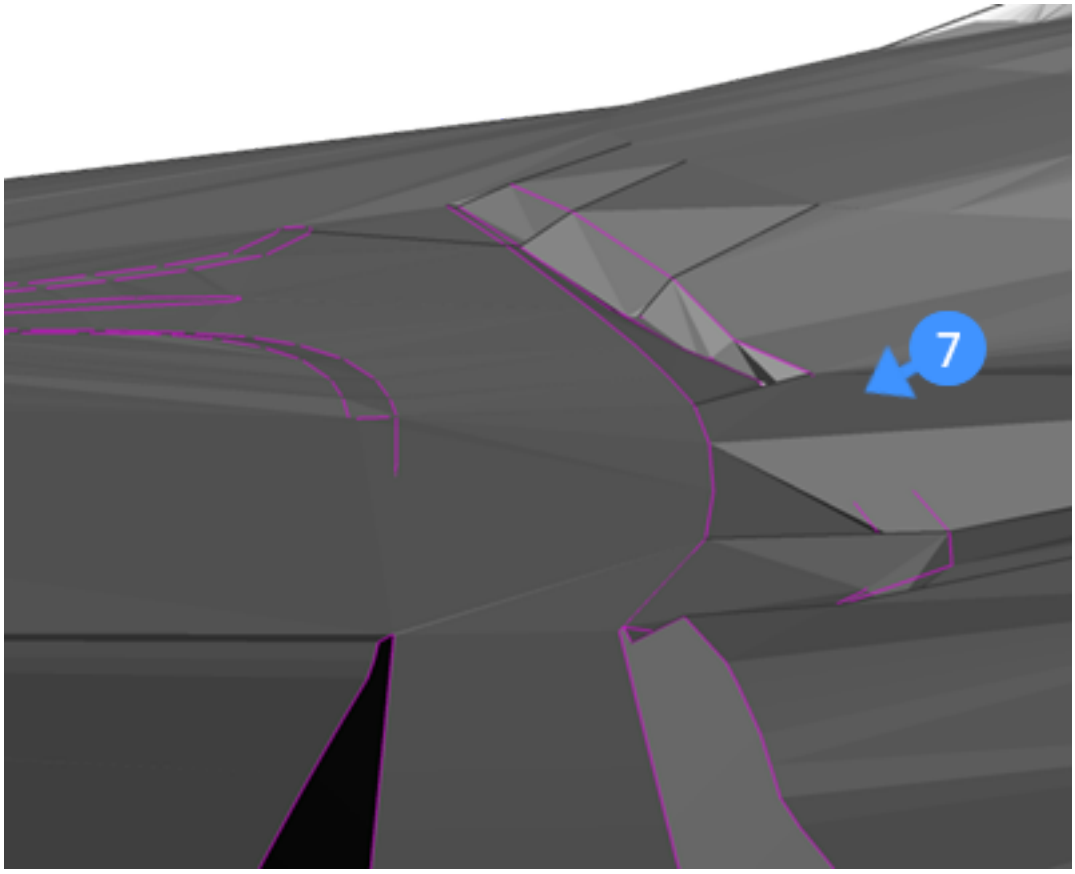
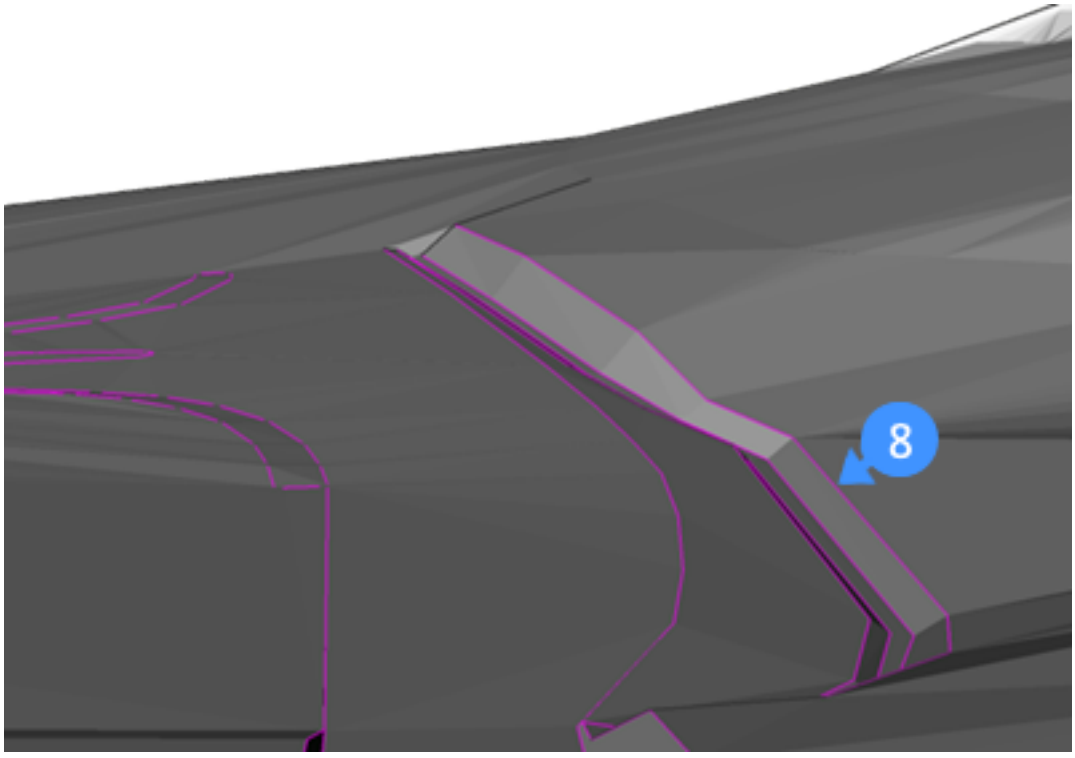


Figure 1.830



Adding boundaries to a Toposurface

1. Open a TIN surface you want to create a surface boundary.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/TINSurfaceBoundaries.png
Figure 6, a sample TIN surface (9)

2. Create a closed boundary on the location you want to clip the surface.

Note: Make sure that Entity Snap Tracking (ESNAP) is active. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
Figure 7, the closed boundary (10) on a TIN surface

3. Select the TIN surface, and click the Add to Site toolin the BIM Toolbar.

Orlaunch the SITEEDIT command in the command line **You are prompted: Pick points to add to the site or [Add Points/Add Breaklines/Add bOundaries/Remove Points/Remove Breaklines/Remove bOundaries]:**

4. Select the Add Boundaries option or type AO in the command line.

You are prompted:Select entities for boundary of type: [Outer/Show/Hide] <Outer>: Click on the closed boundary to use in the surface boundary definition, and press Enter. By default, the outer option is selected, which allows to create an outer surface boundary (11). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modeling
Optional Select the bounding entity (12)(e.g. polyline) to resize the surface boundary. Moving the grip (13) of the entity changes the surface dynamically (14). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Model

Figure 1.831

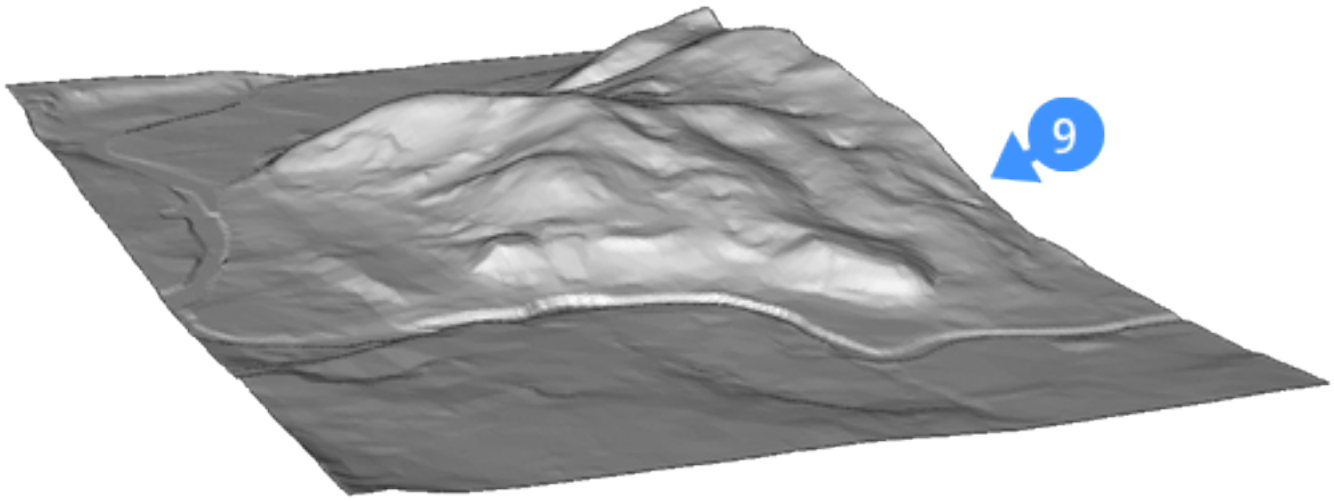


Figure 1.832

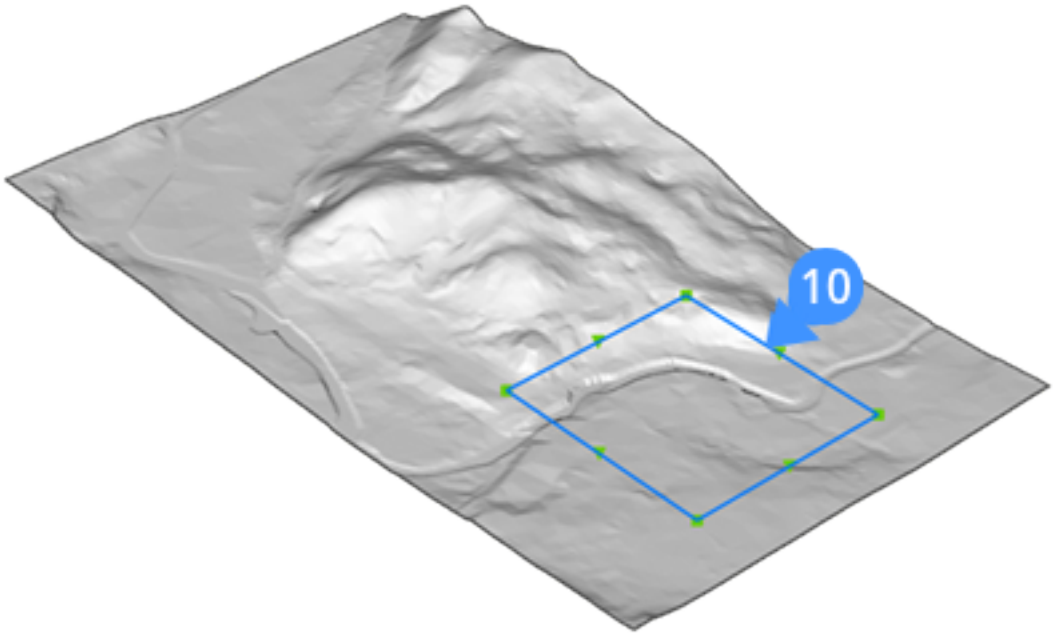
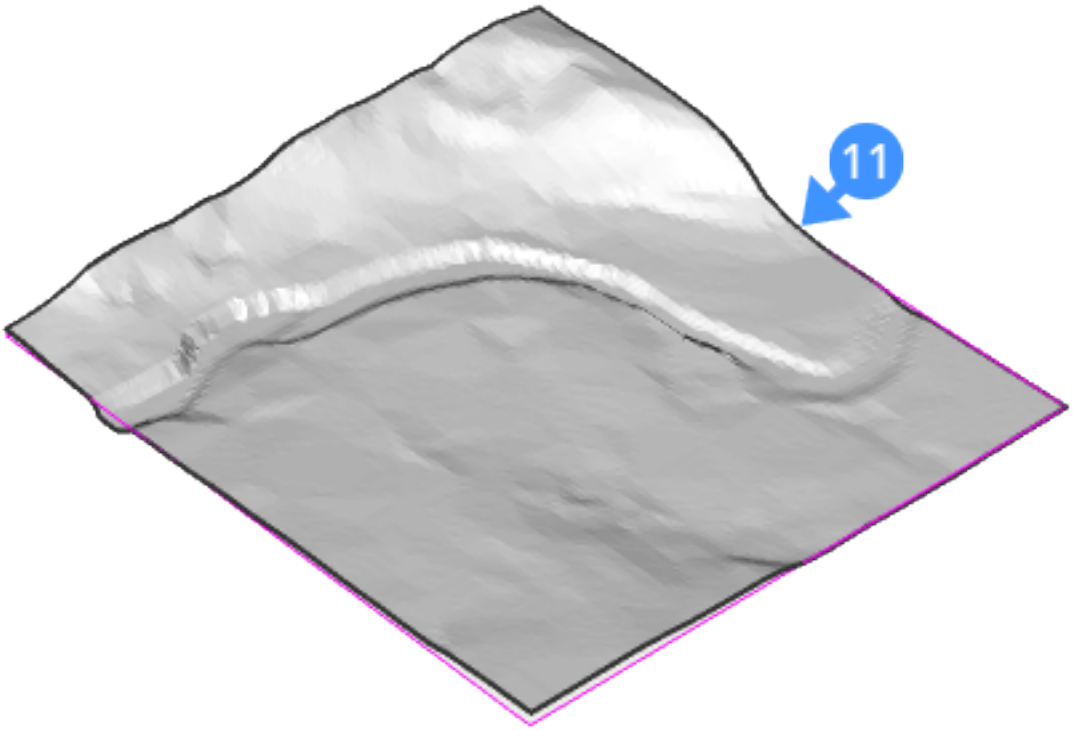


Figure 1.833



Optional Select the hide option to exclude a part of the surface from the TIN when calculating the total area. **Optional** Select the show option to reveal the hidden surface in the TIN. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he **Figure 8, the hidden part according to the closed boundary (15) is shown, the surface of the hidden part revealed (16) using the show option.**

Figure 1.834

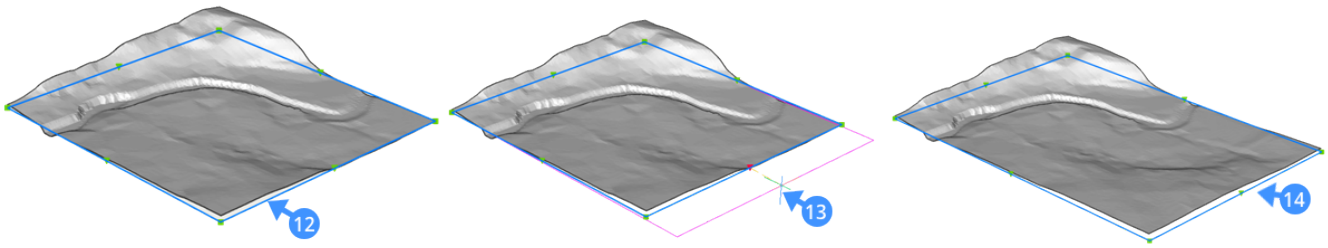
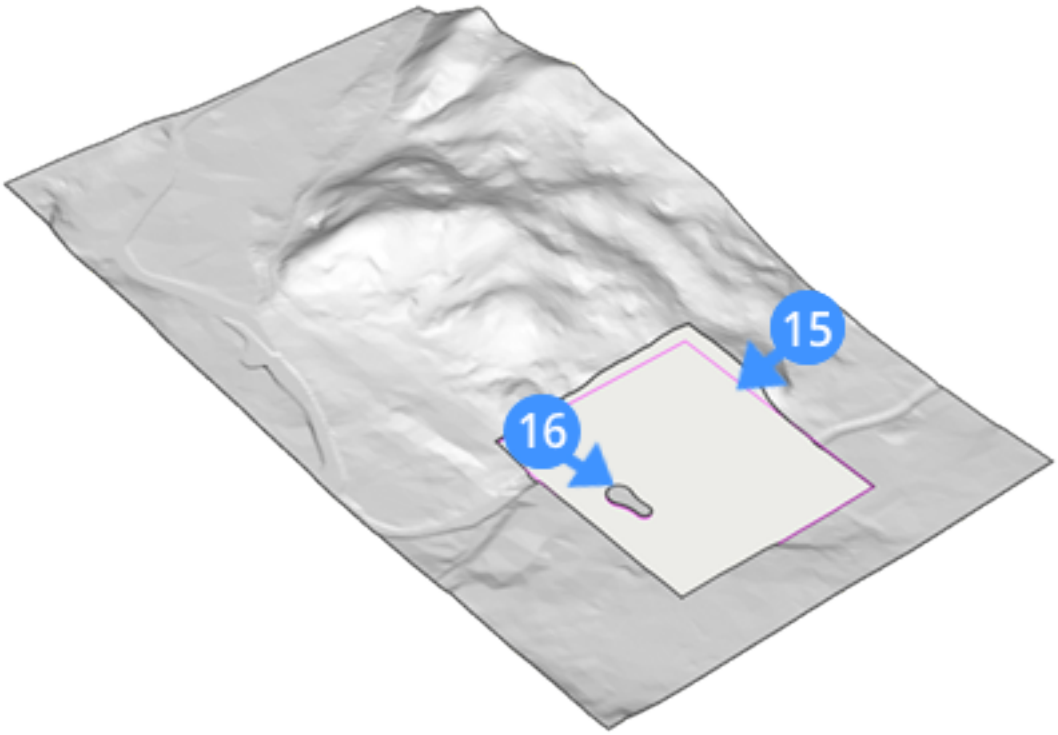


Figure 1.835



1.19 Parametric Modeling

1.19.1 Parameters Manager

Commands: PARAMETERSPANELOPEN (6.24.8), PARAMETERSPANELCLOSE (6.8.30). The Parameters Manager (a.k.a. Parameters panel) allows you to browse and manage global parameters of your model in tabular form. Every row in this table corresponds to one global parameter, while every column represents a particular property of a parameter - name, expression, value. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10D
With Parameters Manager you can:

- Create a new parameter
- Delete an existing parameter
- Click a parameter name field and change it
- Click a parameter expression field and change it
- Search parameter by name
- Sort all parameters by name, expression, and value
- Hide all parameters except ones, which drive the selected parameter
- Hide all parameters except ones, which are driven by the selected parameter
- Animate parameter value

To open the Parameters Manager do one of the following:

- Choose **Parameters Manager** in the **Components**menu
- Place the cursor on a toolbar, then right click and choose **Parameters**in the context menu.
- CallPARAMETERSPANELOPEN command

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/

Figure 1.836

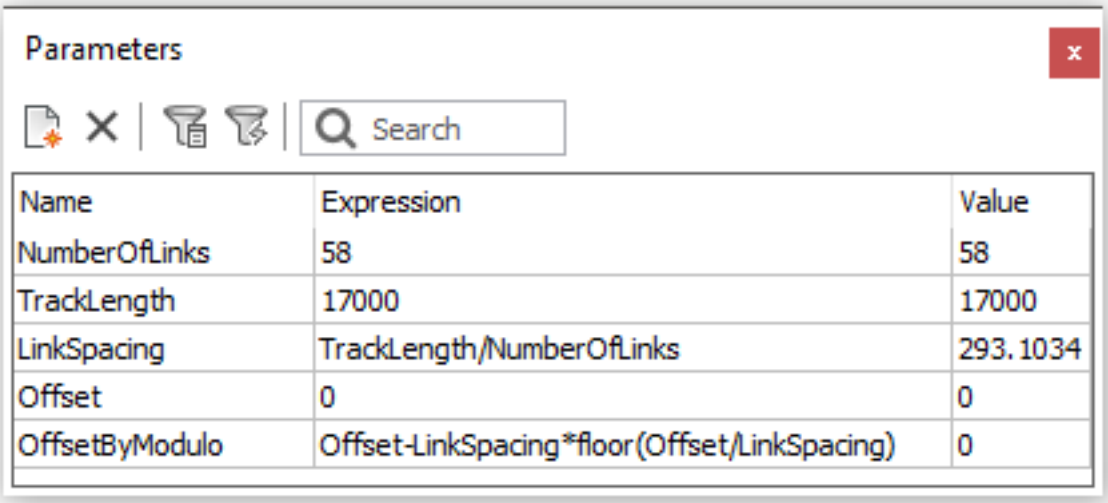
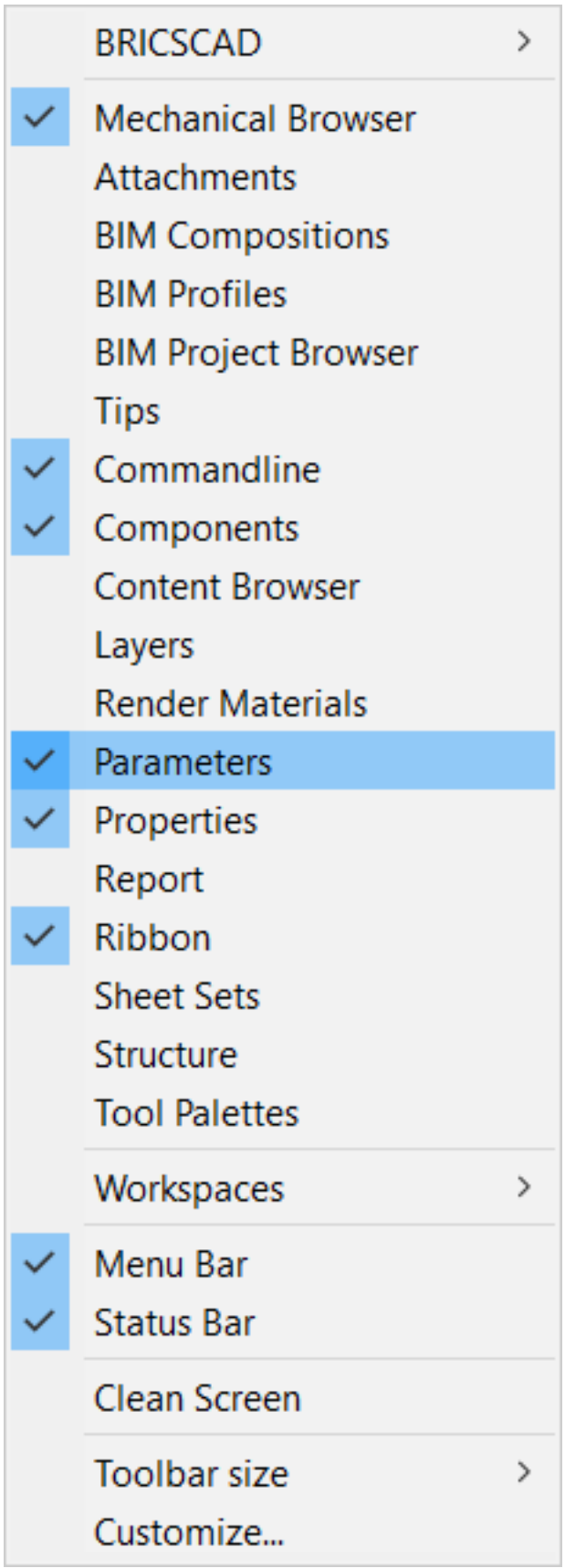

Parameters
Search
Name
Expression
Value
NumberOfLinks
58
58
TrackLength
17000
17000
LinkSpacing
TrackLength/NumberOfLinks
293.1034
Offset
0
0
OffsetByModulo
Offset-LinkSpacing*floor(Offset/LinkSpacing)
0

Figure 1.837


BRICSCAD
Mechanical Browser
Attachments
BIM Compositions
BIM Profiles
BIM Project Browser
Tips
Commandline
Components
Content Browser
Layers
Render Materials
Parameters
Properties
Report
Ribbon
Sheet Sets
Structure
Tool Palettes
Workspaces
Menu Bar
Status Bar
Clean Screen
Toolbar size
Customize...

1.19.2 Parametric Modeling Overview

BricsCAD provides a rich toolset to control 2D and 3D geometric model via parameters of different kinds, including:

- Global parameters, which includes parameters of 2D/3D dimensional constraints, user-defined parameters, and configuration parameters of design table,
- Local parameters, which includes parameters of arrays and constrained entities inside arrays, and parameters of inserts of parametric components.

All parameters and their current values are displayed under different sections in the Mechanical Browser (3.2.2). Parameter values can be linked together via expressions. When the value of a parameter is changed, the geometric model is updated automatically. This toolset allows BricsCAD users to add parametric behavior to any geometry and easily explore design variants.

NOTE	In BricsCAD there are no parent-child dependencies between geometric elements: for example, if you change a sketch used to create an extruded 3d solid, the solid is not changed accordingly. But you can always create any kind of dependencies using parameters and expressions.

Global and local parameters

Commands: -PARAMETERS (6.24.5), MECHANICALBROWSEROPEN (6.8.29), CLEANUNUSEDVARIABLES (6.9.23)
Parameters in BricsCAD can be global and local. Local parameters are attached to a particular entity: an associative array or a mechanical component. Global parameters are not attached to a particular entity; they are subdivided into four groups:

- parameters of 2D dimensional constraints,
- parameters of 3D dimensional constraints,
- user-defined parameters,
- configuration parameters of design tables.

To learn how to create 2D dimensional constraints, please read Using Dimensional Constraints (1.15.6). To learn how to create 3D dimensional constraints, see3D Constraints (1.19.4). To create a user-defined parameter, follow the procedure described in Creating a New Parameter (1.15.6). To create a configuration parameter for a design table, see Creating a design table to drive Parametric Blocks parameters (1.18.7).

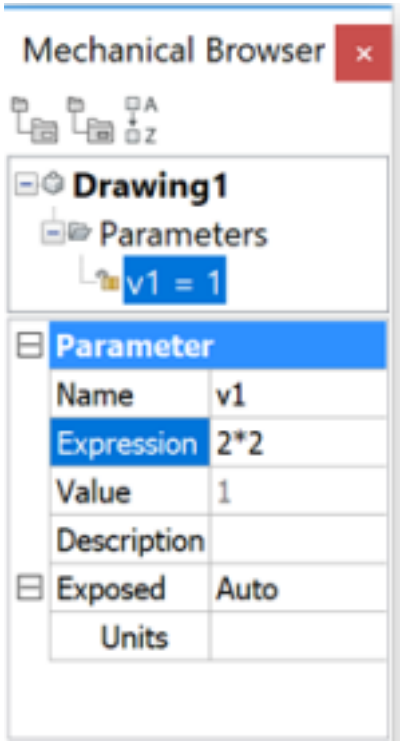
Setting an expression to a parameter

You can set an expression to any parameter, being it global or local. The simplest expression is just a number of a name of a global parameter (local parameters cannot be referenced by names in expressions). A more complex expression can also include standard operators and functions (see below). To set an expression to a parameter, click on the parameter name in the Mechanical Browser (3.2.2), and fill out the **Expression** field: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/ParameterExpress

NOTE	You can also set an expression to a parameter in command line using the -PARAMETERS (6.24.5) command.

You can use operators (1.15.6) and functions (1.15.6) to compose an expression.

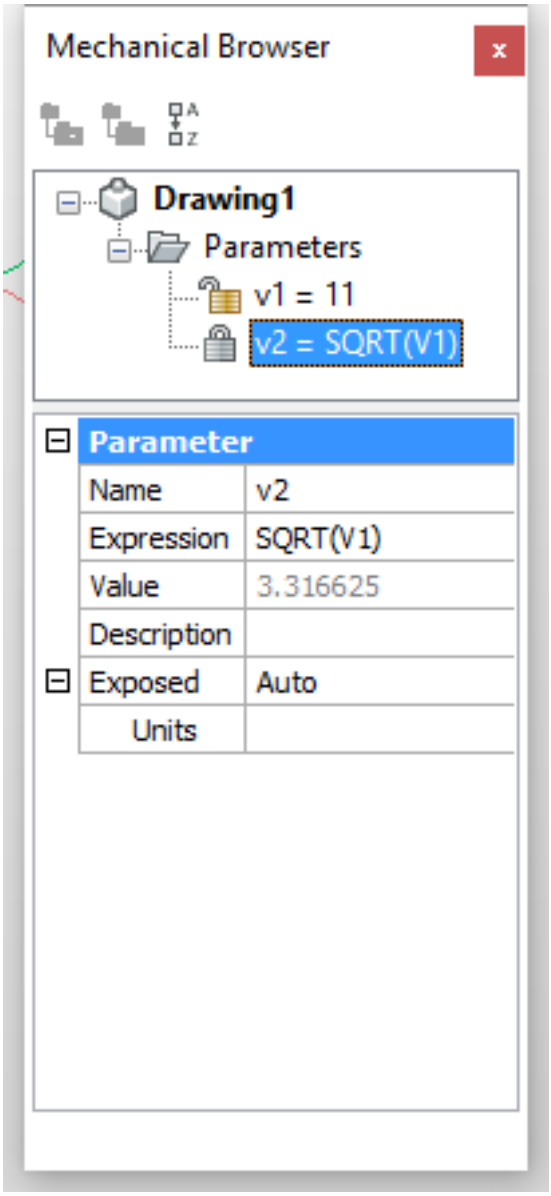
Figure 1.838



Driving and driven parameters

In expressions, you can use not only numbers, but also the names of global parameters. Such expressions are called non-constant. If a non-constant expression is set to a parameter, its icon, an open padlock ( ) in the Mechanical Browser (3.2.2)changes to the closed padlock ( ): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/U
This means that the parameter becomes driven by other (driving) parameters: its value will be automatically recalculated when the values of the driving parameters will be changed. You can create a chain of parameters, where every next element is driven by the previous ones. The length of such chain is not limited. Obviously you cannot create a loop, where the next element in a chain drives one of the previous elements, because it can result in infinite evaluation of expressions in the loop. BricsCAD automatically detects such expressions and therefore does not accept expressions, which lead to a parametric loop.

Figure 1.839


Mechanical Browser
Drawing1
Parameters
v1 = 11
v2 = SQRT(V1)
Parameter
Name
v2
Expression
SQRT(V1)
Value
3.316625
Description
Exposed
Auto
Units

Parameters of Associative Arrays

Commands: ARRAY (6.4.16), ARRAYRECT (6.4.25), ARRAYPOLAR (6.4.23), ARRAYEDIT (6.4.20). Properties of associative arrays that can be controlled by an expression:

- Rectangular arrays: Number of columns, Column spacing, Number of rows, Row spacing, Row elevation increment, Number of levels, Level spacing, Axis angle.
- Polar arrays: Radius, Number of Items, Angle between items, Fill angle, Number of rows, Row spacing, Row elevation increment, Number of Levels, Level spacing.

To set an expression to any of these properties, select the corresponding array entity, either in the drawing area or directly in the Mechanical Browser (3.2.2) and enter the expression in the corresponding properties field in the Mechanical Browser (3.2.2) or in the Properties bar (1.12.4). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
If an associative array contains entities constrained with 2D or 3D dimensional constraints, then parameters of these constraints can also be evaluated with expressions. These nested parameters are listed in the Mechanical Browser (3.2.2), and if you select any of them, you can set an expression to it: (\vref {fig:C:/Dev/Task-2448/images/staticbricsy

Figure 1.840

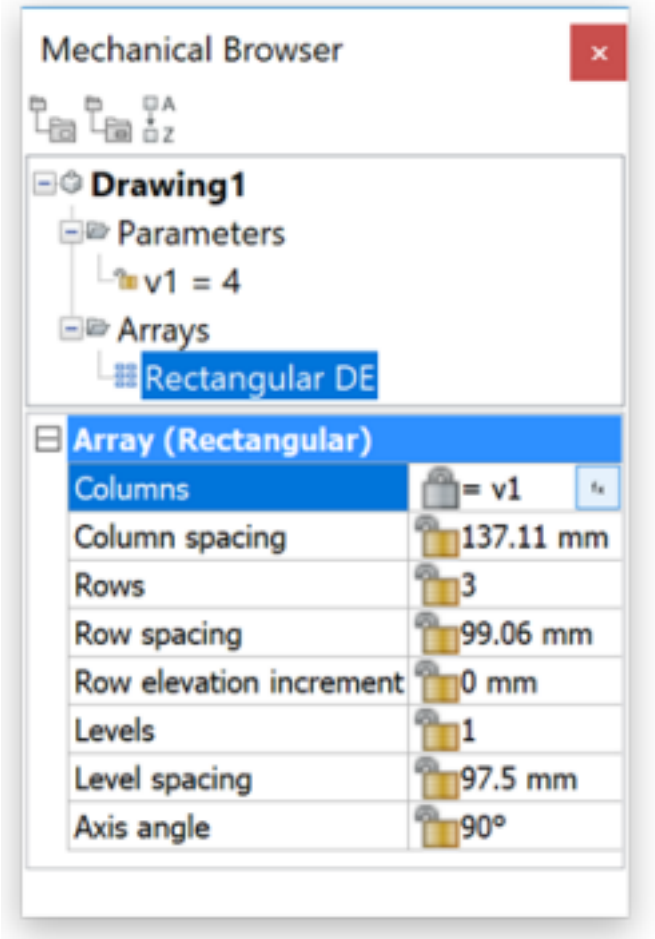
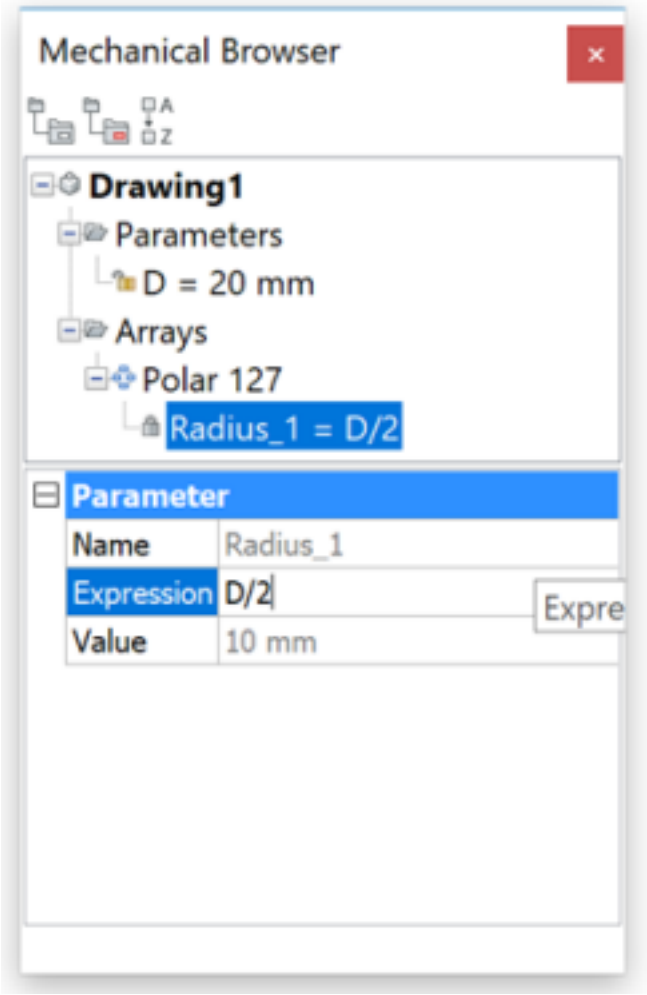

Mechanical Browser
Drawing1
Parameters
v1 = 4
Arrays
Rectangular DE
Array (Rectangular)
Columns
= v1
Column spacing
137.11 mm
Rows
3
Row spacing
99.06 mm
Row elevation increment
0 mm
Levels
1
Level spacing
97.5 mm
Axis angle
90°

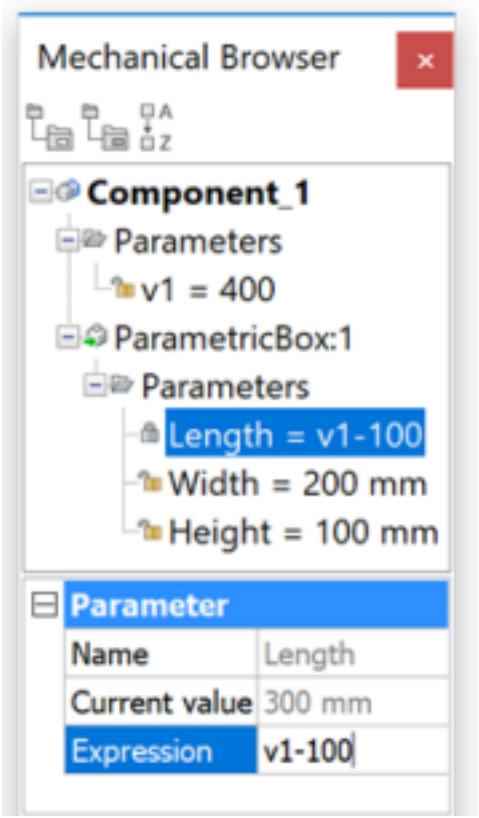
Figure 1.841


Mechanical Browser
Drawing1
Parameters
D = 20 mm
Arrays
Polar 127
Radius_1 = D/2
Parameter
Name
Radius_1
Expression
D/2
Expre
Value
10 mm

Parameters of Mechanical Components

Commands: -BMPARAMETERS (6.8.19), BMUPDATE (6.8.25), The BMUPDATEMODE (1.13.11) system variable controls whether external assembly components are always reloaded or only in case they are modified. If you insert a mechanical component or just a plain .dwg file into your drawing with BMINSERT (6.8.12) command, and the insert contains global exposed parameters (1.22.22), then such parameters become local in the current drawing. They are associated with the corresponding component insert and are visible in the Mechanical Browser (3.2.2) under the name of this insert. Click such parameter to edit: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG

Figure 1.842



Design Tables

Commands: DESIGNTABLE (6.10.26), -DESIGNTABLEEDIT (6.10.27) A design table allows BricsCAD users to evaluate several global parameters at once using the value of one configuration parameter. Design tables can be created from scratch in BricsCAD or imported from a .csv file (a comma-separated values text file), which can be created with any spreadsheet software or a text editor.

Creating a Design Table from a .csv file The expected format of the .csv file is: Configuration_Name,Parameter1, Parameter2, ... , ParameterN Configuration_1,Value1Parameter1,Value1Parameter2,...,Value1ParameterN Configuration_2,Value2Parameter1,Value2Parameter2,...,Value2ParameterN Configuration_3,Value3Parameter1,Value3Parameter2,... ... The first row lists the parameter names separated by a comma. The first name should not coincide with the name of any global parameter persistent in your drawing. It is called the configuration name.No spaces are allowed in the configuration name. Other names in the first row can (and usually do) coincide with the names of global parameters in your drawing. The next rows list possible values for these parameters, which can either be strings or numbers. Example: Size,Length,Width,Height Small,300,200,100 Medium, 450,300,150 Large,600,400,200 Importing such .csv file into BricsCAD using te DesignTable (6.10.26) command results in creating a special global parameter by the name of the first name in the first row in the .csv file (= Size in the above example), which is called the configuration parameter. It will drive the values of other parameters in the design table, which are then called configuration-driven parameters.

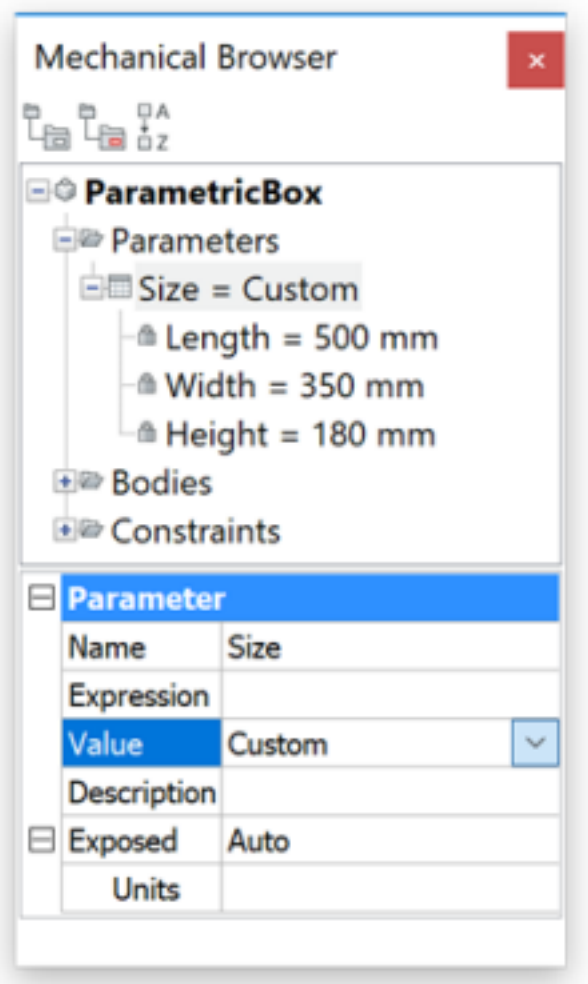
NOTE	BricsCAD does not keep the association with the .csv file, which was used to create a design table: if this file has been modified then the design table is not updated automatically. Use the -DESIGNTABLEEDIT (6.10.27) command to replace a design table using a new or modified .csv file.

Direct Lookup in a Design Table To evaluate a configuration parameter:

- Select it in the Mechanical Browser (3.2.2) and type the desired value in its **Expression** field.
- Select a value in the drop-down list of the **Value** field:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/
Changing the value of a configuration parameter will result in automatic changing the values of all configuration-driven parameters of the corresponding design table. The values of these parameters are taken from the row in the design table, of which the first element coincides with the current value of the configuration parameter.

Figure 1.843


Mechanical Browser
ParametricBox
Parameters
Size = Custom
Length = 500 mm
Width = 350 mm
Height = 180 mm
Bodies
Constraints
Parameter
Name
Size
Expression
Value
Custom
Description
Exposed
Auto
Units

Reverse Lookup in a Design Table Parameters, which are driven by a configuration parameter via a design table, can be evaluated independently. To do so, select the driven parameter and apply an expression to it. If the value of this expression matches one of the values for this parameter in the corresponding design table, then the configuration parameter and all driven parameters of the design table will be automatically set to the values specified in the corresponding row in the table. If several configurations are possible for a particular value of a driven parameter, BricsCAD selects the one, which is closest to the current values of other parameters. If you enter an expression of which the result is not available in the design table, or if you enter expressions for two or more design table parameters, of which the values cannot be found in a single configuration (= the values are not found in the same row in the design table), then the values of the configuration parameter are changed to 'Custom', and the values of the other driven parameters are not changed. This case corresponds to a configuration, which does not exist in the design table.

Adding a New Configuration to a Design Table You can add any 'Custom' configuration to a design table under a particular name. To do so, right click the configuration parameter in the Mechanical Browser (3.2.2) and select **Save current configuration** in the context menu. The configuration will be added to the design tables 'Configuration_N', where 'Configuration' is the name of the configuration parameter, and 'N' is an index number, which is one more than the current number of configurations in the design table. If you want to save a configuration in the design table under a different name, launch the -DesignTableEdit (6.10.27) command.

Creating an Empty Design Table Using the procedure described above (adding a new configuration to a design table), it is possible to start your design with an empty design table, and then fill it with the desired values. To do so, use the 'Empty Design Table' tool, which asks you:

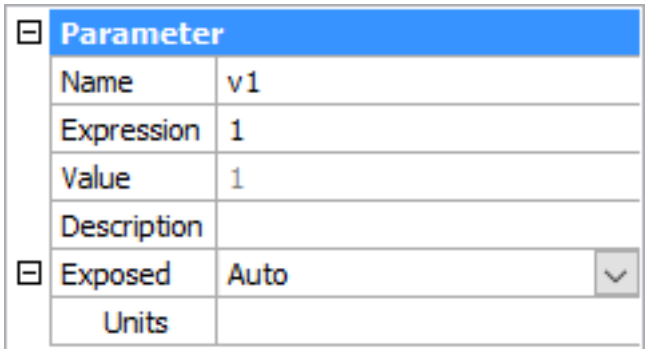
- The name of a new design table (which is also the name of the corresponding configuration parameter),
- The names of global parameters, which will be driven by this design table.

Using a Parametric Component with Design Tables You can expose both configuration parameters and configuration-driven parameters to be used in assemblies. If you then insert such a component into your drawing, its parameters will include the exposed parameters of design tables, for which drop-down lists of admissible values will be available. Both direct and reverse lookup actions are supported for such parameters.

Other Actions with Design Tables The -DESIGNTABLEEDIT (6.10.27) command allows to:

- Delete a design table.
- Replace a design table.
- Link a new parameter to the design table or unlink an existing one.
- Add new configurations, remove an existing one, or export a design table into a .csv file.

Figure 1.844



1.19.3 Parameters and their Properties in Mechanical Browser

All parameters such as; user-defined, 3D constraints, design tables, parameters of parametric components and arrays can be managed via Mechanical Browser (3.2.2). The following context menu items and properties are available:

- Add new parameter: Creates a new parameter, which will control the component (1.22.22).

Context Menu	
Parameter context menu	Right click a component parameter: Main component parameter context menu (left), subcomponents parameter context menu (right): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10 {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10 Delete: delete the selected parameter from file. Create design table: creating a design table (1.18.7) to drive parametric block parameters Animate: animate parameter value within some range. Link to parameter: link subcomponent parameter to the main level parameter. Collapse all: collapses the main component and all components and subcomponents. Expand all: expands the main component and all components and subcomponents.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/

- **Name:** Identifies the parameter. The parameter name can be used in expressions of other parameters in the same **component.**
- **Expression**: Type a value or an expression.
- **Value:** Displays the current value of the parameter.
- **Description**: Optional description of the parameter.
- **Exposed:** Defines whether the parameter is visible and can be modified when the component is inserted in an assembly. Select the field, then click the down-arrow button and choose an option. A new parameter gets the **Auto** option by default.
  - **Auto** = the parameter is exposed only if it does not depend on other parameters.
  - **ON** = the parameter is always exposed.
  - **OFF** = the parameter is never exposed**.**
- **Units:** Sets the dimension of the parameter: Linear, 2-dimensional or 3-dimensional. Select the field, then click the down-arrow button and choose an option.

Figure 1.845

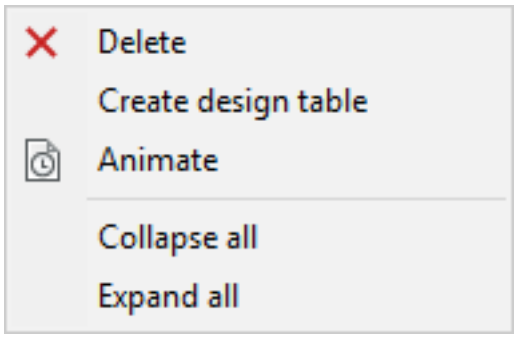
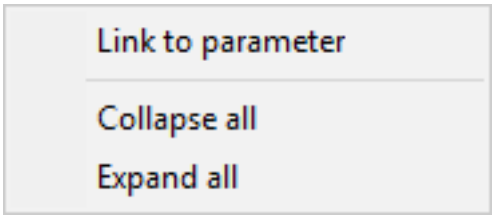


Figure 1.846



1.19.4 3D Constraints

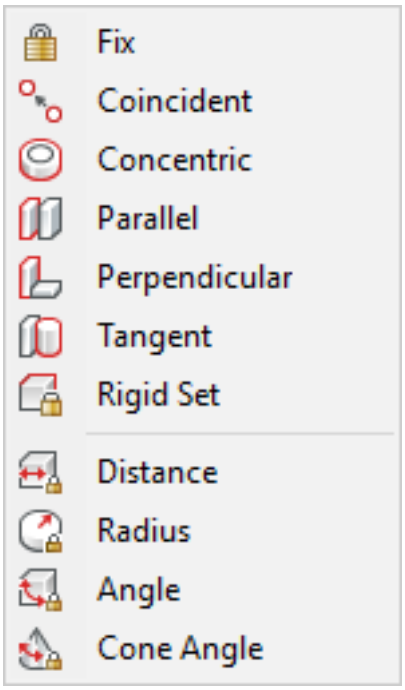
Commands: MechanicalBrowserOpen (6.8.29), DMUPDATE (6.12.33), DMFIX3D (6.12.15), DMCOINCIDENT3D (6.12.5), DMCONCENTRIC3D (6.12.6), DMPARALLEL3D (6.12.18), DMPERPENDICULAR3D (6.12.19), DMTANGENT3D (6.12.30), DMDISTANCE3D (6.12.12), DMRADIUS3D (6.12.21), DMANGLE3D (6.12.1)

Variational Direct Modeling

BricsCAD offers powerful tools for parametric direct solid and surface modeling by using geometric constraints solving (aka variational direct modeling). Solid and surface models can be modified by defining constraints between their elements (edges and faces), as soon as a constraint is added BricsCAD automatically modifies the model and these constraints are updated with subsequent modifications to the model. The key advantage of variational direct modeling is that all constraints are taken into account simultaneously and the model behavior does not depend on the constraints creation order. It allows you to parameterize any feature of the 3D model without worrying about the model creation history. Dimensional constraints control the dimensions of the model. When such a constraint is introduced or the value of its parameter is changed BricsCAD automatically updates the solid and surface geometry. For example, the dimensions of a box can be controlled by the parameters of three distance constraints applied to the opposite faces. Constraints can be created using the **3D Constraints** toolbar, the **Parametric/3D Constraints** menu or the **Constraints** command group in the Quad (1.12.5). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/ART/033DConstraintsMenu.png}) BricsCAD provides the following 3D constraints:

- Geometrical constraints:**Fix**( ),**Coincident**( ),**Concentric**( ),**Parallel**( ),**Perpendicular**( ),**Tangent**( ), and**Rigid Set**( ).
- Dimensional 3D constraints:**Distance**( ) ,**Radius**( ),**Angle**( ), **Cone Angle** ( ).

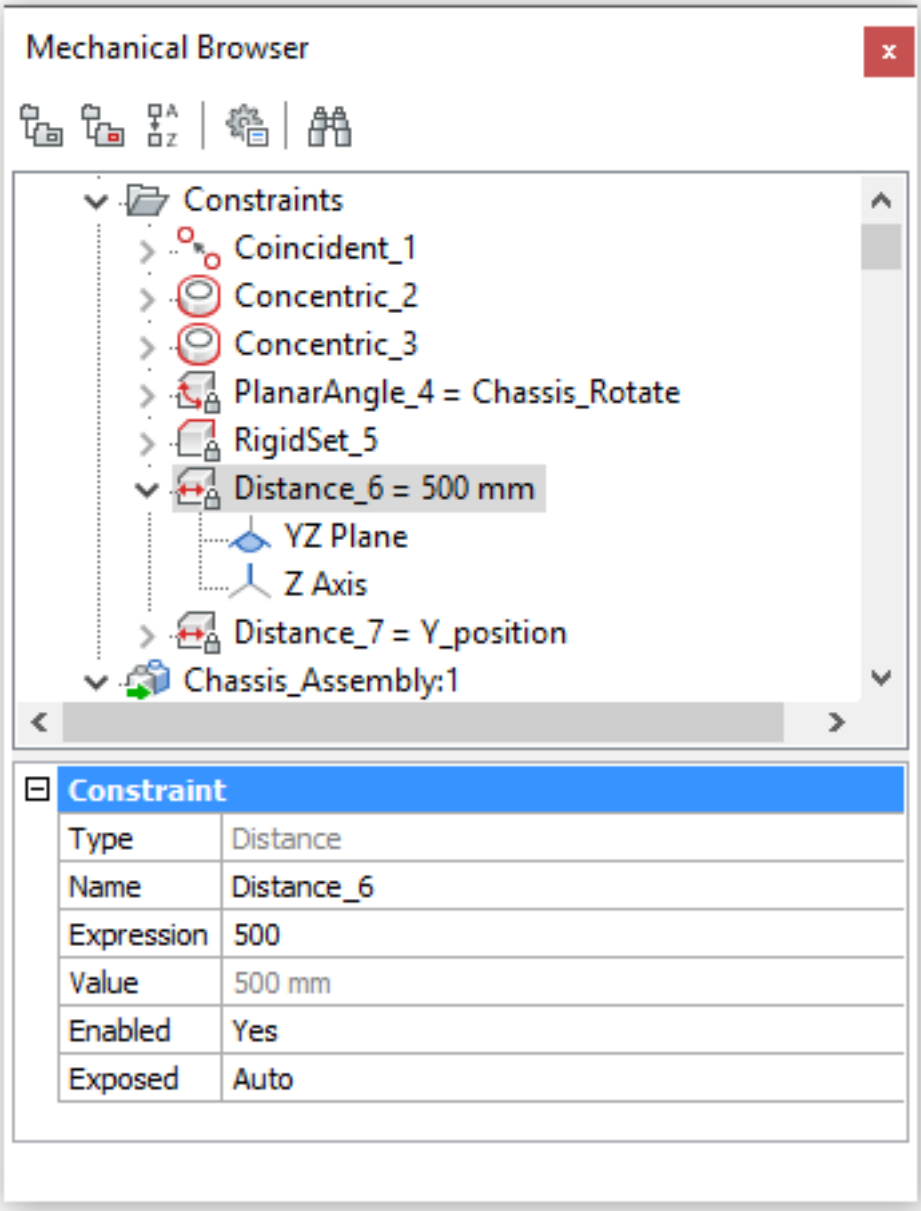
Figure 1.847



Mechanical Browser

The Mechanical Browser (3.2.2) ( ) navigates through all the constraints in the model edits dimensional constraints. If you select a constraint in the Mechanical Browser, you can check and edit its properties (1.19.5). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/ART/MechanicalBrowse

Figure 1.848


Mechanical Browser
Constraints
Coincident_1
Concentric_2
Concentric_3
PlanarAngle_4 = Chassis_Rotate
RigidSet_5
Distance_6 = 500 mm
YZ Plane
Z Axis
Distance_7 = Y_position
Chassis_Assembly:1
Constraint
Type Distance
Name Distance_6
Expression 500
Value 500 mm
Enabled Yes
Exposed Auto

3D Constraints types

The following entities are supported for constraints creation:

- Faces of 3D solids and surfaces of planar, cylindrical, spherical, toroidal and conical geometry.
- Linear and circular edges of 3D solids and surfaces.
- Lines, rayes, xlines, circles and arcs.
- Arbitrary curves for Coincident constraint with a point.
- Planes, axes and origin points of the world or a block coordinate system.

3D constraints have a number of properties which allow to control how BricsCAD treats them in various situations. For full description see 3D Constraints Properties page (1.19.5).

3D Constraints and Direct Modeling

3D Constraints are taken into account when direct modeling operations are applied. If there is a 3D constraint which fixes the placement of faces or edges this constraint will be preserved during Push/Pull, Move and Rotate operations. The dynamically presented result of the operation respects existing constraints as well. A 3D constraint can be temporary disabled or erased permanently using Mechanical Browser Constraints section. If some faces or edges disappear in a direct modeling or in a Boolean solid editing operation 3D constraints applied to such entities are removed. However, if there is an exact correspondence between the initial faces and the faces obtained, 3D constraints are automatically applied to new entities. Depending on the value of the DMRECOGNIZE (6.27.26) system variable, BricsCAD automatically recognizes geometrical relations between the surfaces of a solid and preserves them during direct modeling operations (see Design Intent Recognition (1.18.3)).

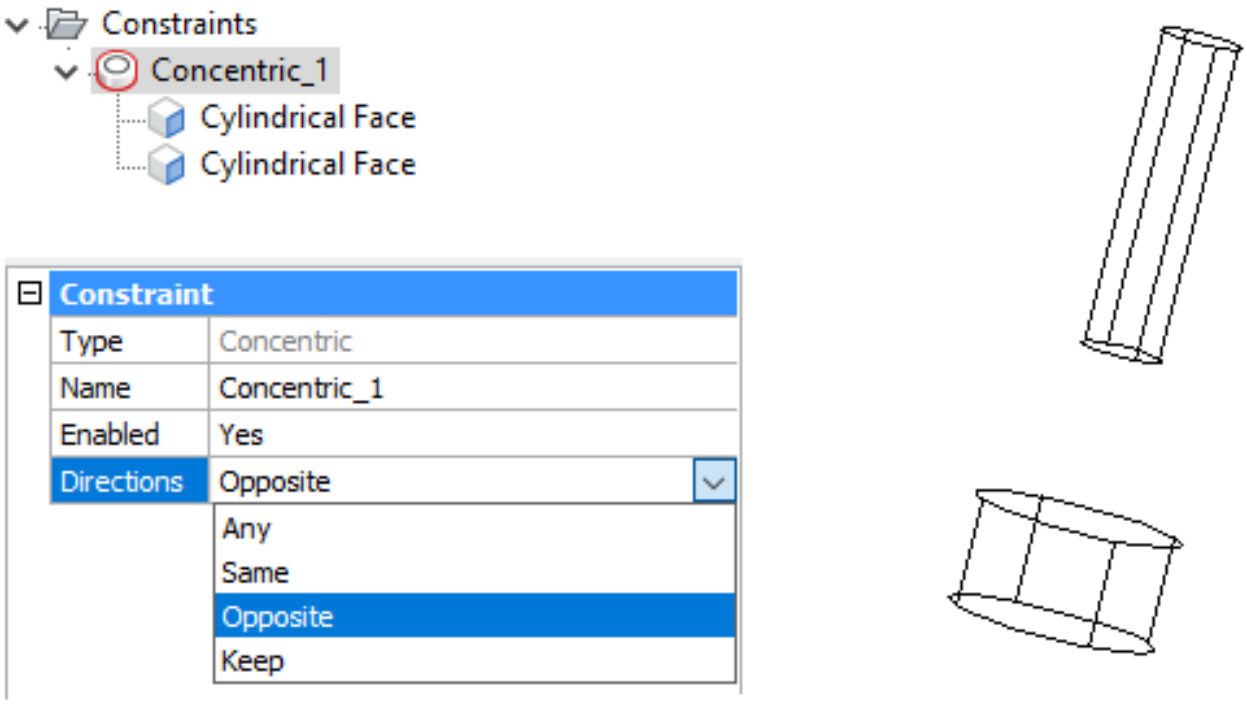
1.19.5 3D Constraints Properties

BricsCAD provides a set ofpropertiesfor 3D constraints which allow to control mutual placement of the constrained geometrical entities (or constraint arguments) upon 3D constraints recalculation. The properties may be assigned both to a constraint itself (**Directions**property) and/or to the constraints arguments, for each argument separately (**Placement**and**Use as**properties).

"Directions" property

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/ART/Directionspro
Directionsproperty specifies whether the directions of the constraint arguments are**Same**or**Opposite**. Changing from one to another results flips constrained entities with respect to each other.. Direction has sense for lines, planes, circles, cylinders, cones and torii. It has no sense for points and spheres. Direction of a plane is its normal vector. Direction of a circle is the normal vector of the circle's plane. Direction of a cylinder, cone, or torus is the direction of its axis. The direction of a planar face of a 3D solid is additionally adjusted to be external w.r.t. the 3D solid object. **Any**option means that the directions may change upon direct modelling operations or upon a 3D constraint parameter change. **Keep**option forces BricsCAD to keep the current directions. In other words, this option forces BricsCAD to analyze the mutual location of the arguments prior to 3D constraints solving and use either**Same**or**Opposite**(not both).

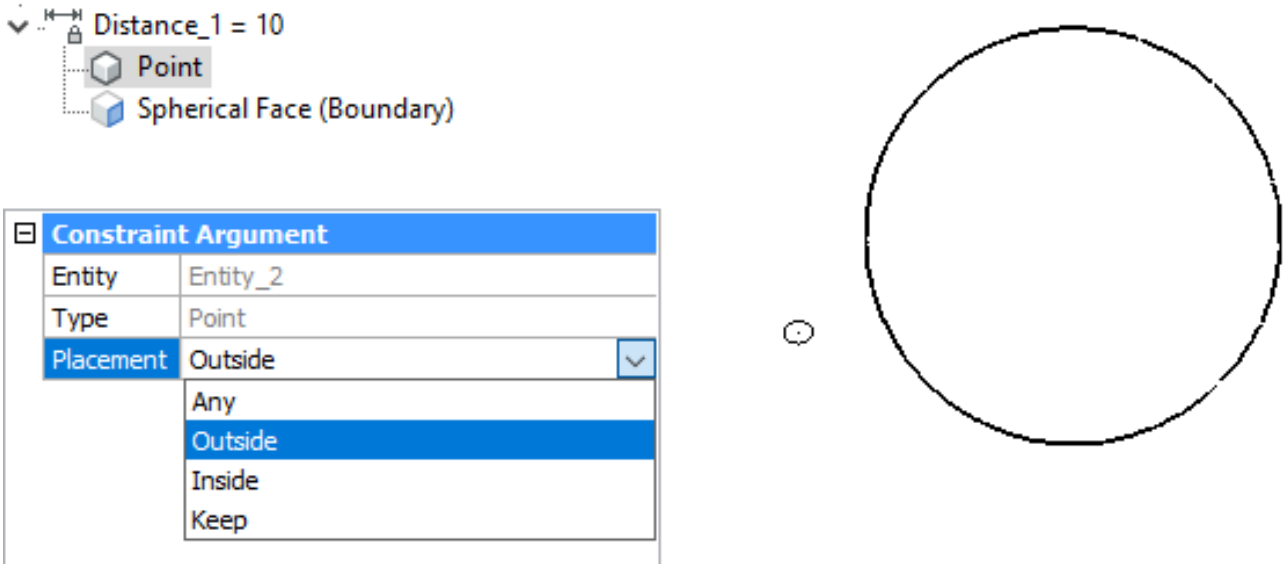
Figure 1.849



"Placement" property

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/ART/Placementpr
Placementproperty specifies how to place an argument with respect to**another**argument of the same constraint. For example, a point in a point-sphere distance may be placed either**Outside**or**Inside**of a sphere. Changing this property for a constraint argument moves the argument to the other side of another argument. The sides are defined for planes, spheres, cylinders, cones, and torii. And not defined for points, lines, and circles. **Any**option means that both**Outside**and**Inside**placement is suitable for user's intentions. **Keep**option forces BricsCAD to keep the current placement of the argument. In other words, this option forces BricsCAD to analyze the mutual location of the arguments prior to 3D constraints solving and use either**Outside**or**Inside**(not both). By default**Keep**option is used.

Figure 1.850



"Use as" property

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/ART/UseAsproper
Use asproperty is assigned to the arguments of the**Distance**and**Concentric**constraints. It has sense for circles, spheres, cylinders, cones and torii and defines which geometry is used in a 3D constraint to measure the distance or to made coincident with the other entity. There are four options for the**Use as**property:

- **Boundary**means that the constraint is applied to the boundary of the argument;
- **Axis**is applied to an argument that has axis, namely : circle, cylinder, cone, or torus;
- **Center**is applied to an argument that has central point, namely : circle, sphere, or torus;
- **Vertex**can be applied to the cone vertex only.

Figure 1.851

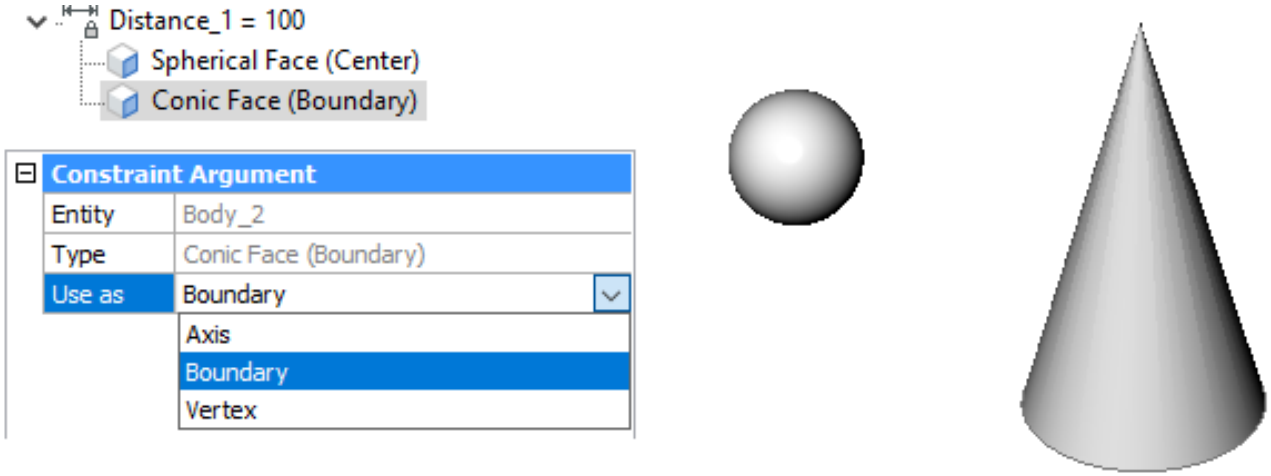


Figure 1.852

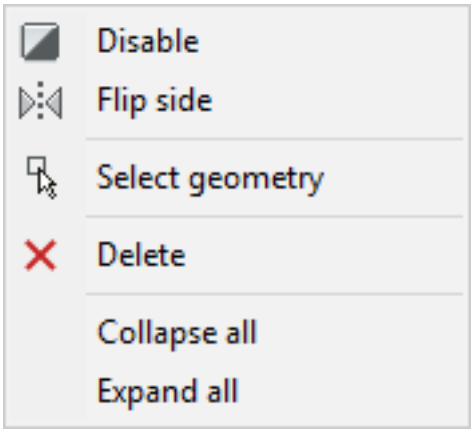
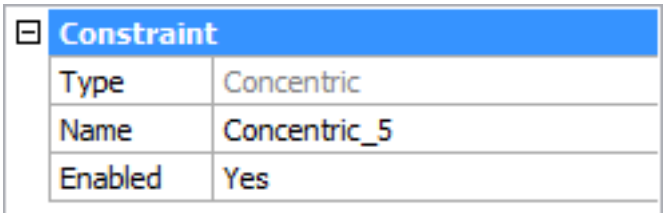


Figure 1.853



Constraint context menu	Right click a constraint: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M • Disable/ Enable: Toggles the selected constraint ON/OFF. The **Enabled** field in the Properties grid is adjusted accordingly. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou • Flip side: Allows to change the relative position of an entity (vectors normal of the selected faces point in the same direction or in the opposite direction). This option can be applied to **Parallel**, **Coincident** and **Concentric** constraints only and on condition the constraint is applied to faces. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sour (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sour • Select geometry: Selects the geometry, that is affected by the constraint, in the drawing. • Delete: Deletes the selected constraint. • Collapse all: collapses the main component and all components and subcomponents. • Expand all: expands the main component and all components and subcomponents.

Figure 1.854

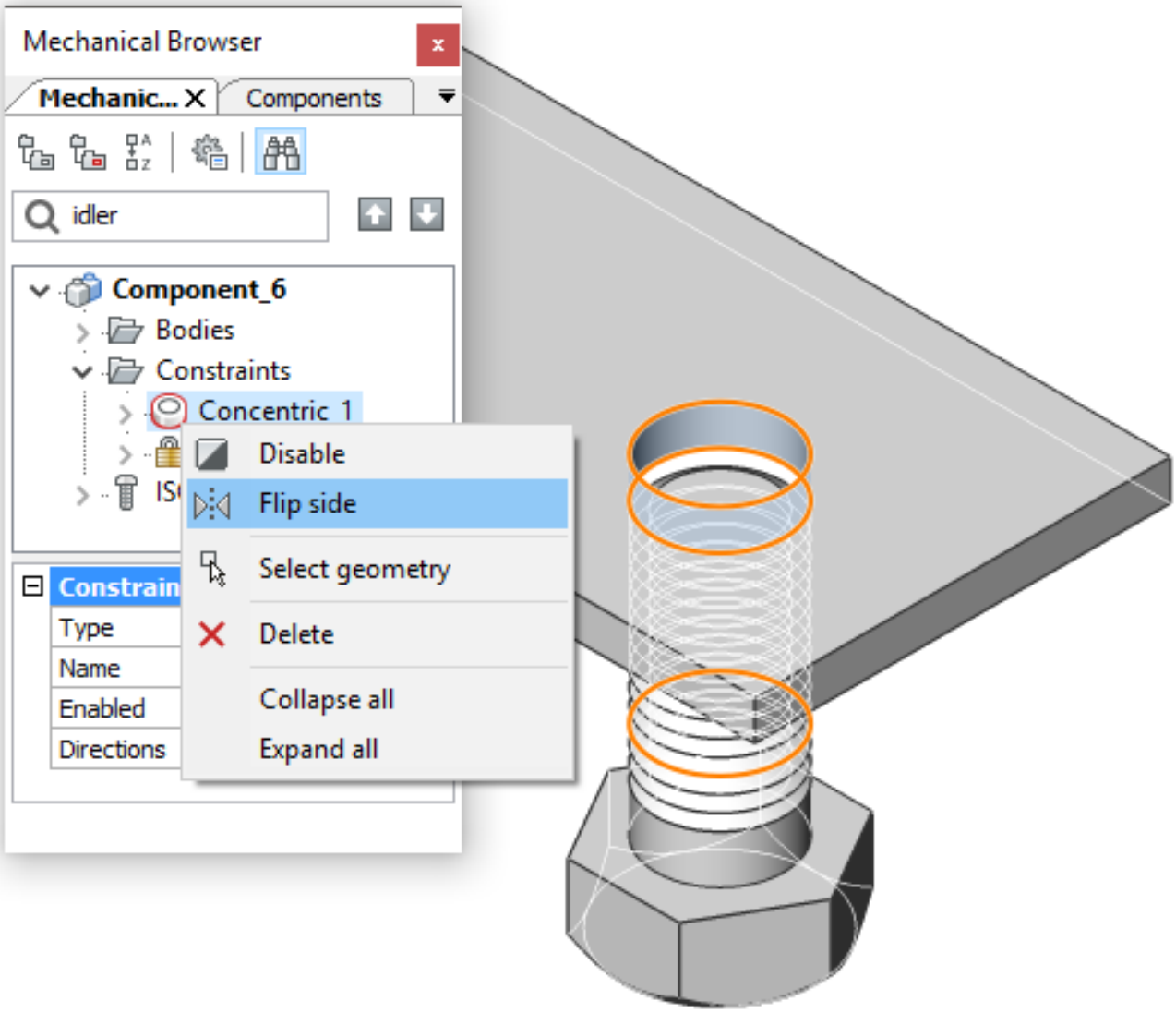
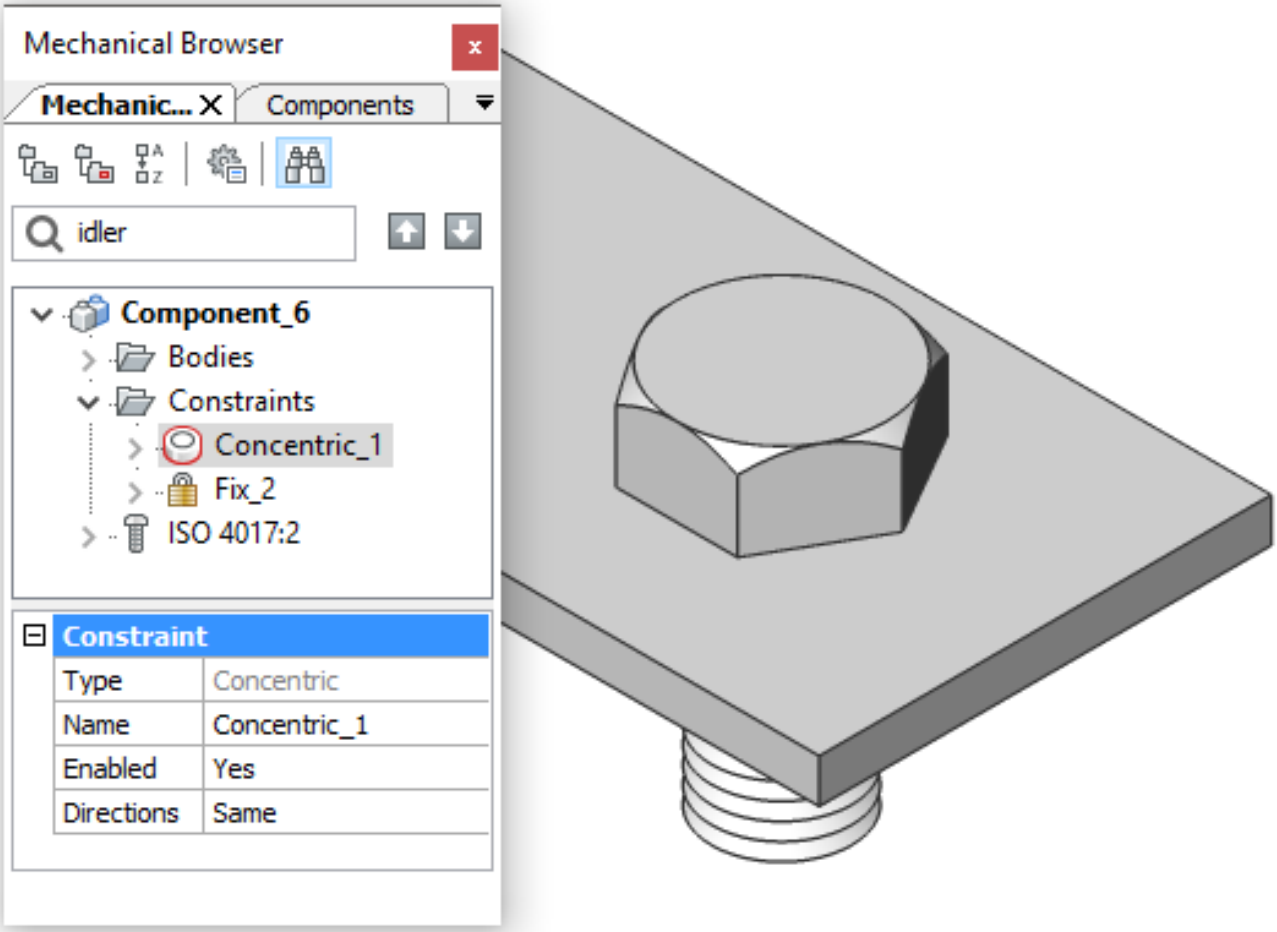

Mechanical Browser
Mechanic...
Components
idler
Component_6
Bodies
Constraints
Concentric 1
Disable
Flip side
Select geometry
Delete
Collapse all
Expand all
Constrain
Type
Name
Enabled
Directions

Figure 1.855


Mechanical Browser
Mechanic...
Components
idler
Component_6
Bodies
Constraints
Concentric_1
Fix_2
ISO 4017:2
Constraint
Type
Concentric
Name
Concentric_1
Enabled
Yes
Directions
Same

1.20 Point Cloud

1.20.1 Working with Point Clouds

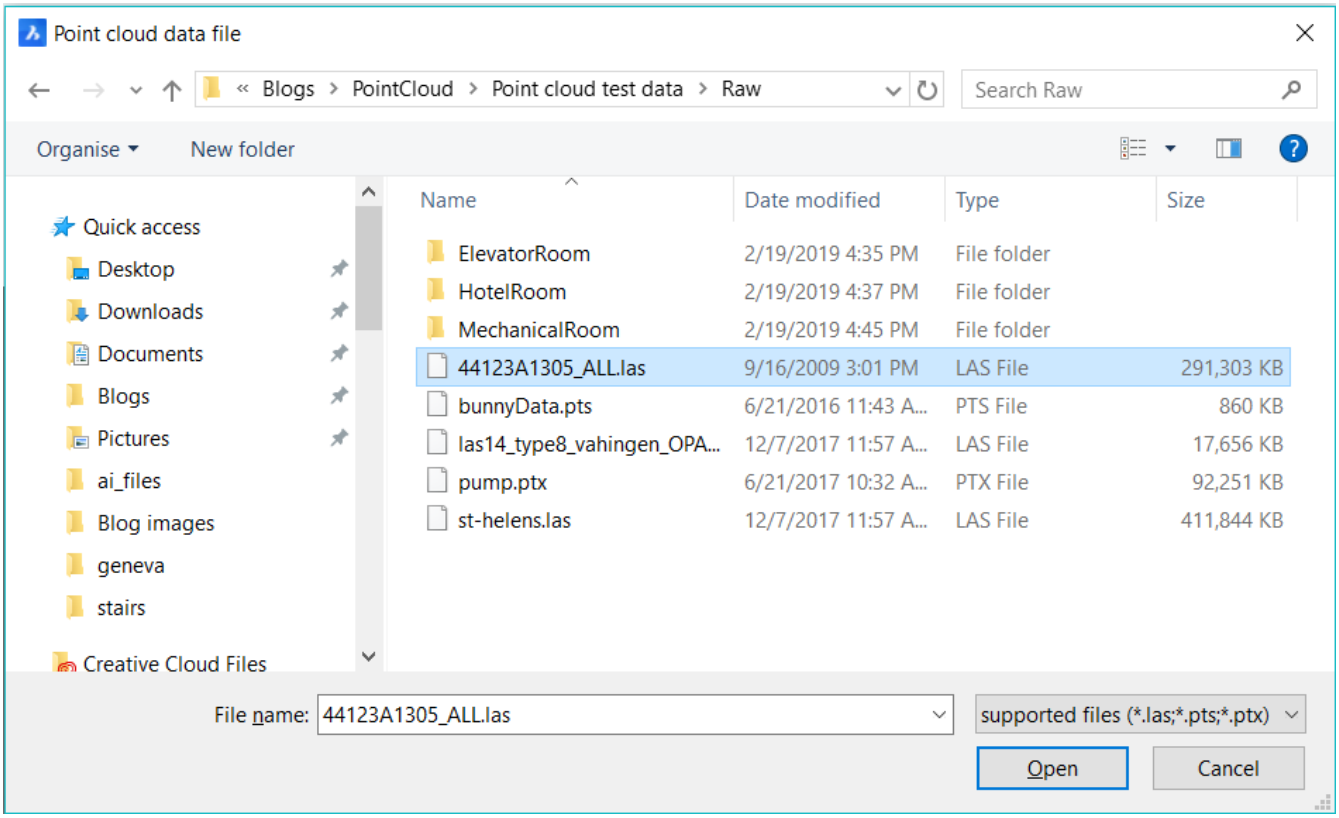
Command: PointCloud, (6.24.32) PointCloudAttach, (6.24.34) PointCloudBoundary, (6.24.35) PointCloudColorMap, (6.24.36) PointCloudCrop, (6.24.38) PointCloudPointSize, (6.24.39) PointCloudPointSize_Minus, (6.24.40) PointCloudPointSize_Plus, (6.24.59) PointCloudPointMax, (6.24.37) PointCloudPreprocess, (6.24.60) PointCloudUncrop, (6.24.61) PointCloud2dVsDisplay (6.24.33) Overview: In this article, you learn how to use point clouds in BricsCAD.

1. Prepare the data into the right format using preprocessing.
2. Attach the prepared data to a drawing in BricsCAD.
3. Use the drawing with the points, eg as a blueprint to start a drawing.

Preprocessing

File types supported: .las .pts .ptx. To be able to navigate large point cloud datasets quickly, its data must be preprocessed. Use the command POINTCLOUDPREPROCESS (6.24.60). This opens a file dialog that lets you select input point clouds. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PointCloud/Pointclouddatafi
The next step is to select the name of the preprocessed output file. By default, the file dialog suggests the same name of the input file (without numbers before the extension).This name can be modified. The file type is .bpt (Bricsys Point Tree). **For AutoCAD® users:** This is the equivalent of creating a .rcp file in Recap.

Figure 1.856



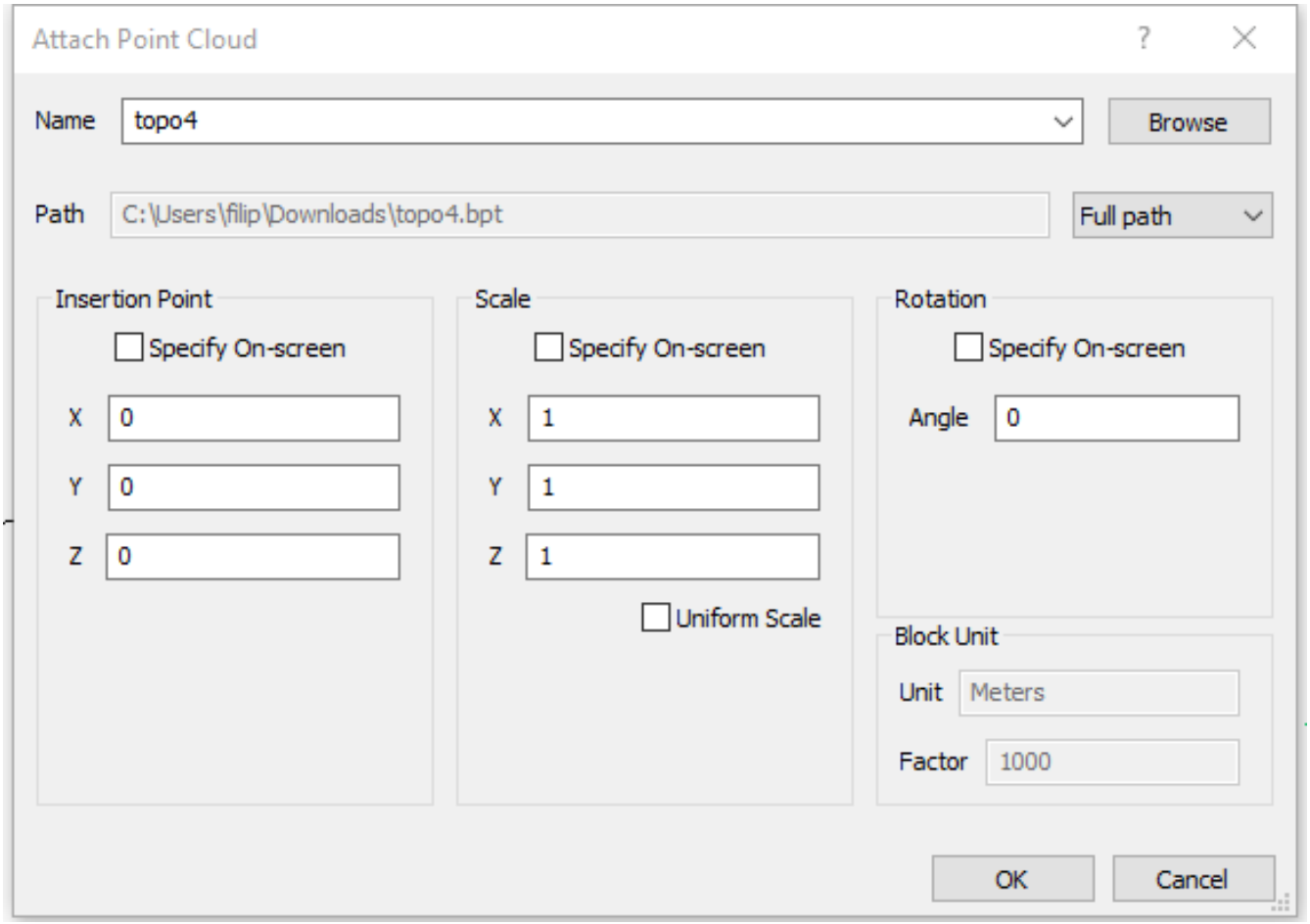
Attach to a drawing

Use the command POINTCLOUDATTACH (6.24.34) to attach the preprocessed .bpt file to the current .dwg file. **Note:** A point cloud can NOT be attached to an empty template. The .dwg has to be saved first. This is because the attached point cloud uses a relative path. When the .bpt to be attached is selected, a dialog appears. This allows the user to specify how to insert the point cloud into the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU **Figure 1: point cloud attach dialog. Here the user gets the possibility to choose how the point cloud is to be inserted in the drawing.** Options are:

- Insertion point
- Scale
- Rotation

When the checkbox ”specify on screen” is checked, an insertion box will appear to let the user determine translation/rotation/scaling whilst placing the point cloud on the drawing. For the other options, the values in the dialog boxes will be used.

Figure 1.857



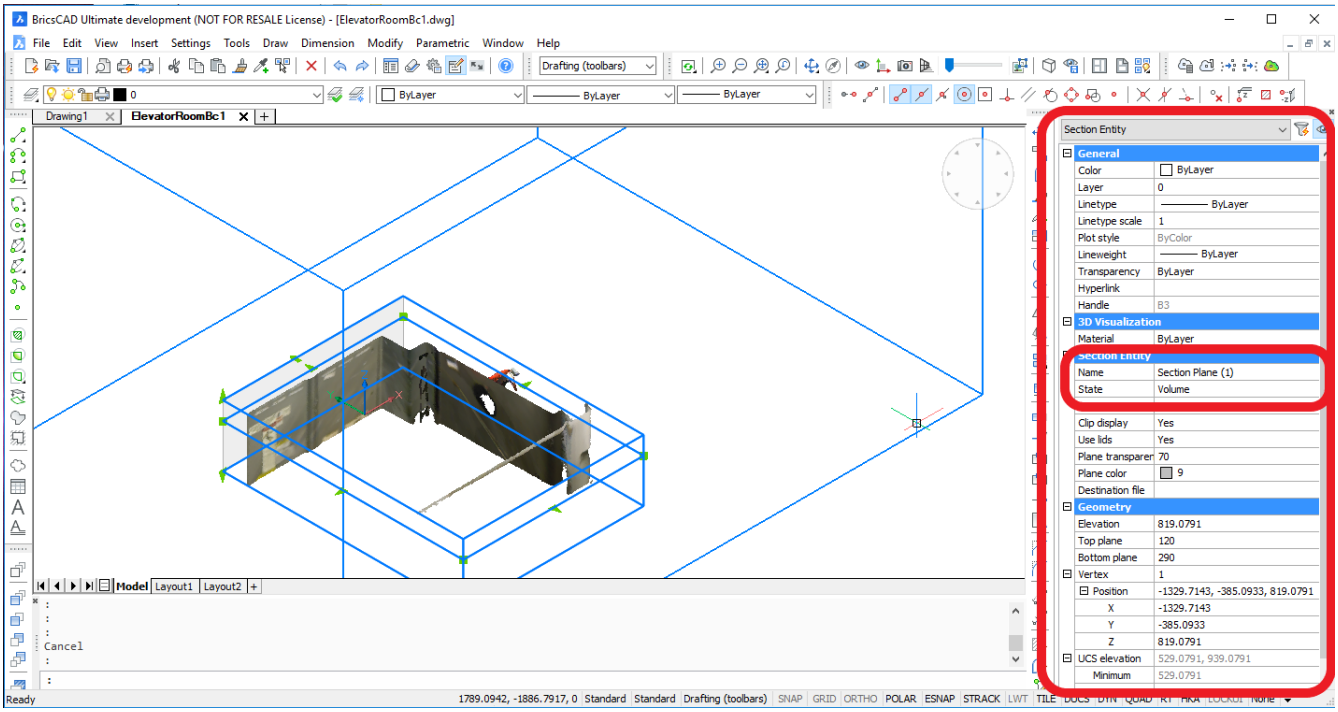
Procedure: Manipulate point cloud in .dwg

Once the point cloud is attached, it can be manipulated like other entities. Moved /rotated/scaled using the Quad. It also responds to section planes. Use the command**SECTIONPLANE** to hide parts outside section planes (Figure 2). **Note:** A section plane has three states:

- Plane: Clip everything away at one side of the plane.
- Boundary: Define an infinite extrusion of the contour drawn on the screen.
- Volume: The same as Boundary, but with a given top and bottom plane.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PointCloud/PointCloudSectionplane.png}) **Figure 2: A point cloud with a section plane (inner blue cuboid). In red: the section plane Properties panel. The section plane state is set to "volume".** Snapping to visible points in the cloud is also possible. Turn on endpoint and/or midpoint, to snap to points on a point cloud.

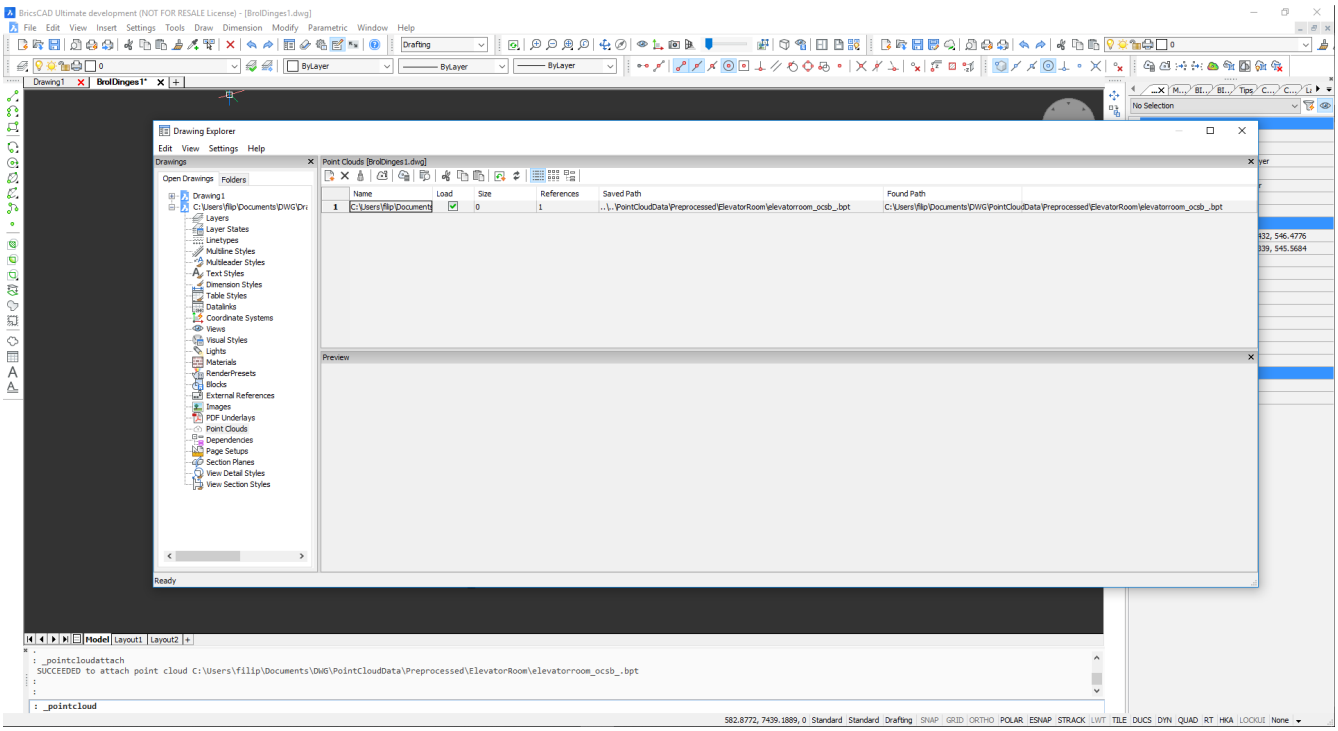
Figure 1.858



Drawing Explorer

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PointCloud/PointcloudDrawingExplorer.png})
Figure 3: An example of the Drawing Explorer for point clouds. The Drawing Explorer also supports point clouds, in particular the definitions of the point clouds attached to a dwg. When receiving a dwg and a point cloud of which the path cannot be referred correctly on your computer, it is possible to adjust the path for the point cloud there. The Drawing Explorer can be found via the GUI menus or via the command PointCloud (6.24.32)

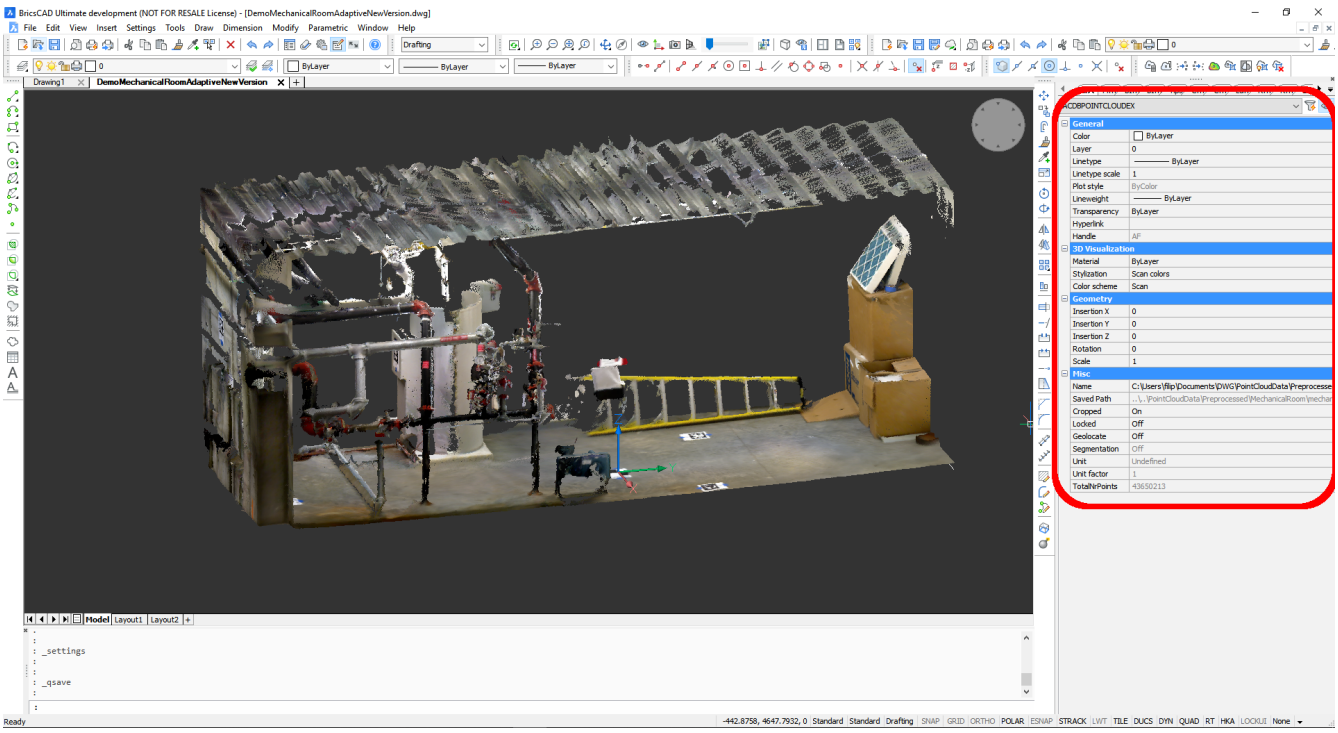
Figure 1.859



Properties panel

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PointCloud/PointcloudPropertiesPanel.png})
Figure 4: A point cloud in BricsCAD. In red left, the point cloud Properties Panel is shown. This panel appears when the point cloud is selected. When clicking (and thus selecting) a point cloud, its properties become available in the properties panel. From there, properties like insertion point, rotation, scaling, colormap, ... can also be adjusted.

Figure 1.860



Other functionalities

- **Colormap** (POINTCLOUDCOLORMAP (6.24.36)): False colors can be assigned to the points corresponding to intensity or elevation. Choose single spectrum colors such as red or blue, or go the whole spectrum.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PointCloud/PointCloudColormaps.png})
Figure 5: some examples of colormaps in function of the point elevation. The left part has colormap "Spectrum"; the middle part colormap "Hydro" and right part has colormap "Earth".

- **Cropping** (POINTCLOUDCROP (6.24.38)): Crop only the current point cloud, leaving other entities unaffected. A section plane crops everything in the scene.
- **Point Size** (POINTCLOUDPOINTSIZE (6.24.39)): Change display size of points. (Can be useful to make virtually fill in gaps.)
- **Maximum number of points displayed** (POINTCLOUDPOINTMAX (6.24.37)): Change the maximum number of points displayed. Since a computer screen has only a few million pixels, it makes no sense to display billions of points at the same time. This parameter sets the maximum for the number of points to be shown at any given instance on screen. Make no mistake, the actual point cloud can be many times larger than this (easily a few hundred million points and more).
- **Boundary** (POINTCLOUDBOUNDARY (6.24.35)): Show or hide the point cloud bounding box.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PointCloud/PointCloudBoundary.png})
Figure 6: point cloud has been sectioned and the left image shows the point cloud with boundary whereas the right image shows the point cloud without boundary.

Figure 1.861

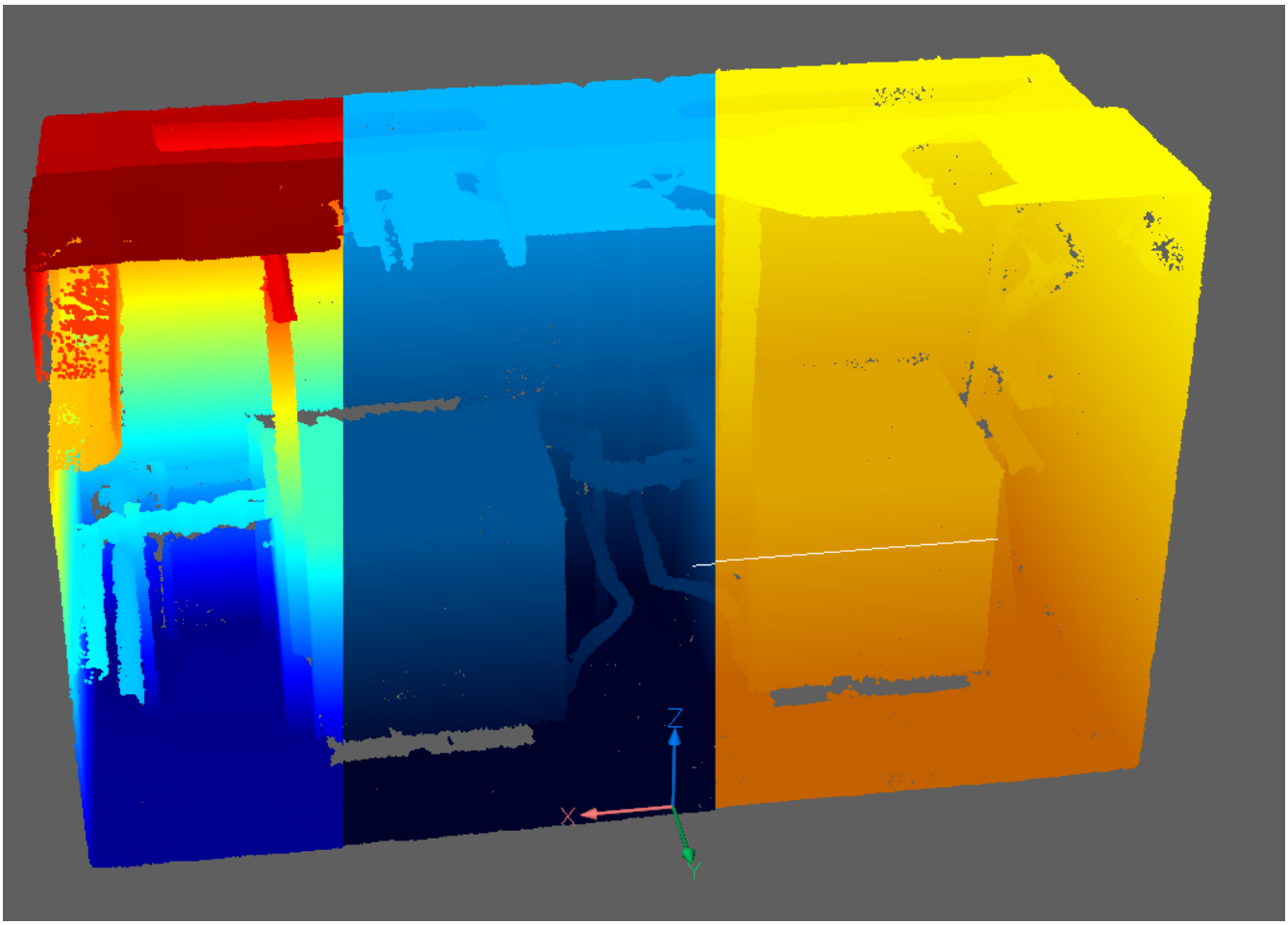


Figure 1.862



AutoCAD® compatibility

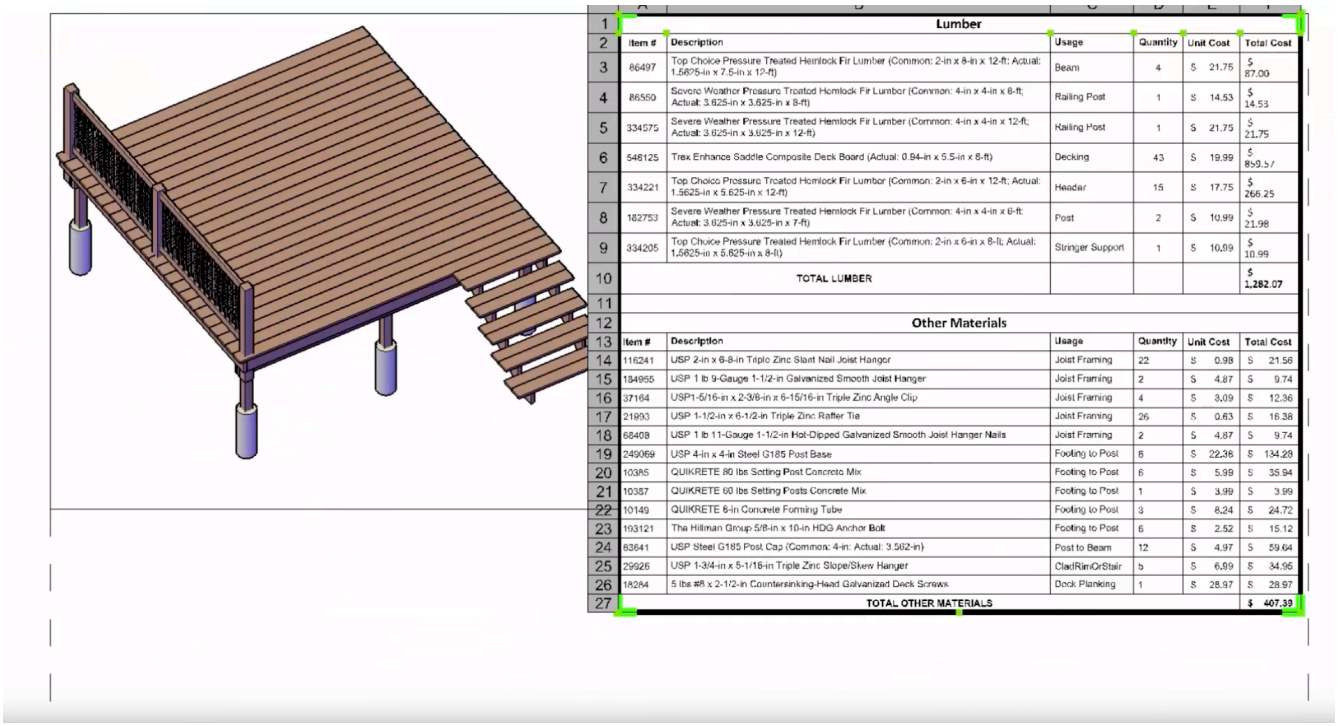
AutoCAD® can open and use the .dwg with the same point cloud, clippings and other entities present in the drawing. When the user also has AutoCAD® and Recap, save the Recap preprocessed, point cloud file in the same location with the same name as our .bpt file. The extension will be different (.rcp).

1.21 Annotate

1.21.1 Link data from Excel sheets

Commands: DATALINK (6.10.2), DATALINKUPDATE (6.10.3), TABLE (6.29.1) **DataLink**:Links data from an Excel spreadsheet directly into drawing tables for bi-directional updating. **DataLinkUpdate**:Synchronizes the linked data in tables in the current drawing with data in the linked source file. **Table**:Creates a table entity in the drawing through a dialog box. Link data from an Excel spreadsheet directly into drawing tables for bi-directional updating. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Annotate/LinkdatafromExcelsheets.png})

Figure 1.863



Lumber

Item #	Description	Usage	Quantity	Unit Cost	Total Cost
86497	Top Choice Pressure Treated Hemlock Fir Lumber (Common: 2-in x 8-in x 12-ft; Actual: 1.5625-in x 7.5-in x 12-ft)	Beam	4	$ 21.75	$ 87.00
86550	Severe Weather Pressure Treated Hemlock Fir Lumber (Common: 4-in x 4-in x 8-ft; Actual: 3.625-in x 3.625-in x 8-ft)	Railing Post	1	$ 14.53	$ 14.53
334575	Severe Weather Pressure Treated Hemlock Fir Lumber (Common: 4-in x 4-in x 12-ft; Actual: 3.625-in x 3.625-in x 12-ft)	Railing Post	1	$ 21.75	$ 21.75
548125	Trex Enhance Saddle Composite Deck Board (Actual: 0.94-in x 5.5-in x 8-ft)	Decking	43	$ 19.99	$ 859.57
334221	Top Choice Pressure Treated Hemlock Fir Lumber (Common: 2-in x 6-in x 12-ft; Actual: 1.5625-in x 5.625-in x 12-ft)	Header	15	$ 17.75	$ 266.25
182753	Severe Weather Pressure Treated Hemlock Fir Lumber (Common: 4-in x 4-in x 8-ft; Actual: 3.625-in x 3.625-in x 7-ft)	Post	2	$ 10.99	$ 21.98
334205	Top Choice Pressure Treated Hemlock Fir Lumber (Common: 2-in x 6-in x 8-ft; Actual: 1.5625-in x 5.625-in x 8-ft)	Stringer Support	1	$ 10.99	$ 10.99
	TOTAL LUMBER				$ 1,282.07

Other Materials

Item #	Description	Usage	Quantity	Unit Cost	Total Cost
116241	USP 2-in x 6-8-in Triple Zinc Skant Nail Joist Hangor	Joist Framing	22	$ 0.98	$ 21.56
184955	USP 1 lb 9-Gauge 1-1/2-in Galvanized Smooth Joist Hanger	Joist Framing	2	$ 4.87	$ 9.74
37164	USP1-5/16-in x 2-3/8-in x 6-15/16-in Triple Zinc Angle Clip	Joist Framing	4	$ 3.09	$ 12.36
21993	USP 1-1/2-in x 6-1/2-in Triple Zinc Rafter Tie	Joist Framing	26	$ 0.63	$ 16.38
68408	USP 1 lb 11-Gauge 1-1/2-in Hot-Dipped Galvanized Smooth Joist Hanger Nails	Joist Framing	2	$ 4.87	$ 9.74
249069	USP 4-in x 4-in Steel G185 Post Base	Footing to Post	6	$ 22.38	$ 134.28
10385	QUIKRETE 80 lbs Setting Post Concrete Mix	Footing to Post	6	$ 5.99	$ 35.94
10387	QUIKRETE 60 lbs Setting Posts Concrete Mix	Footing to Post	1	$ 3.99	$ 3.99
10149	QUIKRETE 6-in Concrete Forming Tube	Footing to Post	3	$ 8.24	$ 24.72
193121	The Hillman Group 5/8-in x 10-in HDG Anchor Bolt	Footing to Post	6	$ 2.52	$ 15.12
63641	USP Steel G185 Post Cap (Common: 4-in; Actual: 3.562-in)	Post to Beam	12	$ 4.97	$ 59.64
29926	USP 1-3/4-in x 5-1/16-in Triple Zinc Slope/Skew Hanger	CladRimOrStair	6	$ 6.99	$ 34.95
18284	5 lbs #8 x 2-1/2-in Countersinking-Head Galvanized Deck Screws	Deck Planking	1	$ 28.97	$ 28.97
	TOTAL OTHER MATERIALS				$ 407.39

1.21.2 Dimensioning tools overview

All tools to create dimension entities can be found either on the **Dimensions** toolbar or in the **Dimensions** menu.

Toolbar
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/01Toolbar07Dim
Menu
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/01UserInterface/Art/00DimensionMe

Icon	Tool name	Command	Description
	Linear (1.21.8)	DIMLINEAR (6.11.14)	Creates horizontal or vertical linear dimensions.
	Aligned (1.21.8)	DIMALIGNED (6.11.3)	Creates linear dimensions which are aligned with the dimension points.
	Arc (1.21.8)	DIMARC (6.11.5)	Creates arc length dimensions
	Radius (1.21.10)	DIMRADIUS (6.11.17)	Creates radial dimensions in circles and circular arcs.
	Diameter (1.21.10)	DIMDIAMETER (6.11.10)	Creates diametrical dimensions in circles and circular arcs.
	Angular (1.21.9)	DIMANGULAR (6.11.4)	Creates angular dimensions
	Baseline (1.21.8)	DIMBASELINE (6.11.6)	Creates stacked linear dimensions.
	Continue (1.21.8)	DIMCONTINUE (6.11.9)	Creates a chain of linear dimensions, starting from an existing linear dimension entity.
	Ordinate (1.21.12)	DIMORDINATE (6.11.15)	Creates ordinate dimensions.
	Rotated (1.21.8)	DIMLINEAR (6.11.14)	Creates rotated linear dimensions.
	Make Oblique (1.21.15)	DIMEDIT (6.11.12) + O	Makes the extension lines of the selected dimension(s) oblique.
	Multileader (1.21.14)	MLEADER (6.21.20)	Creates Mleaders (multileaders) using a multileader style.
	Edit Multileader	MLEADEREDIT (6.21.23)	Adds leader lines to and removes leader lines from a multileader entity.
	Multileader add leaders	AIMLEADEREDITADD	Adds leader lines to a multileader.
	Multileader remove leaders	AIMLEADEREDITREMOVE	Removes leader lines from a multileader.
	Tolerance (1.21.16)	TOLERANCE (6.29.17)	Defines the layout of geometric tolerances.
	Center Lines (1.21.10)	DIMCENTER (6.11.8)	Draws center lines in circles and circular arcs.
	Edit Dimension Text (1.21.15)	DIMEDIT (6.11.12)+ E	Defines the dimension text of the selected dimension(s).
	Rotate Dimension Text (1.21.15)	DIMEDIT (6.11.12)+ R	Rotates the dimension text of the selected dimension(s).
	Reposition Dimension Text (1.21.15)	DIMTEDIT (6.11.23)	Lets you reposition the dimension text.

Figure 1.864



	Restore Text Position (1.21.15)	DIMEDIT (6.11.12)	Undoes the rotation or repositioning of the dimension text of the selected dimension(s).
	Apply Style (1.21.7)	-DIMSTYLE (6.11.21) + A	Applies the current dimension style to the selected dimension(s).
	Save Style (1.21.7)	-DIMSTYLE (6.11.21) + S	Saves the current dimension settings in a new dimension style.
	Restore Style (1.21.7)	-DIMSTYLE (6.11.21)	Sets the current dimension style.
	Dimension Variable Status (1.21.5)	-DIMSTYLE (6.11.21) + ST	Displays the status of the dimension variables in the command bar and the Prompt History window.
	Leader (1.21.13)	DIMLEADER (6.11.13)	Creates a leader with or without text.
	Quick Leader (1.21.13)	QLEADER (6.25.1)	Draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.

Figure 1.865

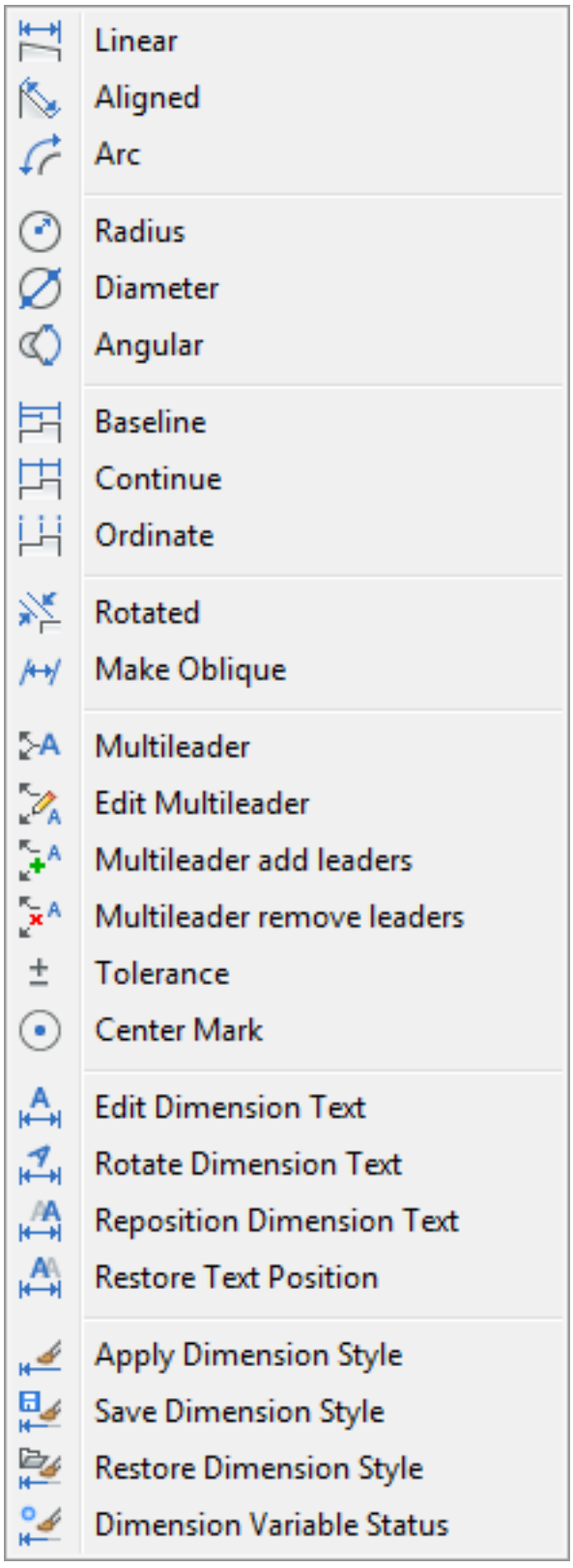
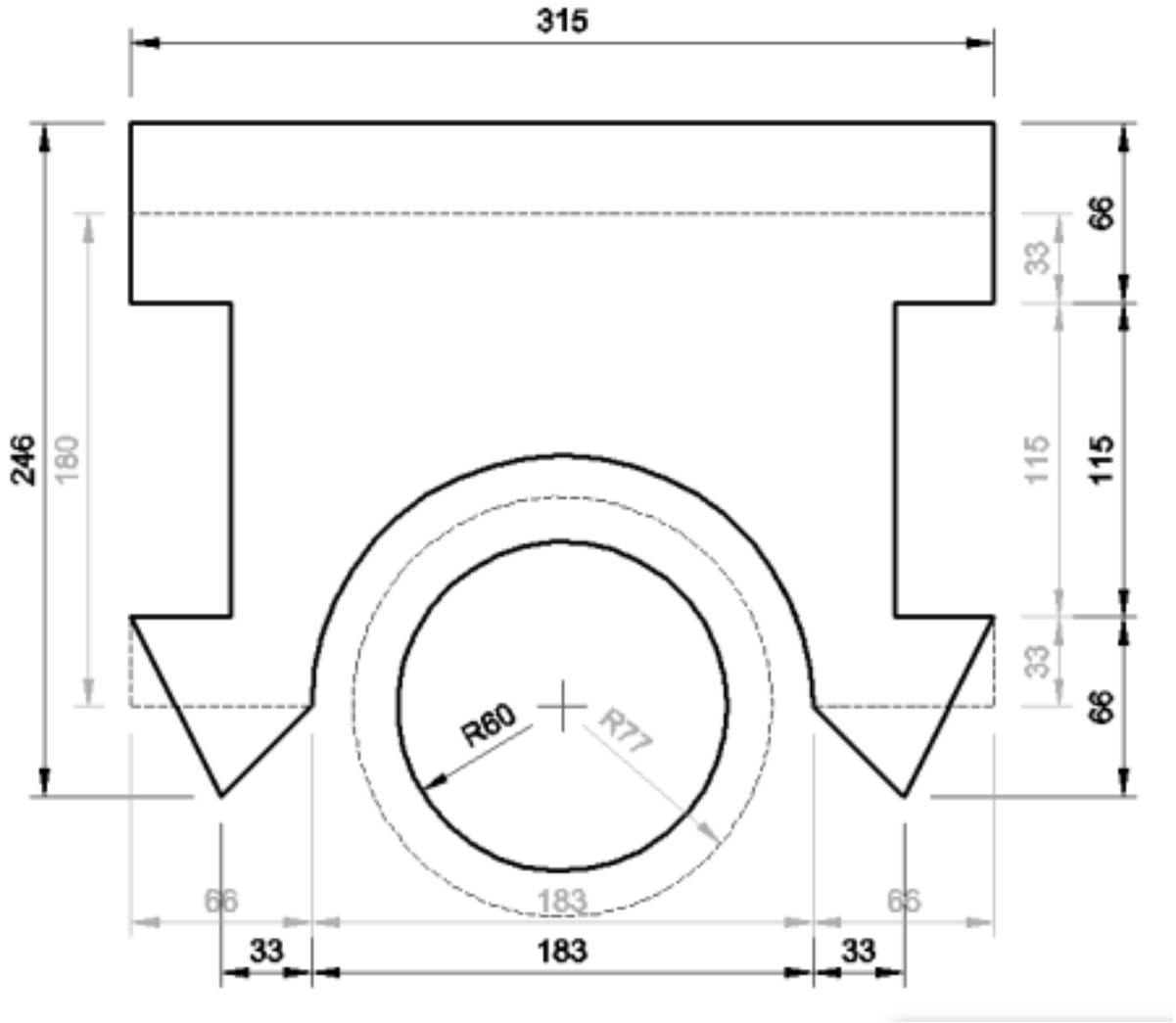

Linear
Aligned
Arc
Radius
Diameter
Angular
Baseline
Continue
Ordinate
Rotated
Make Oblique
Multileader
Edit Multileader
Multileader add leaders
Multileader remove leaders
Tolerance
Center Mark
Edit Dimension Text
Rotate Dimension Text
Reposition Dimension Text
Restore Text Position
Apply Dimension Style
Save Dimension Style
Restore Dimension Style
Dimension Variable Status

Figure 1.866



1.21.3 Associative Dimensions

Dimensions created in BricsCAD are associative: if the dimensioned entity is moved or modified, the dimensions will update automatically. The associative property of dimensions also applies to dimensions which are placed in paperspace and are associated with model space entities. If necessary, you can use the DIMREGEN (6.11.19) command to update all associative dimensions. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sc
The following Entity Snaps (1.15.15) can be used to create associative dimensions: endpoint, midpoint, center, perpendicular, quadrant, insertion, point, intersection and apparent intersection.

1.21.4 Dimension Styles explorer

Command: DIMSTYLE (6.11.20) A dimension style is a collection of settings and definitions which determine the type of arrowheads, text style, color of text, lines and arrowheads, unit type, precision, etc. Dimension styles are saved in the drawing. You can create, edit and delete dimension styles. Dimension styles can be copied between drawings. Dimension styles which have the **Annotative** property create annotative dimensions (1.21.32).

NOTES

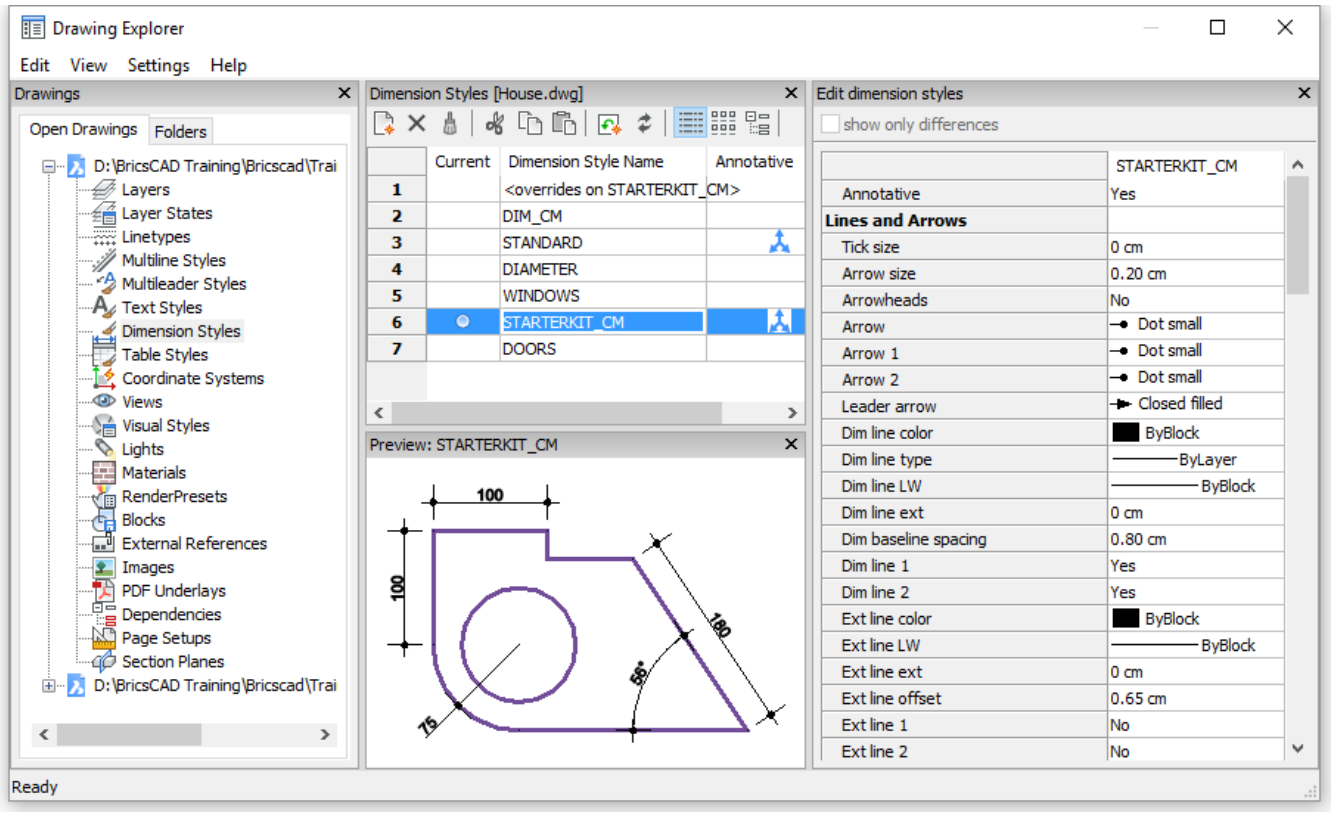
- You cannot delete the current dimension style.
- You cannot delete a dimension style that is used in the drawing.
- Each drawing contains at least one dimension style.

Exploring dimension styles

1. To open the **Dimension Styles Explorer** , do one of the following:
   - Choose **Dimension Styles ...** in the **Tools > Drawing Explorer** menu.
   - Double click the **Dimension Style** field in the Status bar (1.12.2).
   - Right click the **Dimension Style** field in the Status bar (1.12.2), then choose **Properties** in the context menu.
   - Type **dimstyle** in the command bar, then press Enter.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06Workingw
   The current dimension style is marked in the **Current** column.
2. (option) Select a dimension style to see a preview.
3. (option) Select a dimension style then right click:
   - Choose **New** to create a new dimension style (1.21.6).
   - Choose **Delete** to delete the selected dimension style (1.21.6).
   - Choose **Cut** selected dimension style in the current drawing, then paste it in another open drawing (1.21.6).
   - Choose **Copy** , then paste it in another open drawing (1.21.6).
   - Choose **Paste** , to copy a previously cut or copied dimension style in the current drawing.
   - Choose **Rename** , to rename (1.21.6) of the selected dimension style.
   - Choose **Set current** in the context menu to make it the current dimension style (1.21.6).
   - Choose **Save overrides to current style** , to save all current overrides to the current style.
   - Choose **Save to new style,** to create a new dimension style as a copy of the selected dimension style.
4. (option) Modify a dimension style definition.
5. (option) Select **<overrides>** to define override settings for the current dimension style.

Figure 1.867



Comparing Dimstyles

1. Open the Drawing Explorer > (1.21.4) Dimension Styles dialog.
2. Select 2 or more dimension styles. Press and hold the Ctrl-key to add more dimension style to the selection set. The properties of the selected styles display in the Editor pane. Properties that are different in one of the selected styles display on a yellow background. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/
3. Check the **Show only differences** option to display the properties that are different for at least one of the selected dimension styles only. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10

Figure 1.868

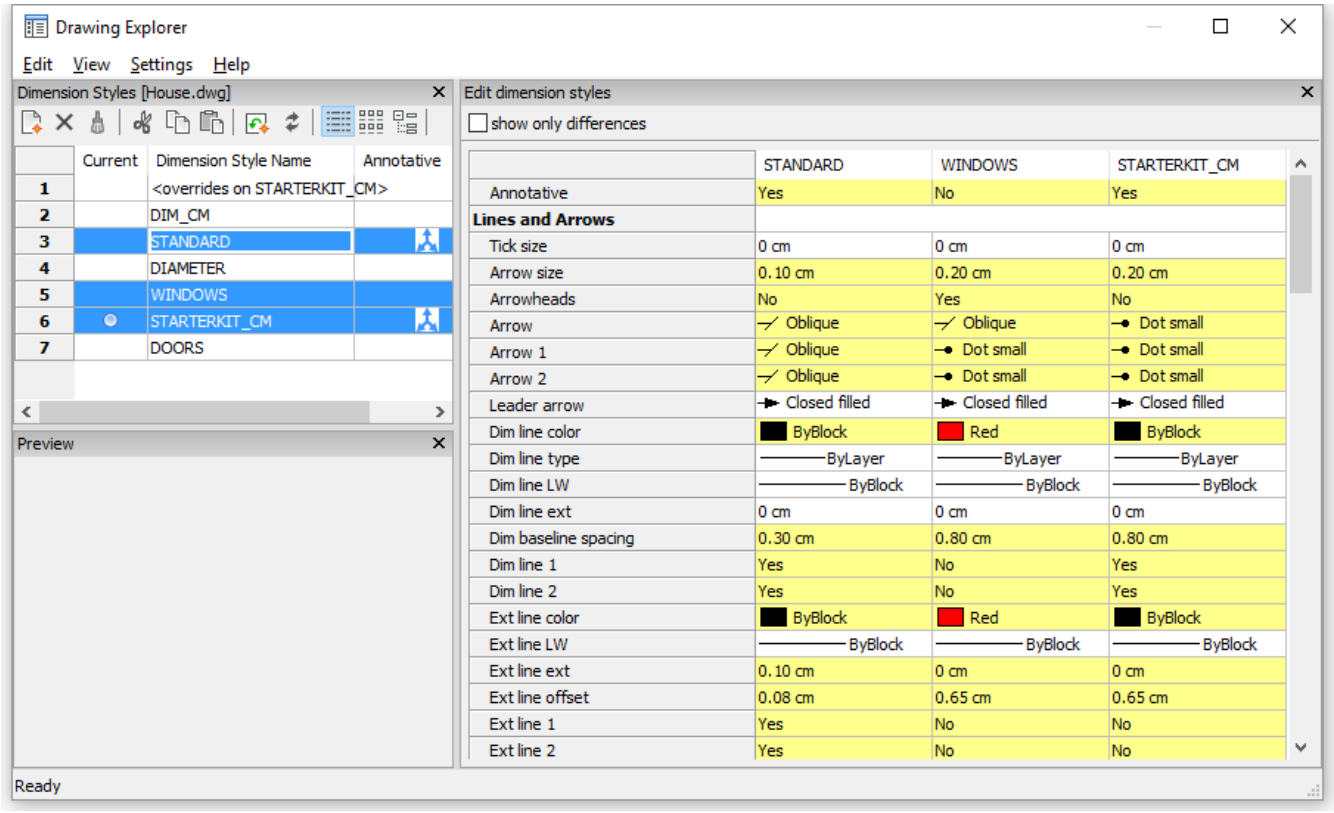
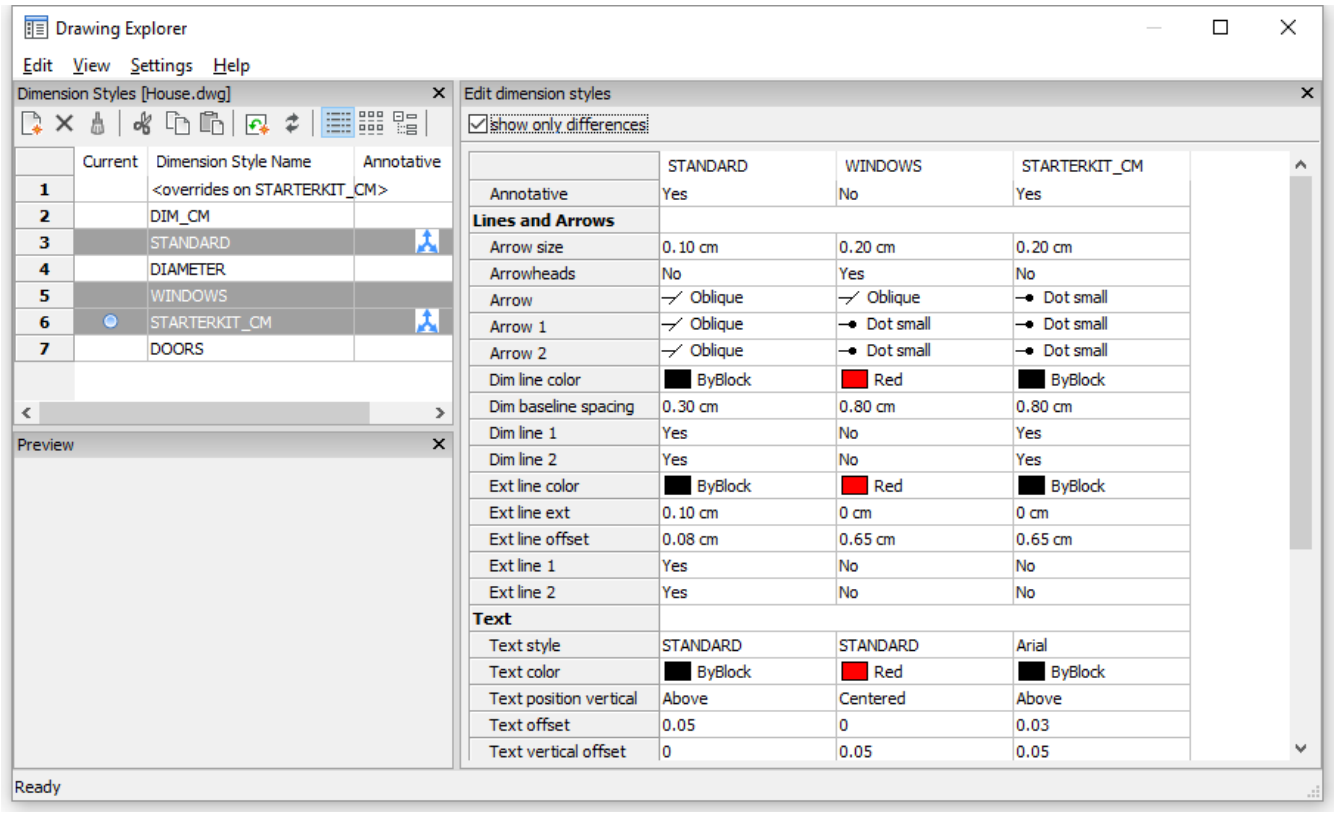


Figure 1.869



1.21.5 Dimension Settings

Commands: DIMSTYLE (6.11.20), -DIMSTYLE (6.11.21) A dimension style consists of six settings groups

- Lines and Arrows (6.11.20)
- Text (6.11.20)
- Fit (6.11.20)
- Primary Units (6.11.20)
- Alternate Units (6.11.20)
- Tolerances (6.11.20)

To list the current status of the dimension variables

1. Do one of the following:

   - Click the **Dimension Variable Status** tool button () on the **Dimensions** toolbar.
   - Choose **Dimension Variable Status** in the **Dimensions** menu.

   A list of all dimension variables displays in the command bar.

2. Press the **F2** function key to open the **BricsCAD Prompt History** window.
3. Press Enter to continue the listing.
4. Press the **F2** function key to close the **BricsCAD Prompt History** window.

1.21.6 Working with dimension styles

In the Dimension Styles explorer (1.21.4) you can:

- Create a new dimension style
- Edit a dimension style
- Define dimension style overrides
- Rename a dimension style
- Set a dimension style current
- Copy dimension styles between drawings
- Move dimension styles between drawings
- Delete a dimension style

To create a new dimension style

1. Open theDimension Styles explorer (1.21.4)dialog.
2. Do one of the following:
   - Click the **New** button ( ).
   - Place the cursor on the dimension style list, then right click and choose **New** in the context menu.

   A **New Dimension Style** is added.
3. Select the **Name** field and type a name for the new dimension style.
4. Define the settings (1.21.5) for the new dimension style.

NOTE

- The **New** tool creates a new dimension style as a copy of the current dimension style.
- Use the **Save to new style** option of the context menu to create a new dimension style as a copy of the selected dimension style.
- Use the **Save overrides to current style** option of the context menu to create a new dimension style from the current override settings. This option is available if you select **<overrides>** only.
- Alternatively you can use the Save Style tool (1.21.7) to create a new dimension style.

To create a dimension child style

For each dimension style you can create a child style for each dimension type: Linear, Angular, Radius, Diameter, Ordinate and Leader. If the current dimension style has a child style for a dimension type, the child style will be used automatically when such dimension type is created.

1. Select the parent dimension style, then right click and choose a child style in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06WorkingwithDim
   The name of the child style is: <parent_style>:<child_style>
2. Select the child style and define the properties that need to be different from the parent style.

Figure 1.870



To edit a dimension style

1. Open theDimension Styles explorer (1.21.4)dialog in **Tree view** mode ( ).
2. Expand the dimension style.
3. Adjust the dimensionsettings (1.21.5).
4. Close the **Dimension Styles explorer** dialog.

To define dimension style overrides

1. Open theDimension Styles explorer (1.21.4)dialog in **Tree view** mode ( ).
2. Select **<overrides>**.
3. Define the dimension override settings.
4. Close the **Dimension Styles explorer** dialog.

NOTES

- It is not recommended to create dimensions with overrides.
- Use the dimension override settings to test changes, without modifying the current dimension style definition, then use the **Save overrides to current style** or **Save to new style** option.
- Setting another style current will discard the existing dimension style overrides.

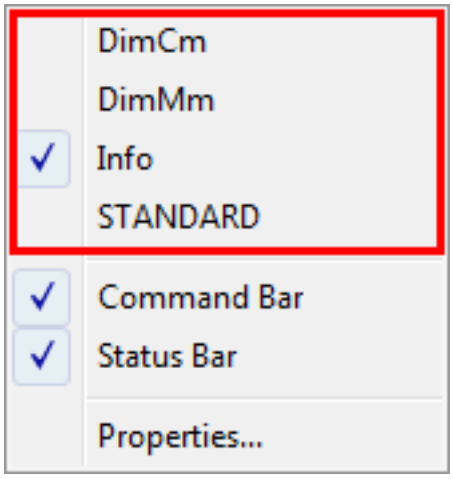
To rename a dimension style

1. Open theDimension Styles explorer (1.21.4)dialog.
2. Select the dimension style you want to rename.
3. Right click, then choose **Rename** in the context menu.
4. Type a new name in the **Name** field, then press enter.

To set a dimension style current

1. Open theDimension Styles explorer (1.21.4)dialog. The current dimension style is marked in **Tree view** and **Detail view** mode.
2. Select the dimension style.
3. Right click and choose Set current in the context menu. In **Tree view** and **Detail view** mode a mark indicates the selected dimension style is made current.

Figure 1.871



2. Right click. A list of available dimension styles displays in the context menu. The current dimension style is checked. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06

3. Select the new current dimension style. The selected dimension style name displays in the **Dimension style** field of the **Status bar**.

NOTE Alternatively you can use the Restore Style tool (1.21.7) to make a dimension style current.

To copy a dimension style between drawings

1. Open theDimension Styles explorer (1.21.4)dialog.
2. In the source drawing, select the dimension style you want to copy.
3. Do one of the following:
   - Click the **Copy** button ( ).
   - Right click, then choose **Copy** in the context menu.
4. In the **Open Drawings** section, expand the target drawing.
5. Select **Dimension Styles** in the target drawing. The list of the dimension styles in the target drawing displays in the **Details** section.
6. Do one of the following:
   - Click the **Paste** button ( )
   - Place the cursor on one of the existing dimension style names, then right click and choose **Paste** in the context menu.

NOTE To select multiple dimension styles, choose **Detail view** ( ) or **Icon view** ( ) in the **Drawing Explorer** dialog.

To move a dimension style between drawings

1. Open theDimension Styles explorer (1.21.4)dialog.
2. In the source drawing, select the dimension style you want to move.
3. Do one of the following:
   - Click the **Cut** button ( ).
   - Right click, then choose **Cut** in the context menu.
4. In the **Open Drawings** section, expand the target drawing.
5. Select **Dimension Styles** in the target drawing. The list of the dimension styles in the target drawing displays in the **Details** section.

NOTE

- You cannot cut the current dimension style.
- You cannot cut a dimension style that is used in the drawing.
- To select multiple dimension styles, choose **Detail view** () or **Icon view** () in the **Drawing Explorer** dialog.

To delete a dimension style

1. Open theDimension Styles explorer (1.21.4)dialog.
2. Select the dimension style you want to delete.
3. Do one of the following:
   - Click the **Delete** button ().
   - Right click, then choose **Delete** in the context menu.

The selected dimension style is deleted.

NOTE

- You cannot delete the current dimension style.
- You cannot delete a dimension style that is used in the drawing.
- To select multiple dimension styles, choose **Detail view** () or **Icon view** () in the **Drawing Explorer** dialog.

1.21.7 Using the dimension style tools

Command: -DIMSTYLE (6.11.21) Using the dimension style tools you can:

- Apply the current dimension style to a selection of dimension entities.
- Create a new dimension style.
- Restore a dimension style.
- List the current status of the dimension settings.

To apply the current dimension style

1. Do one of the following:

   - Click the **Apply Style** tool button () on the **Dimensions** toolbar.
   - Choose **Apply Style** in the **Dimensions** menu.
   - Type **-dimstyle** in the command window, press Enter then type **A** or **apply** and press Enter.

   You are prompted: Select dimensions to apply the current style.

2. Select the dimensions (1.23.6), then press Enter.

NOTE	Since the dimension style settings only apply to dimension entities, you can select all entities in step 2 to apply the current dimension style to all dimensions in the drawing.

To save a dimension style

1. Do one of the following:

   - Click the **Save Style** tool button () on the **Dimensions** toolbar.
   - Choose **Save Style** in the **Dimensions** menu.
   - Type **-dimstyle** in the command window, press Enter then type **S** or **save** and press Enter.

   You are prompted: Save style: ? to list styles/Name for new dimension style:

2. Type a name for the new dimension style, then press Enter.

3. (option) If the name already exists, the command bar prompts you to redefine it or not. **You are prompted:** That name is already in use. Redefine it? <N>: Do one of the following:

   - Press Enter to not overwrite the existing style. **You are prompted** to type a new name.
   - Type **Y** , then press Enter to update the existing dimension style.

To restore a dimension style

1. Do one of the following:

   - Click the **Restore Style** tool button () on the **Dimensions** toolbar.
   - Choose **Restore Style** in the **Dimensions** menu.
   - Type **-dimstyle** in the command window, press Enter then type **R** or **restore** and press Enter.

   You are prompted: ? to list styles/Enter to select dimension/<Dimension style>:

2. Do one of the following:

   - Type a dimension style name and press Enter.
   - Press Enter, then click a dimension entity in the drawing. The dimension style of the selected entity is the current dimension style now.
   - Type ? to see a list of the dimension styles in the current drawing.

To list the settings in the current dimension style

Do one of the following:

- Click the **Dimension Variable Status** tool button () on the **Dimensions** toolbar.
- Choose **Dimension Variable Status** in the **Dimensions** menu.
- Type **-dimstyle** in the command window, press Enter then type **ST** or **status** and press Enter.

The status of the settings in the current dimension style are listed in the **Prompt History** window.

1.21.8 Creating linear dimensions

Commands: DIMLINEAR (6.11.14), DIMALIGNED (6.11.3), DIMBASELINE (6.11.6), DIMCONTINUE (6.11.9), and DIMARC (6.11.5) Linear dimensions annotate linear distances or lengths and can be oriented either horizontally, vertically or aligned parallel to an existing entity or to the selected origin points. Starting from a linear dimension, you can add a **baseline** dimension or a **continued** dimension.

- A baseline dimension inserts an additional dimension from a common first extension line origin of a previous linear dimension.
- A continued dimension continues a linear dimension from the second extension line of a previous linear dimension.

To create a horizontal or vertical linear dimension

1. Do one of the following:

   - Click the **Linear** tool button ( ) on the **Dimensions** toolbar.
   - Choose **Linear** in the **Dimensions** menu.
   - Type **dimlinear** in the command bar, then press Enter.

   You are prompted: ENTER to select entity/<Origin of first extension line>:

2. Do one of the following:

   - Press Enter, then select the line or linear segment of a polyline you want to dimension.
   - Specify the origin of the first extension line, then specify the origin of the second extension line.

   The dimension line displays dynamically. Move the cross hairs vertically to create a horizontal dimension, or horizontally to create a vertical dimension.

3. Position the dimension line.

To create an aligned linear dimension

1. Do one of the following:

   - Click the **Aligned** tool button ( ) on the **Dimensions** toolbar.
   - Choose **Aligned** in the **Dimensions** menu.
   - Type **dimaligned** in the command bar, then press Enter.

   You are prompted: ENTER to select entity/<Origin of first extension line>:

2. Do one of the following:

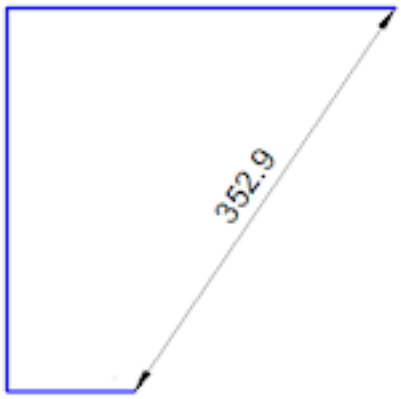
   - Press Enter, then select the line or linear segment of a polyline you want to dimension.
   - Specify the origin of the first extension line, then specify the origin of the second extension line.

3. Do one of the following:

   - Click a point to position the dimension line.
   - Press **0** (zero) to place the dimension line at the dimension points, thus omitting the extension lines.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06Workingw

Figure 1.872



To create a rotated linear dimension

1. Do one of the following:

   - Click the **Rotated** tool button ( ) on the **Dimensions** toolbar.
   - Choose **Rotated** in the **Dimensions** menu.

   You are prompted: ENTER to select entity/<Origin of first extension line>:

2. Do one of the following:

   - Press Enter, then select the line or linear segment of a polyline you want to dimension.
   - Specify the origin of the first extension line, then specify the origin of the second extension line.

   You are prompted: Angle of dimension line <0>:

3. Do one of the following:

   - Type the rotation angle in the command bar, then press Enter.
   - Enter the rotation angle by specifying two points.

   The dimension line displays dynamically.

4. Position the dimension line. You can place the dimension line either at the specified rotation angle or perpendicular to this direction.

To create baseline dimensions

1. Do one of the following:

   - Click the **Baseline** tool button ( ) on the **Dimensions** toolbar.
   - Choose **Baseline** in the **Dimensions** menu.
   - Type **dimbaseline** in the command bar, then press Enter.

   You are prompted: Baseline: ENTER to select starting dimension/<Origin of next extension line>: A baseline dimension displays dynamically starting from the most recently added linear or angular dimension (if any).

2. (option) Press enter, then select the starting dimension (see note below).

3. Specify the origin of the next extension line. **You are prompted:** Baseline: ENTER to select starting dimension/<Origin of next extension line>:

4. Do one of the following:

   - Repeat steps 2 and 3.
   - Press the Esc key to stop creating baseline dimensions.

NOTES

- When selecting the starting dimension make sure to click the first extension line or the first half of the dimension line in order to use the first origin point of the starting dimension as the origin point for the baseline dimension.
- If you select an angular dimension, stacked angular dimensions are created.
- The distance between the stacked dimension lines is controlled by the **Dim baseline spacing** setting in the dimension style.

To create continued dimensions

1. Do one of the following:
   - Click the **Continue** tool button ( ) on the **Dimensions** toolbar.
   - Choose **Continue** in the **Dimensions** menu.
   - Type **dimcontinue** in the command bar, then press Enter.

   You are prompted: Continue: ENTER to select starting dimension/<Origin of next extension line>: A continued dimension displays dynamically starting from the most recently added linear or angular dimension (if any).
2. (option) Press enter, then select the starting dimension (see note below).
3. Specify the origin of the next extension line. **You are prompted:** Continue: ENTER to select starting dimension/<Origin of next extension line>:
4. Do one of the following:
   - Repeat steps 2 and 3.
   - Press the Esc key to stop creating continued dimensions.

NOTE

- When selecting the starting dimension make sure to click the second extension line or the second half of the dimension line in order to use the second origin point of the starting dimension as the origin point for the continued dimension. Otherwise the new dimension line will partly overlap the starting dimension.
- If you select an angular dimension, continued angular dimensions are created.

To create an arc length dimension

1. Do one of the following:
   - Click the **Arc** tool button ( ) on the **Dimensions** toolbar.
   - Choose **Arc** in the **Dimensions** menu.
   - Type **dimarc** in the command bar, then press Enter.

   You are prompted: Select arc or polyline arc segment:
2. Select the arc or polyline arc segment. The arc length dimension displays dynamically.
3. Position the dimension line.

1.21.9 Creating angular dimensions

Command : DIMANGULAR (6.11.4) Angular dimensions annotate the angle measured between two lines, two linear segments of a polyline or the angle of a circular arc. You can also dimension an angle by selecting an angle vertex and two endpoints. Starting from an existing angular dimension, you can add a baseline dimension (1.21.8) or a continued dimension (1.21.8). An angular baseline dimension inserts an additional dimension from a common first extension line origin. An angular continued dimension continues an angular dimension from the second extension line of a previous angular dimension.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e2448/images/staticbricsyscom/help/en_US/Draw/AngularDimensions/AngularDimensionMeasuredBetweenTwoLines.png} p. 988) angular dimension measured between two lines	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en_US/Draw/AngularDimensions/AngularDimensionOfAnArc.png} p. 988) angular dimension of an arc	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en_US/Draw/AngularDimensions/AngularDimensionDefinedByAVertexAndTwoEndpoints.png} p. 988) angular dimension defined by a vertex and two endpoints

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en_US/Draw/AngularDimensions/AngularBaselineDimensions.png} p. 988) angular baseline dimensions (1.21.8)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en_US/Draw/AngularDimensions/AngularContinuedDimensions.png} p. 988) angular continued dimensions (1.21.8)

To dimension an angle between two lines

1. Do one of the following

- Click the **Angular** tool button ( ) on the **Dimensions** toolbar.
- Choose **Angular** in the **Dimensions** menu.
- Type **dimangular** in the command bar, then press Enter.

You are prompted: Press Enter to specify angle/<select line, arc, or circle>: 2. Select the first line or line segment. You are prompted: Other line for angular dimension: 3. Select the second line or line segment. The angular dimension displays dynamically. You are prompted: Angle/Text/<Location of dimension arc>: 4. Do one of the following:

- Position the dimension arc.
- Type **A** and press Enter to define the angle of the dimension text, then position the dimension arc.
- Type **T** and press Enter to modify the dimension text, then position the dimension arc.

To dimension an angle encompassed by an arc

1. Do one of the following

- Click the **Angular** tool button ( ) on the **Dimensions** toolbar.
- Choose **Angular** in the **Dimensions** menu.
- Type **dimangular** in the command bar, then press Enter.

You are prompted: Press Enter to specify angle/<select line, arc, or circle>: 2. Select the arc. The angular dimension displays dynamically. You are prompted: Angle/Text/<Location of dimension arc>: 3. Do one of the following:

- Position the dimension arc.
- Type **A** and press Enter to define the angle of the dimension text, then position the dimension arc.
- Type **T** and press Enter to modify the dimension text, then position the dimension arc.

To create an angular dimension defined by a vertex and two endpoints

1. Do one of the following

- Click the **Angular** tool button ( ) on the **Dimensions** toolbar.
- Choose **Angular** in the **Dimensions** menu.
- Type **dimangular** in the command bar, then press Enter.

You are prompted: Press Enter to specify angle/<select line, arc, or circle>: 2. Right click or press Enter. You are prompted: Vertex of angle. 3. Specify the vertex of the angle. You are prompted: first side of angle. 4. Specify the endpoint of the first side of the angle. You are prompted: Other side of angle. 5. Specify the endpoint of the second side of the angle. The angular dimension displays dynamically. You are prompted: Angle/Text/<Location of dimension arc>: 6. Do one of the following:

- Position the dimension arc.
- Type **A** and press Enter to define the angle of the dimension text, then position the dimension arc.
- Type **T** and press Enter to modify the dimension text, then position the dimension arc.

Figure 1.873

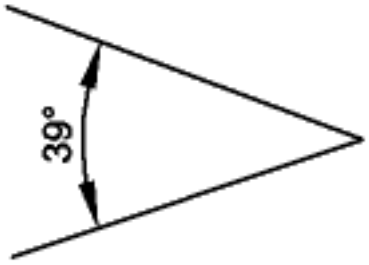


Figure 1.874

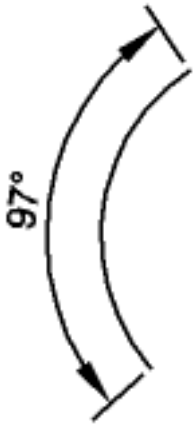


Figure 1.875

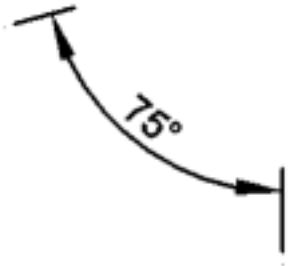


Figure 1.876

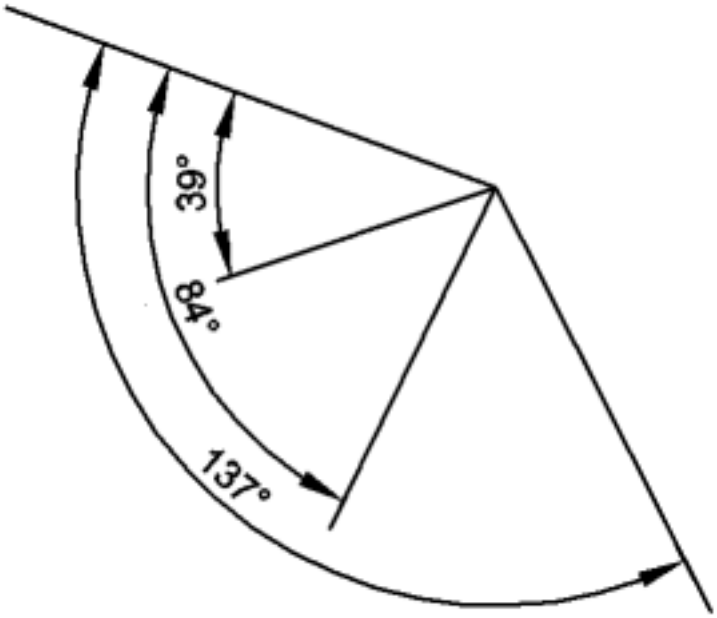
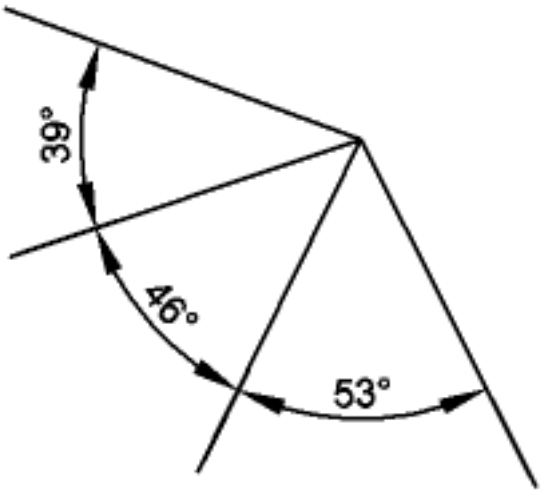


Figure 1.877



1.21.10 Creating diametrical and radial dimensions

Commands: DIMDIAMETER (6.11.10) and DIMRADIUS (6.11.17) Diametrical and radial dimensions annotate the radii and diameters of circles and circular arcs. The DIMCENTER (6.11.8) command creates center marks in the center point of circles and circular arcs.

To create a diametrical dimension

1. Do one of the following

   - Click the **Diameter** tool button ( ) on **Circles** toolbar or the **Circles** flyout of the **Dimensions** toolbar.
   - Choose **Diameter** in the **Dimensions** menu.
   - Type **dimdiameter** in the command bar, then press Enter.

   You are prompted: Select arc or circle

2. Select the arc or circle you want to annotate. The diametrical dimension displays dynamically.

3. Position the dimension line.

To create a radial dimension

1. Do one of the following

   - Click the **Radius** tool button ( ) on **Circles** toolbar or the **Circles** flyout of the **Dimensions** toolbar.
   - Choose **Radius** in the **Dimensions** menu.
   - Type **dimradius** in the command bar, then press Enter.

   You are prompted: Select arc or circle

2. Select the arc or circle you want to annotate. The radial dimension displays dynamically.

3. Position the dimension line.

Creating center marks or center lines

1. Do one of the following

   - Click the **Center** tool button ( ) on **Circles** toolbar or the **Circles** flyout of the **Dimensions** toolbar.
   - Choose **Center** in the **Dimensions** menu.
   - Type **dimcenter** in the command bar, then press Enter.

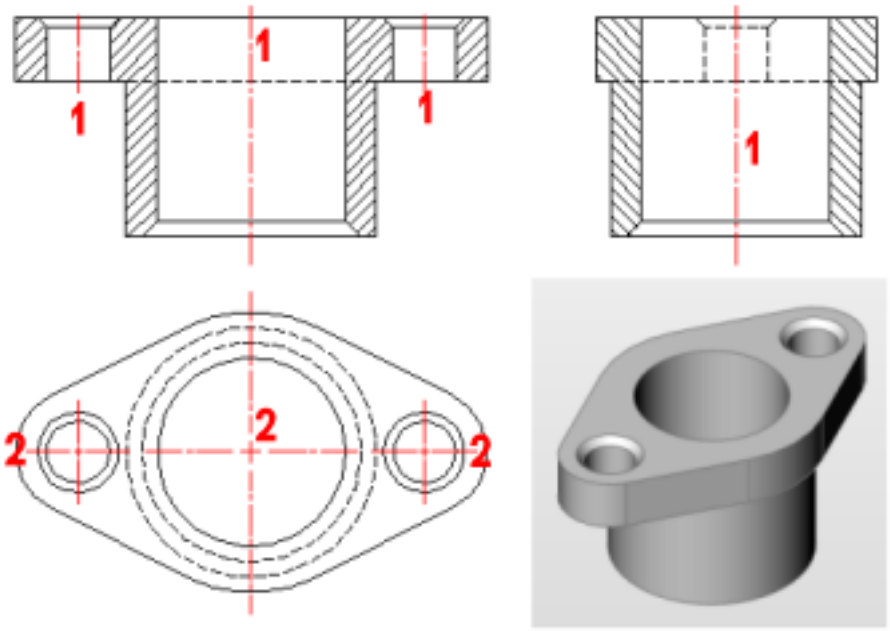
   You are prompted: Select arc or circle to dimension:

2. Select the arc or circle. The center marks are created.

NOTE

- The **Center mark** setting of the dimension style controls whether center marks or center lines are created. The **Center mark size** settings of the dimension style controls the size of the center mark.
- The center mark settings are saved to the **DIMCEN** system variable.

Figure 1.878



1.21.11 Centerlines and Center Marks

Commands: CENTERLINE (6.9.5), CENTERMARK (6.9.6), CENTERREASSOCIATE (6.9.7), CENTERDISASSOCIATE (6.9.4), CENTERRESET (6.9.8) **CenterLine** command: creates a centerline entity associated with two selected lines. **CenterMark** command: creates a center mark entity associated with a selected circle or arc. **CenterReassociate** command: recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc. **CenterDisassociate** command: breaks the association of centerline and a center mark entities. **CenteReset** command: resets centerline and center mark entities. Centerlines and Center Marks are drawing reference entities to indicate axes of symmetry and to centers of circular openings. Centerlines and Center Marks are associative entities: when the referenced lines, circles or circular arcs are modified they are adjusted automatically. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06Wo

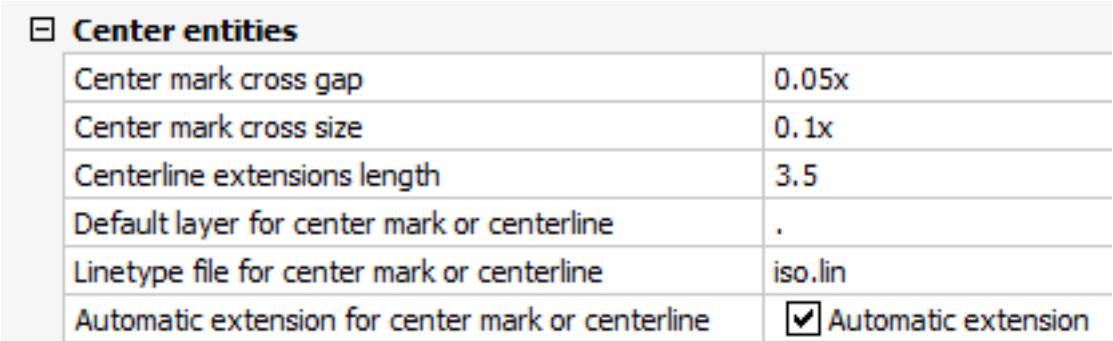
1. Centerline
2. Center Mark

System variables for center entities

The properties and geometry of centerline and center mark entities are controlled through system variables. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06WorkingwithDimensio

Name	Title	Description

Figure 1.879



CENTERCROSSGAP	Center mark cross gap	Defines the gap between the center mark and its centerlines. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 Type **centercrossgap** in the command line. **You are prompted:** New value for CENTERCROSSGAP, or . for none/<"0.05x">: • Type a value to specify the absolute size of the gap in drawing units. • Type a value followed by x to set the size of the gap relative to the diameter of the circle or circular arc. The default value 0.05x creates a gap of 5/100 of the diameter. • . (dot) creates no gap.
CENTERCROSSSIZE	Center mark cross size	Defines the size of the associative center mark. Type **centercrosssize** in the command line. **You are prompted:** New value for CENTERCROSSSIZE, or . for none/<"0.1x">: • Type a value to specify the absolute size of the center mark lines in drawing units. • Type a value followed by x to set the size of the center mark relative to the diameter of the circle or circular arc. The default value 0.1x creates center marks 1/10th of the diameter. • . (dot) creates center marks with zero length lines.
CENTEREXE	Centerline extensions length	Defines the length of the centerline extensions. The value is expressed in drawing units. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1

CENTERLAYER	Default layer for center mark or centerline	Specifies the default layer for new centerlines and center marks. Type **centerlayer** in the command line. **You are prompted:** New value for CENTERLAYER, or . for none/<"">: • Type a name. If such layer not yet exists, the layer will be created when a centerline or centemark is created. • Type . (dot) to create centerlines and center marks on the current layer.
CENTERLTYPEFILE	Linetype file for center mark or centerline	Specifies the linetype file used to create centerlines and center marks. Default in imperial unit drawings: **default.lin** Default in metric unit drawings: **iso.lin**
CENTERLTYPE	Linetype for center mark or centerline	Specifies the linetype for newly created centerlines and center marks. Type **centerltype** in the command line. **You are prompted:** New value for CENTERLTYPE, or . for none/<"CENTER2">: • Type a new linetype name or press Enter to accept the current name. • Type . (dot) for continuous lines.
CENTERLTSCALE	Linetype scale for center mark or centerline	Specifies the linetype scale for newly created centerlines and center marks. Type
CENTERMARKEXE	Automatic extension for center mark or centerline	Determines whether centerlines extend (left) or not (right) from new center marks. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1

Figure 1.880



Figure 1.881

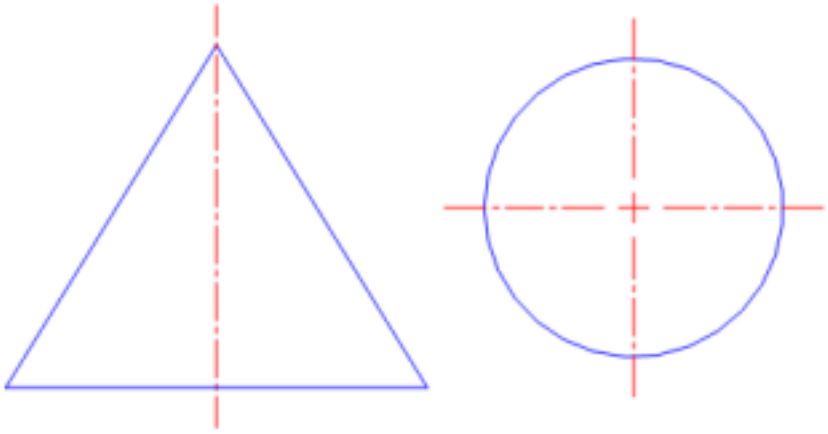
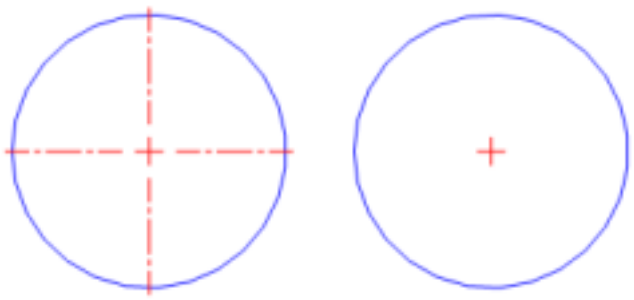


Figure 1.882



1.21.12 Creating ordinate dimensions

Command: DIMORDINATE (6.11.15) An ordinate dimension annotates the perpendicular distance from an origin or base point (the origin of the current user coordinate system (1.15.18) [UCS]). Ordinate dimensions consist of an x- or y-coordinate and a leader. An x-ordinate dimension measures distances along the x-axis, while a y-ordinate dimension measures distances along the y-axis. As you select ordinate points, the program automatically determines whether the point is an x- or y-ordinate based on which direction you drag the second point. You can also specify whether the ordinate represents an x- or y-ordinate. Ordinate dimension text is always aligned with the ordinate leader lines, regardless of the text orientation specified by the current dimension style.

To create an ordinate dimension

1. Do one of the following:

   - Click the **Ordinate** tool button ( ) on the **Dimensions** toolbar.
   - Choose **Ordinate** in the **Dimensions** menu.
   - Type **dimordinate** in the command bar, then press Enter.

   You are prompted: Select point for ordinate dimension:

2. Specify the origin point for the ordinate dimension. The ordinate dimension displays dynamically.
3. Position the ordinate dimension.

NOTE	Adjust the origin point of the current user coordinate system (1.15.18) (UCS) before placing ordinate dimensions if necessary.

1.21.13 Creating leaders and annotations

Commands: DIMLEADER (6.11.13), LEADER (6.20.20) and QLEADER (6.25.1) A leader consists of an arrowhead, a line, polyline or a spline and an annotation. The general procedure to place an annotation is:

- Specify the position of the arrowhead.
- Create the leader line segments.
- Type the annotation text.

To create a leader and an annotation

1. Do one of the following:

   - Click the **Leader** tool button ( ) on the **Dimensions** toolbar.
   - Choose **Leader** in the **Dimensions** menu.
   - Type **leader** or **dimleader** in the command bar, then press Enter.

   You are prompted: Start of leader

2. Specify the position of the arrowhead. **You are prompted:** Next point.

3. Specify the endpoint of the first leader segment. **You are prompted:** To point: Format/Undo/<Annotation>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Drawi

4. Add more segments to the leader, then press Enter or right click to stop adding segments. **You are prompted:** First line of annotation text/<options>:

5. Do one of the following:

   - Type the first line of the annotation text, then press Enter. **You are prompted:** Next line of annotation text: Do one of the following:
     - Press Enter or right click to create the leader and annotation.
     - Keep adding lines to the annotation text, then press Enter or right click to create the leader and annotation.
     - Press Enter or right click to see the annotation text options. **You are prompted:** Dimension text options: Block/Copy/None/Tolerance/<Mtext>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06WorkingwithDimension

   - Press the Esc key to abort the **Leader** tool.

Figure 1.883

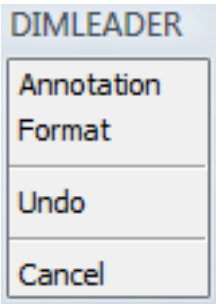
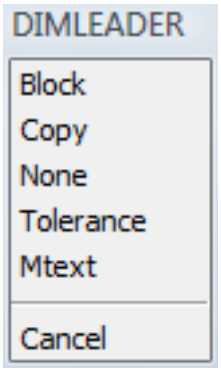


Figure 1.884



To create a leader only

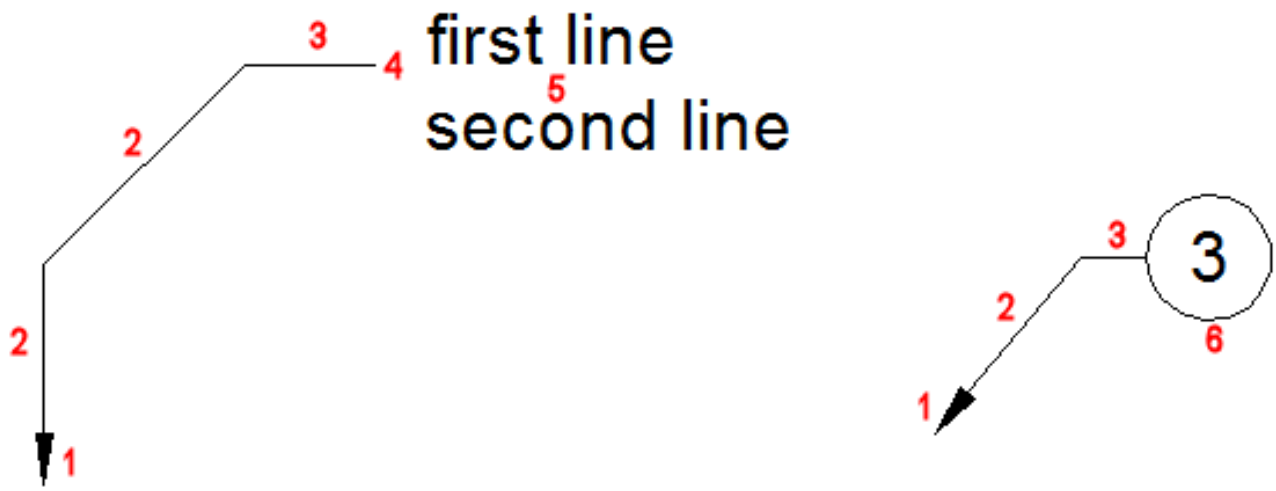
1. Do one of the following:

   - Click the **Leader** tool button ( ) on the **Dimensions** toolbar.
   - Choose **Leader** in the **Dimensions** menu.
   - Type **dimleader** in the command bar, then press Enter.

   You are prompted: Start of leader

2. Specify the position of the arrowhead. **You are prompted:** Next point.
3. Specify the endpoint of the first leader segment. **You are prompted:** To point: Format/Undo/<Annotation>:
4. Add more segments to the leader, then press Enter or right click to stop adding segments. **You are prompted:** First line of annotation text/<options>:
5. Press Enter, then choose **None** in the prompt menu or type **N** .

Figure 1.885



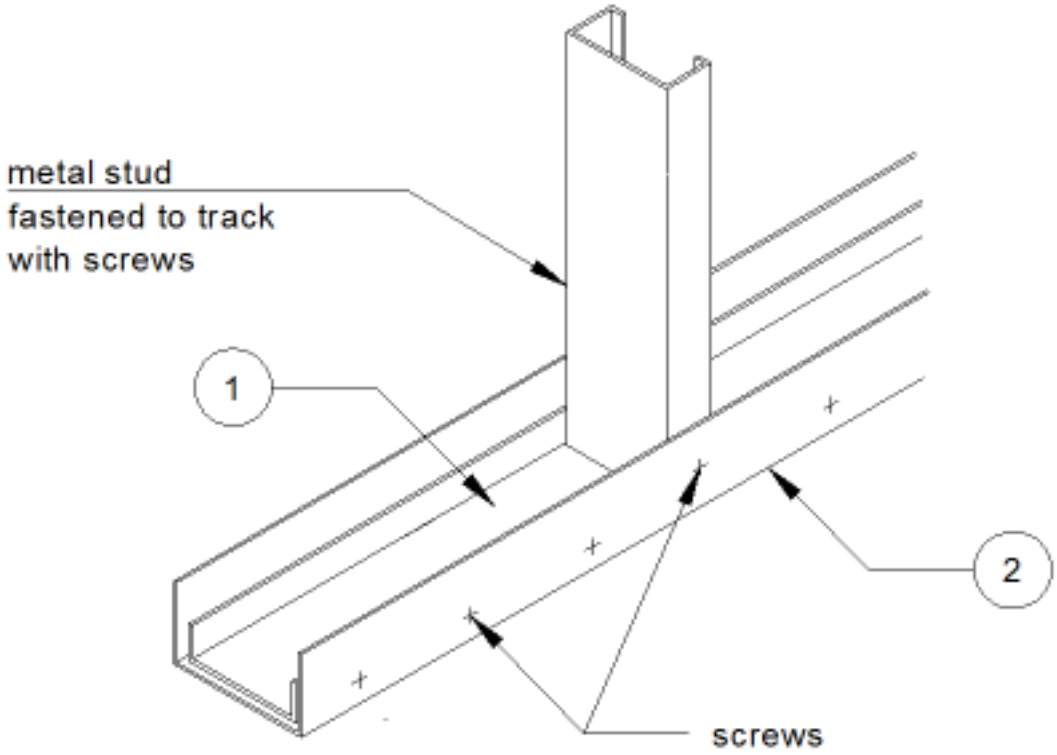
1.21.14 Working with Multileaders

Commands: MLEADER (6.21.20), MLEADERSTYLE (6.21.25), MLEADEREDIT (6.21.23) and MLEADEREDITEXT (6.21.24)
A multileader consists of:

- an arrowhead (1)
- one or more leader lines (2), which can be straight or curved
- optionally a horizontal landing (3) and landing gap (4)
- content: either a multiline text (5) or a block (6)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06WorkingwithDin
Multileaders are created using the current multileader style. Each drawing contains a **Default** leader style, which cannot be renamed or deleted; the **Default** leader style can be customized though. The **MLeader** command allows to define overrides on the current multileader style. Multileaders can be created arrowhead first, landing first or content first. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06Worki

Figure 1.886



To create a multileader

1. (option) Launch the **MLeaderStyle** command, then make a multileader style current.

2. Do one of the following



- Click the **Multileader** tool button ( ) on the **Dimensions** toolbar.

Figure 1.887

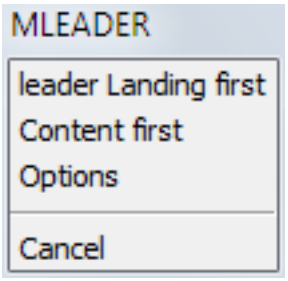
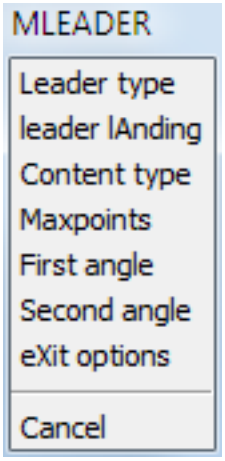


Figure 1.888



- Choose **Multileader** in the **Dimensions** menu.
- Type **multileader** in the command bar, then press Enter.

You are prompted: Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source

3. (option) Select a placement option:

   - arrowhead first
   - landing first
   - content first.

   The most recently used placement option is repeated. The default sequence is: arrowhead - landing - content.

4. (option) Choose **Options** in the prompt menu or type **O** , then press Enter, if you want to define overrides on the current multileader style. **You are prompted:** Enter an option [Leader type/leader lAnding/Content type/Maxpoints/First angle/Second angle/eXit options] <eXit options>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06WorkingwithDimensions/ART/14

5. Specify a point to define the leader arrowhead location. **You are prompted:** Specify leader landing location:

6. Specify a point to define the leader landing location. Depending on the content type:

   - **MText** : The MText (6.21.35) command is launched.
   - **Block** : A block is inserted.
   - **None** : The command is concluded.

Adding vertices to a leader line of a multileader

1. If necessary press the **F12** function key to activate the Quad cursor menu (1.12.5).
2. Pause the cursor over the leader line you want to add a vertex to. The Quad cursor menu displays, showing the most recently used command (*).
3. Move the cursor to the Quad cursor menu and expand the **2D Editing** command group.

4. Click the **Add Vertex** tool ( ). A new vertex is added dynamically.
5. Specify the position of the vertex.
6. (option) Pause the cursor of a leader line, then right click to add another vertex.

(*) Right click to repeat the most recently used Quad command.

Removing vertices from a leader line of a multileader

1. If necessary press the **F12** function key to activate theQuad cursor menu (1.12.5).
2. Pause the cursor over the leader line close to the vertex to be removed. The Quad cursor menu displays, showing the most recently used command(*).
3. Move the cursor to the Quad cursor menu and expand the **2D Editing** command group.
4. Click the **Remove Vertex** tool ( ). The vertex is removed.
5. (option) Pause the cursor of a leader line close to a vertex, then right click to remove the vertex.

Adding leader lines to a multileader

1. If necessary press the **F12** function key to activate theQuad cursor menu (1.12.5).
2. Pause the cursor over a leader line of the multileader entity. The Quad cursor menu displays, showing the most recently used command(*).
3. Move the cursor to the Quad cursor menu and expand the **2D Editing** command group.
4. Click the **Add Leader** tool ( ). A new leader line displays dynamically.
5. Specify the origin of the new leader line. Another new leader line displays dynamically.
6. Do one of the following:
   - Specify the origin of the new leader line.
   - Right click or press Enter to stop adding leader lines.

(*) Right click to repeat the most recently used Quad command.

Removing leader lines from a multileader

1. If necessary press the **F12** function key to activate theQuad cursor menu (1.12.5).
2. Pause the cursor over the leader line you want to remove. The Quad cursor menu displays, showing the most recently used command(*).
3. Move the cursor to the Quad cursor menu and expand the **2D Editing** command group.
4. Click the **Remove Leader** tool ( ). The leader line is removed.

(*) Right click to repeat the most recently used Quad command.

1.21.15 Editing dimensions

Commands:DIMEDIT, (6.11.12)DIMTEDIT (6.11.23) Dimension entities can be edited in the following ways:

- Use grips to edit dimensions
- Edit the dimension text
- Rotate dimension text
- Reposition the dimension text
- Restore the text position
- Make the extension lines oblique

Using grips to edit a linear dimension

1. Select the dimension entity. 5 grips display:
   - 2 origin point grips (1)
   - 2 dimension line grips (2)
   - 1 dimension text grip (3)

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06Workingw
2. (option) Drag the origin point grips to adjust the dimension origin points. The dimension text is adjusted automatically.
3. (option) Drag dimension line grips to move the dimension line.
4. (option) Drag the text grip to move the dimension text.

NOTES

- Depending on the **Text movement** setting, the dimension line will move along with the text or the text will move independently. A leader will be drawn if the **Text movement** setting is set to **Move text, add leader** .
- If two adjacent dimensions (continued dimensions) are selected both, you can move their coinciding grips simultaneously.

Figure 1.889



Editing the dimension text

Dimension text can be edited:

- in the MText editor:
  - using the DDEDIT (6.10.14) command
  - double clicking the dimension entity
- in the **Text override** field in the Properties Bar (1.12.4).
- using the **Edit Dimension Text** tool.

NOTES

The following tags can be used to format the dimension text:

- type **\L** (backslash upper case L) to start underlined text, **\l** (backslash lower case L) to stop underlined text
- type **<>** to add the default dimension text
- type **[]** to add the alternate units, if alternate units is currently switched off in the dimension style.
- type **%%d** to add the degree symbol (°)
- type **%%c** to add the diameter symbol ()
- type **%%p** to add the plus/minus symbol ()

e.g. if you type **%%c\L<> cm** in the **Text override** field, dimension text displays as:  24,43 cm

To edit the dimension text in the MText editor

1. Double click the dimension entity. The **Text Formatting** toolbar displays.
2. Edit the dimension text.
3. Do one of the following:
   - Click the **OK** button on the **Text Formatting** toolbar.
   - Click outside the dimension text.

To edit the dimension text in the Properties Bar

1. Select the dimension entity.
2. In the Properties Bar (1.12.4), select the **Text override** field.
3. Type the new dimension text in the **Text override** field and press Enter.
4. Press the Esc key to stop editing the dimension entity.

To edit the dimension text using the Edit Dimension Text tool

1. Do one of the following:
   - Click the **Edit Dimension Text** tool () in the **Dimensions** toolbar.
   - Choose **Edit Dimension Text** in the **Dimensions** menu.
   - Type **Dimedit** in the command bar, press Enter, then type **new** and press Enter.

   You are prompted: New dimension text <>:

2. Type the new dimension text and press Enter. **You are prompted:** Select dimensions to replace with new text:
3. Select the dimensions (1.23.6)and press Enter.

To rotate the dimension text

1. Do one of the following:

   - Click the **Rotate Dimension Text** tool () in the **Dimensions** toolbar.
   - Choose **Rotate Dimension Text** in the **Dimensions** menu.
   - Type **Dimedit** in the command bar, press Enter, then type **R** and press Enter.

   You are prompted: Angle of dimension text:
2. Type the rotation angle, then press Enter. **You are prompted:** Select entities to rotate text:
3. Select the dimensions (1.23.6)and press enter.

To reposition the dimension text

1. Do one of the following:

   - Click the **Reposition Dimension Text** tool () in the **Dimensions** toolbar.
   - Choose **Reposition Dimension Text** in the **Dimensions** menu.
   - Type **Dimtedit** in the command bar, then press Enter.

   You are prompted: Select dimensions to reposition text:
2. Select the dimension. The dimension text moves dynamically.
3. Click to reposition the dimension text.

To restore the text position

1. Do one of the following:

   - Click the **Restore Text Position** tool () in the **Dimensions** toolbar.
   - Choose **Restore Text Position** in the **Dimensions** menu.
   - Type **Dimedit** in the command bar, then press Enter.

   You are prompted: Select dimensions to restore text to default position:
2. Select the dimensions (1.23.6)and press enter.

To make the extension lines oblique

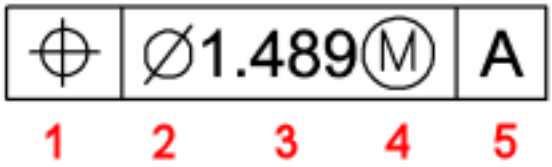
1. Do one of the following:

   - Click the **Make Oblique** tool () in the **Dimensions** toolbar.
   - Choose **Make Oblique** in the **Dimensions** menu.
   - Type **Dimedit** in the command bar, press Enter, then type **O** and press Enter.

   You are prompted: Select dimensions to make oblique:
2. Select the dimensions (1.23.6)and press enter. **You are prompted:** Enter obliquing angle:
3. Type the obliquing angle, then press Enter.

NOTE	To restore non-rotated extension lines, execute the above procedure and type 0 (zero) when asked to enter the obliquing angle.

Figure 1.890



1.21.16 Adding Geometric Tolerances

Command: TOLERANCE (6.29.17) Geometric tolerances specify the maximum variation that is allowed in form or position from true geometry. The geometric tolerance is, in essence, the tolerance zone within which a surface or axis of hole or cylinder can lie which results in resulting feature being acceptable for proper function and interchangeability. BricsCAD draws geometric tolerances using a feature control frame, which is a rectangle divided into compartments. Each feature control frame consists of at least two compartments. The first compartment contains a geometric tolerance symbol that indicates the geometric characteristic to which the tolerance is applied, such as location, orientation, or form. For example, a form tolerance may indicate the flatness or roundness of a surface. The second compartment contains the tolerance value. When appropriate, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol. The material conditions apply to features that can vary in size. The tolerance value can then be followed by primary, secondary, and tertiary datum reference letters, along with the material conditions of each datum. Datum reference letters are generally used as reference tolerances to one of up to three perpendicular planes from which a measurement is made, although datum reference letters can also indicate an exact point or axis. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/06WorkingwithDimensions/ART/11Geom

1. Geometric characteristic symbol
2. Diameter symbol
3. Tolerance value
4. Material condition symbol
5. Datum reference

When two tolerances apply to the same geometry, you can also add a composite tolerance consisting of a primary tolerance value followed by a secondary tolerance value. To make a tolerance even more specific, it can also contain a projected tolerance consisting of a height value followed by a projected tolerance symbol. For example, you can use a projected tolerance to indicate the perpendicularity of an embedded part.

To create a geometrical tolerance frame

1. Do one of the following

   - Click the **Tolerance** tool button ( ) in the **Dimensions** toolbar.
   - Choose **Tolerance** in the **Dimensions** menu.
   - Type **tolerance** in the command bar, then press Enter.

   The Geometric Tolerance dialog (1.21.16) opens.

2. Compose the geometrical tolerance frame in the **Geometric Tolerance** dialog.
3. Click the **OK** button. The **Geometric Tolerance** dialog closes.
4. Click to insert the geometric tolerance frame in the drawing.

Geometric Tolerance dialog (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Draw

1. **Geometric Tolerance Symbol** buttons. Click to open the **Symbol** palette. (\vref {fig:C:/Dev/Task-2448/images/staticbrics
2. **Diameter Symbol** option buttons. Click to add the diameter symbol.
3. **Material Condition** buttons. Click to open the **Material Condition** palette. (\vref {fig:C:/Dev/Task-2448/images/staticbric

Figure 1.891

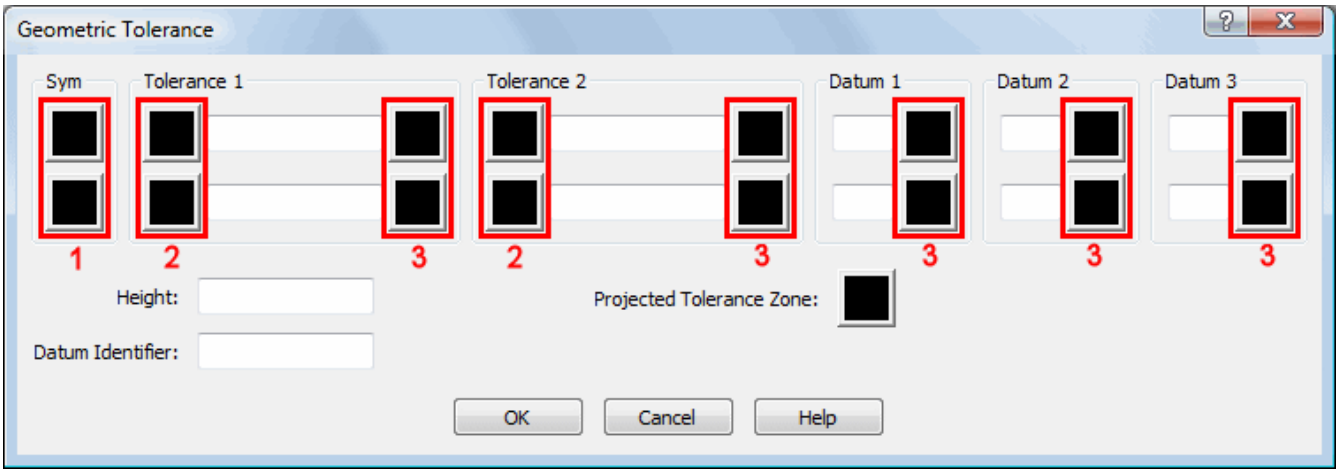


Figure 1.892

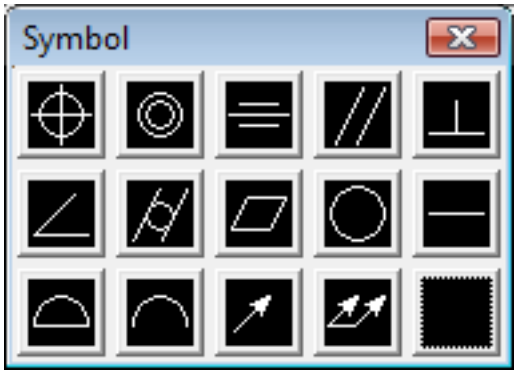


Figure 1.893

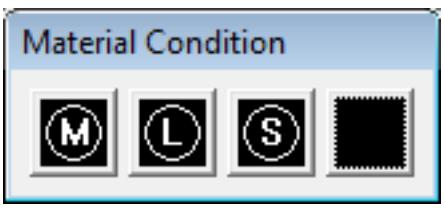


Figure 1.894



1.21.17 Text Variables

System variables can be set using the Settings dialog (1.13.11). Some system variables which apply to text entities are:

Variable Name	Variable Title	Description
FONTALT	Alternate font	Defines the font which will be used if a text font cannot be found.
HIDETEXT	Hide text on hide	Specifies whether the HIDE (6.16.10) command processes text entities.
MIRRTEXT	Mirror text	Determines whether text is mirrored by the MIRROR command or not.
MTEXTED	Multiline text editor	Sets the primary and secondary text editors to use for multiline text entities.
MTEXTFIXED	Multiline text fixed	Controls whether BricsCAD zooms, rotates and/or pans the view to fit the Mtext being edited.
QTEXTMODE	Quick text mode	Controls how text entities are displayed. If ON, displays text as a box.
TEXTANGLE	Text angle	Stores the angle of the last added text entity.
TEXTFILL	Text fill	Determines whether True Type fonts display either filled or outlined. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
TEXTQLTY	Text quality	Defines the smoothness of True Type fonts for plotting and rendering.
TEXTSIZE	Text size	Sets the default height for new text entities. TEXTSIZE has no effect if the current text style has a fixed height
TEXTSTYLE	text style	Stores the name of the current text style.
TSPACEFAC	Text space factor	Specifies the line spacing distance of multiline text measured as factor of the text height. Values between 0.25 and 4 are accepted.
TSPACETYPE	Text space type	Specifies the type of line spacing used for multiline text. At least: adjusts line spacing based on the tallest character(s) in a line. Exactly: used the specified line spacing, regardless of in individual character sizes.
TSTACKALIGN	Text stack align	Determines the vertical alignment of stacked text: bottom, center or top.
TSTACKSIZE	Text stack size	Specifies the percentage a which the stacked text fraction height is relative to the current height of the selected text. Values between 25 and 125 are accepted.

1.21.18 Working with text styles

Command: STYLE (6.27.53) When you add text to a drawing, it uses the current text style, which determines the font, size, angle, orientation and other properties of the new text entity. Every drawing has a default text style, named **Standard** , which initially uses the **Arial** font. You cannot delete or rename the **Standard** style, but you can modify it. If you change the **font** or **orientation** of an existing style, all existing text entities created using that style are automatically updated to reflect the change. Changing any other property has no effect on existing text. You can create and use an unlimited number of additional text styles. Annotative text styles create annotative text entities (1.21.30). In an annotative text style the **Height** defines the paper space height of the text; the model space height depends on the current annotation scale as defined by the CANNOSCALE and CANNOSCALEVALUE system variables.

To open the Text Style Explorer dialog

Do one of the following:

- In the **Tools > Drawing Explorer** menu choose **Text Styles ...**
- Double click the **Style** field in the Status Bar (1.12.2).
- Type style (6.27.53) in the command bar and press Enter.

The **Drawing Explorer - Text Styles** dialog opens.

To create a text style

1. Open the Text Style Explorer dialog.
2. Do one of the following:
   - Click the **New** tool button ( ) in the **Details** section of the **Drawing Explorer** dialog.
   - Select an existing style, then right click and choose **New** in the context menu.

   A **NewStyle** text style is created, inheriting the properties of the currently selected style.
3. Type a name in the **Text Style Name** field, replacing the **NewStyle** default name.
4. Define the text style properties.
5. Close the **Text Style Explorer** dialog.

To edit a text style

1. Open the text style explorer dialog.
2. Select the text style.
3. Define the text style properties.
4. Close the **Text Style Explorer** dialog.

To make a text style current

1. Open the text style explorer dialog.
2. Do one of the following:
   - When in **Detail view** ( ), click the blank tile in front of the text style name. A blue filled circle indicates the text style is set current.
   - When in **Detail view** ( ) or **Icon view** ( ), right click then choose **Set current** in the context menu.

NOTE

- Alternatively you can right click the **Text style** field in the Status Bar (1.12.2), then choose the new current style. The current style is marked in the style list.
- When you use the **Style** option in the create text procedures, the selected style will become the current text style.

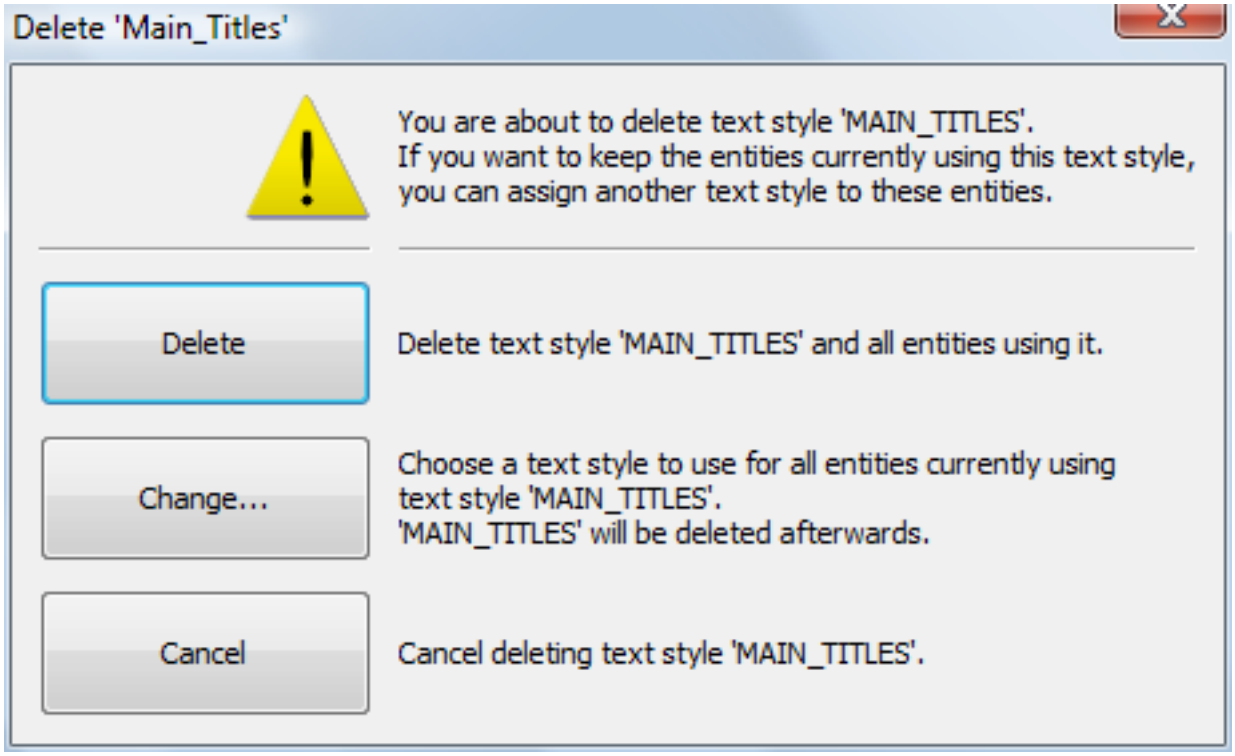
To delete a text style

1. Open the text style explorer dialog.
2. Select the text style.
3. Do one of the following:
   - Click the **Delete** tool button ( ) in the **Details** section of the **Drawing Explorer** dialog.
   - Right click, then choose **Delete** in the context menu.

   If the text style is not used in the drawing or in a dimension style, the text style is deleted, else an alert box displays: (\vref{fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05Wo
4. Do one of the following:
   - Click the **Delete** button to delete the text style and all entities using it.
   - Click the **Change...** button to delete the text style and assign a different text style to the entities, which currently use the selected style.
   - Click the **Cancel** button to cancel the delete procedure.

NOTE It is not possible to delete the current text style.

Figure 1.895


Delete 'Main_Titles'
You are about to delete text style 'MAIN_TITLES'.
If you want to keep the entities currently using this text style, you can assign another text style to these entities.
Delete
Delete text style 'MAIN_TITLES' and all entities using it.
Change...
Choose a text style to use for all entities currently using text style 'MAIN_TITLES'.
'MAIN_TITLES' will be deleted afterwards.
Cancel
Cancel deleting text style 'MAIN_TITLES'.

1.21.19 Editing text

Command: DDEDIT (6.10.14) You can edit and modify a text entity as you would any other drawing entity. That is, you can delete, move, rotate, and scale text. You can change the text color or layer in the Properties Bar (1.12.4). To edit a multiline text, you can use the built-in text editor or choose an external program, such as Wordpad. The MTEXTFIXED (1.13.11) system variable controls whether the view is zoomed, rotated and/or panned to fit the edited Mtext. The TEXTED (1.13.11) setting controls the text edit behavior:

- 0 = in-place editor
- 1 = pop up dialog
- 2 = in-place editor with repeated input

To edit a text entity

1. Do one of the following:
   - Type **DDEDIT** in the command bar. **You are prompted:** Select entity to modify:
   - Double click the text entity. Then go to step 3.
2. Click a single line text (1.21.20) entity. Depending on the value of the TEXTED system variable: TEXTED = 1: The **Edit Text** window opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10 TEXTED = 0 or 2: The text is edited 'in place'. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Usr Click a multiline text (1.21.20) entity The **Text formatting** toolbar opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
3. Edit the text.
4. Press the **OK** button on the **Edit Text** window or **Text Formatting** toolbar to apply the changes.
5. Do one of the following:
   - Select another text entity to edit.
   - Right click or press Enter to stop editing text entities.

Figure 1.896

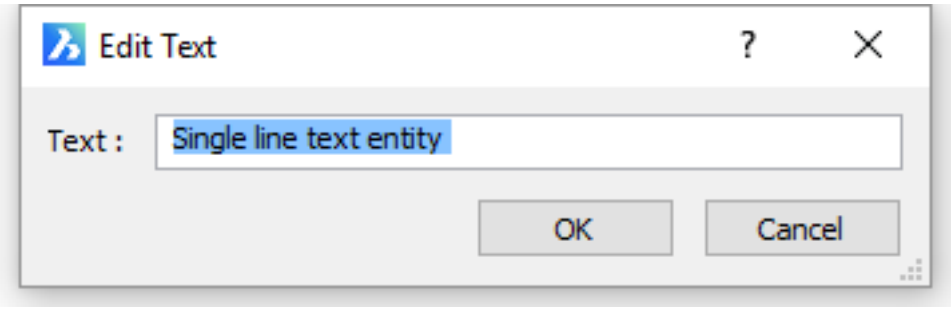


Figure 1.897

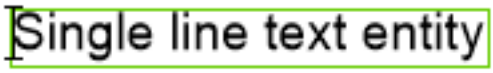
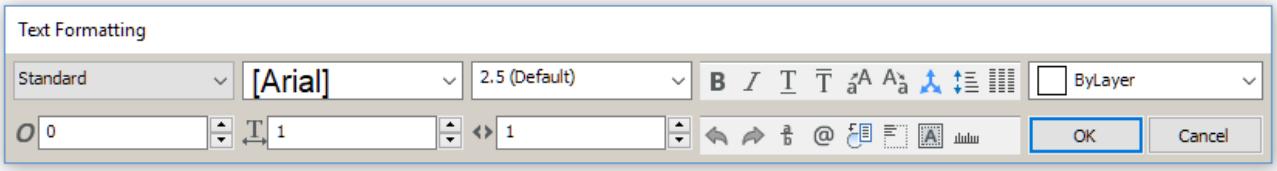


Figure 1.898



To edit a selection of Mtext entities

1. Select the Mtext entities.
2. In the Properties (1.12.4) bar select **Contents**.
3. Press the **Browse** button (...) on the right hand side of the **Contents** field (\vref {fig:C:/Dev/Task-2448/images/staticbrics
4. The **Text formatting** toolbar opens for the first Mtext entity in the selection.
5. Edit the Mtext.
6. Press the **OK** button or click in the drawing. The next Mtext entity in the selection can be edited now.
7. Repeat steps 5 and 6 until the last Mtext is edited.

Figure 1.899



To select a different multiline text editor

Edit the **MTEXTED** system variable under **Drawing/Drafting/Texts/Multiline Texts** in the Settings (1.13.11) dialog.
Or

- Type **MTEXTED** in the command bar, then press Enter. **You are prompted:** New value for MTEXTED, or . for none/<current value>:
- Type the file name and full path of the alternate text editor in the command bar, then press Enter. e.g. C:\Program Files\Windows NT\Accessories\Wordpad.EXE

NOTE	To restore the built-in text editor, type a . (period) in step 2 of the above procedure, then press Enter.

1.21.20 Creating text

Commands: TEXT (6.29.12) and MTEXT (6.21.35) The **Text** command creates single line text, while the **Mtext** command creates text entities which consist of multiple lines.

To create single line text

1. Do one of the following:

   - Click the **Text** tool button (A) on the **Draw** toolbar.
   - Choose **Text** in the **Text** submenu in the **Draw** menu.
   - Type **text** in the command bar, then press Enter.

   You are prompted: Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>:

2. Specify the insertion point of the text. **You are prompted:** Height of text <current height>:

3. Do one of the following:

   - Press Enter or right click to accept the current text height.
   - Type the new text height in the command bar, then press Enter.
   - Define the new text height graphically by specifying two points in the drawing.

   You are prompted: Rotation angle of text <current angle>

4. Do one of the following:

   - Press Enter or right click to accept the current angle.
   - Type the new angle in the command bar, then press Enter.
   - Define the now angle graphically by specifying two points in the drawing.

   You are prompted: Text:

5. Type the text in the command bar, then press Enter. The text is placed in the drawing. **You are prompted:** Text:

6. Do one of the following:

   - Repeat step 5 to create another text entity. The new text is placed below the previous one.
   - Press Enter or right click to conclude the **Text** command.

NOTE	If you have already created text, the most recent text entity highlights when you repeat the **Text** command. If you want to place the new text right below the previously created text, press Enter in step 2 of the above procedure.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/vref{fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en

Single line text justification

To create multiline text

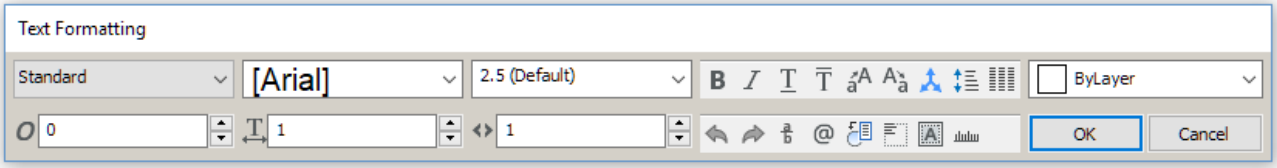
1. Do one of the following:

   - Click the **Multiline Text** tool button (A) on the **Draw** toolbar.
   - Choose **Multiline Text** in the **Text** submenu in the **Draw** menu.
   - Type **mtext** in the command bar, then press Enter.

   You are prompted: First corner for block of text:

2. Specify the first corner (insertion point of the multiline text). **You are prompted:** Justification/Rotation/Style/Height/Direc
   corner for block of text>:

Figure 1.900



3. Do one of the following:

   - Specify the opposite corner of the text block to define the maximum width of the multiline text.
   - Type **@** in the command bar, then press Enter for non-wrapped (zero width) text.
   - If Dynamic Dimensions (1.15.9) are active, the height and width of the text field display dynamically. Type the height and width in the dynamic dimension fields. Press the TAB key to switch; press Enter to confirm.

   The **Text formatting** toolbar opens right above the text block. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel

4. (option) Define the text properties.

5. Type your text.

6. To conclude the command and close the **Text formatting** toolbar do one of the following:

   - Click the **OK** button.
   - Press Ctrl+Enter.
   - Click outside the MText entity.

Button or setting	Name	Description
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/03StyleButton.pr	Text Style	Sets the text style
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/03FontButton.pn	Text Font	Sets the text font
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/03TextHeightFie	Text Height	Sets the text height
	Bold	Creates bold text
	Italic	Creates italic text
	Underscore	Creates underscored text
	Overscore	Creates overscored text
	Uppercase	Converts selected text to uppercase
	Lowercase	Converts selected text to lowercase
	Annotative	Toggles the Annotative property
	Line Spacing	Sets the line spacing
	Columns	Arranges the text in columns
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/03LayerField.png	Color	Sets the color
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/03ObliqueAngleF	Oblique Angle	Sets the oblique angle
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/03WidthFactorFi	Width Factor	Sets the width factor
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/03CharacterSpac	Character Spacing	Sets the character spacing
	Undo	Undoes recent actions

	Redo	Redoes previously undone actions
	Stacked Fraction	Creates stacked fractions
	Special Character	Inserts special characters
	Field	Inserts a variable field
	Justify	Sets the text justification
	Background Mask	Adds a colored background
	Ruler	Toggles the display of the ruler
OK	OK	Closes the **Text formatting** toolbar and applies the changes
Cancel	Cancel	Closes the **Text formatting** toolbar and leaves the Mtext entity unchanged

NOTE	If the line length exceeds the **Width** of the multiline text, the text will be wrapped. Set the **Width** to zero for non-wrapped multiline text.

Text formatting toolbar tools and settings

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e2448/images/staticbricsyscom/help/e2448/images/staticbricsyscom/help/e

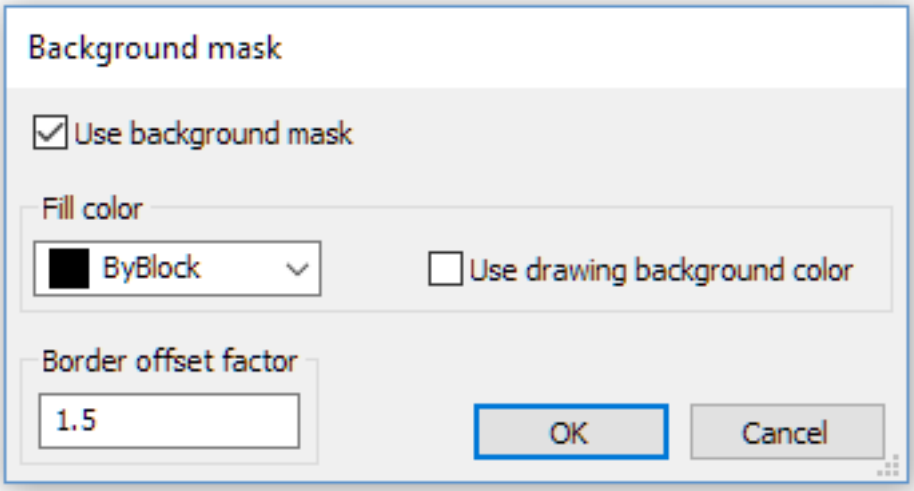
Almost equal	≈	\U+2248
Angle	∠	\U+2220
Center line	℄	\U+2104
Delta	Δ	\U+0394
Electrical phase	Ø	\U+0278
Identity	≡	\U+2261
Not equal	≠	\U+2260
Omega	Ω	\U+03A9
Squared	²	\U+00B2
Cubed	³	\U+00B3

Symbols and Unicode strings (*) If supported by the text font.

To add a background mask

1. When creating or editing a multiline text entity, click the **Background Mask** tool button () on the **Text Formatting** toolbar. The **Background Mask** dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enl
2. Check the **Use background mask** option.
3. Do one of the following:
   - Choose a color in the **Fill color** drop-down list.
   - Choose **Select color...** in the **Fill color** drop-down list and choose an Index color (1.17.4) or a True color (1.17.4) in the **Select color** dialog.
   - Check the **Use drawing background color** option.
4. Define the **Border offset factor** , which defines the margin around the text for the background. The value is based on the text height and must be in the range 1-5. A factor of 1.0 exactly fits the multiline text entity.
5. Click **OK** to create the background mask.

Figure 1.901



To add a text frame

1. Select the Mtext entity.
2. Click the **Text frame** property in the Properties (6.24.49) panel.
3. Click the down arrow and choose **Yes**.

Figure 1.902

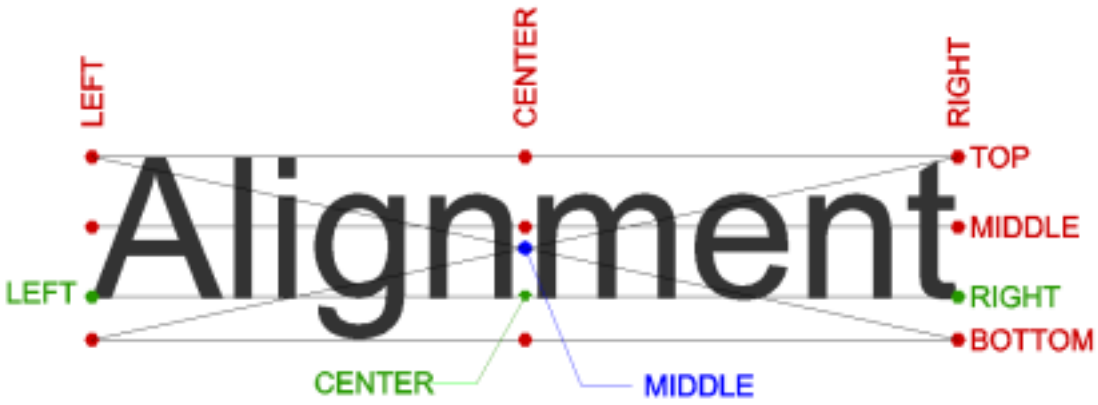


Figure 1.903

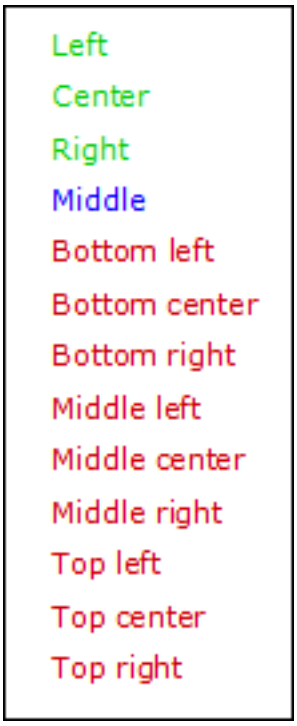


Figure 1.904



Figure 1.905



Figure 1.906

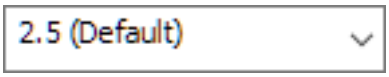


Figure 1.907

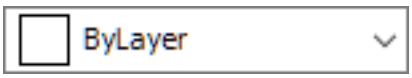


Figure 1.908



Figure 1.909



Figure 1.910



Figure 1.911

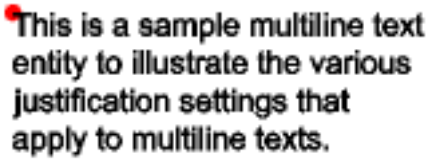


Figure 1.912

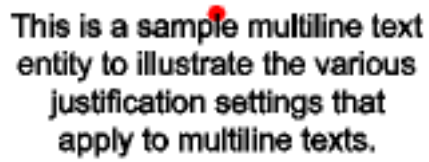


Figure 1.913

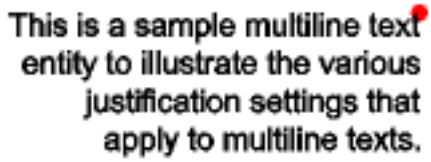


Figure 1.914

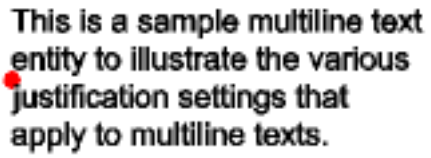


Figure 1.915

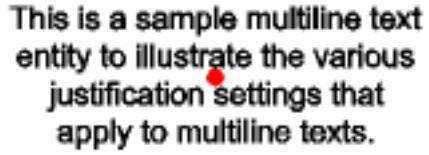


Figure 1.916

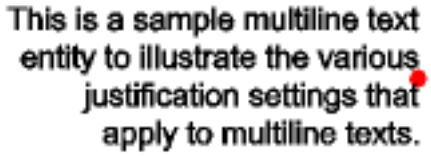


Figure 1.917

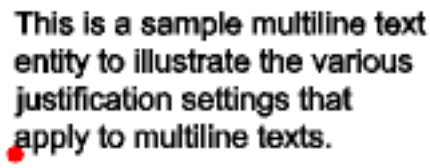


Figure 1.918

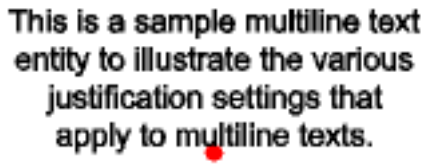
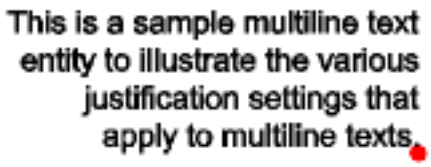


Figure 1.919


This is a sample multiline text
entity to illustrate the various
justification settings that
apply to multiline texts.

1.21.21 Exploding text

Command: TXTEXP (6.29.27) The **TXTEXP** command explodes Text and Mtext entities into 2D polylines (1.17.12).

To explode text

1. Type **TXTEXP** in the command bar, then press Enter. **You are prompted:** Select text to be exploded:
2. Select a single text entity or multiple text entities, then press Enter. The selected text(s) are exploded.

NOTE	The solid fill of certain TTF fonts is removed when exploding such text, leaving the text outline only.

1.21.22 Finding and replacing text

Command: FIND (6.14.8) The **FIND** command lets you specify the text you want to find. You can than find or replace the text. Optionally you can zoom in on the various instances of the text entities that contain the search text.

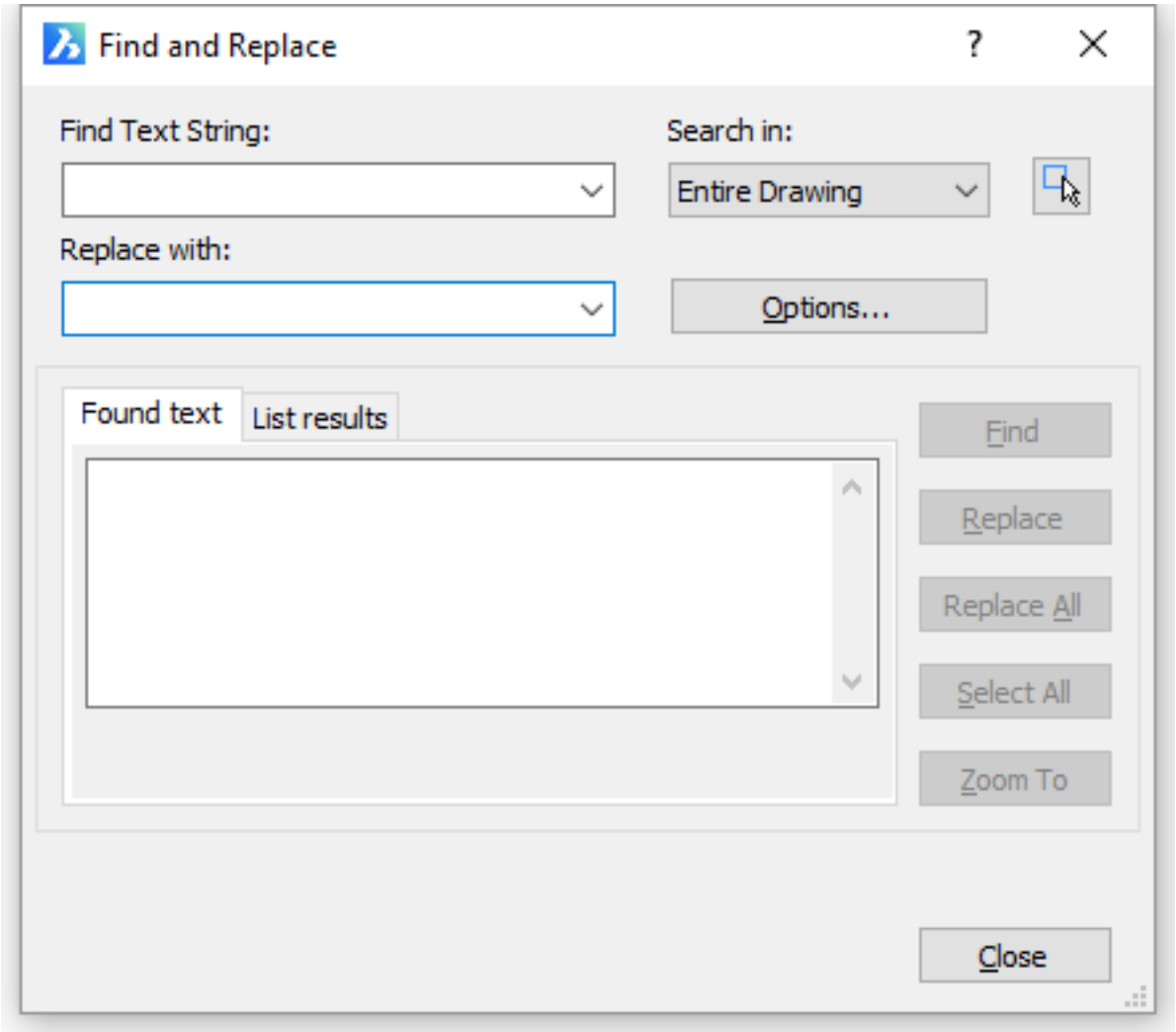
To open the Find and Replace dialog

Do one of the following:

- Choose **Find...** in the **Edit** menu.
- Type **find** in the command bar, then press Enter.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithTex

Figure 1.920

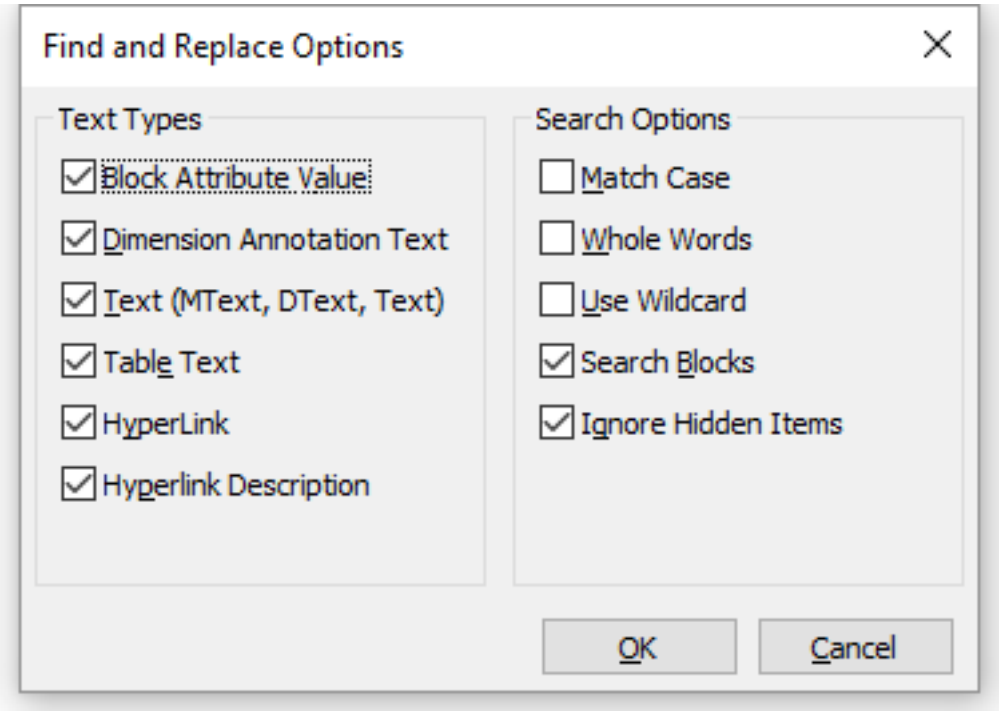

Find and Replace
?
Find Text String:
Search in:
Entire Drawing
Replace with:
Options...
Found text
List results
Find
Replace
Replace All
Select All
Zoom To
Close

To set the Find and Replace options

1. Open the Find and Replace dialog (1.21.22).
2. Click the **Options** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Drav

3. Set the **Find and Replace** options.
4. Click the **OK** button.

Figure 1.921



To find text

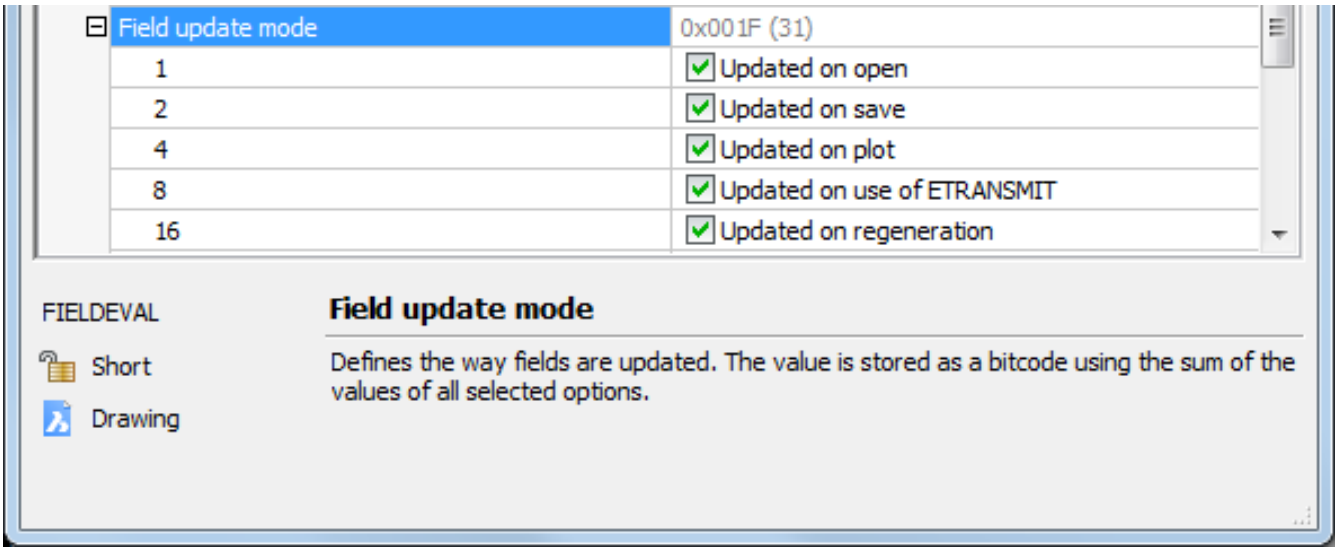
1. Open the Find and Replace dialog (1.21.22).
2. (option) Set the Find and Replace options (1.21.22).
3. Type the text in the **Find Text String** field.
4. (option) Click the **Select Entities** button ( ) to compose a selection set.
   - The **Find and Replace** dialog temporarily closes to let you select entities (1.23.6).
   - Right click to conclude the selection of entities. The **Search in** field reads **Current Selection** instead of **Entire Drawing** .
5. Click the **Find Next** button. The content of the first text entity that contains the search string displays in the **Search results** pane.
6. (option) Click the **Zoom To** button. The drawing is zoomed in on the text entity.
7. Do one of the following:
   - Repeat steps 5 and 6 to find more instances of the same search string.
   - Repeat steps 2 through 6 to find another text string.
   - Press the Esc key or click the **Close** button to stop.

NOTE If a selection is active when you launch the **FIND** command, only text entities in the selection set are searched.

To replace text

1. Open the Find and Replace dialog (1.21.22).
2. (option) Set the Find and Replace options (1.21.22).
3. Type the search text in the **Find Text String** field.
4. Type the replacement text in the **Replace with:** field.
5. (option) Click the **Select Entities** button ( ) to compose a selection set.
   - The **Find and Replace** dialog temporarily closes to let youselect entities (1.23.6).
   - Right click to conclude the selection of entities. The **Search in** field reads **Current Selection** instead of **Entire Drawing** .

Figure 1.922



1.21.23 Working with Fields

Commands: FIELD (6.14.3), UPDATEFIELD (6.30.10), MTEXT (6.21.35) and TABLE (6.29.1) A field is a variable text that. When a field is updated it displays the latest value of the data source it refers to. Fields can be used to insert drawing properties, custom properties or object properties in multiline texts or tables or as an attribute in a block. Fields in attributes are especially useful in title blocks: using drawing, sheet set, subset and sheet properties title blocks can be filled out automatically. See the DWGPROPS (6.10.42) command to learn more about default and custom drawing properties. See the SHEETSET (6.27.31) command to learn more about default and custom sheet set, subset and sheet properties. If no value is available, a field displays hyphens (----). If the FIELDDISPLAY (1.13.11) system variable is ON, fields display on a light gray background which does not print. The FIELDEVAL (1.13.11) system variable controls when fields are updated. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/03FIELDEVALSet

General procedure to create fields

1. Do one of the following:

   - Click the **Insert Field** tool button ( ) on the **Insert** toolbar.
   - Choose **Insert Field...** in the **Insert** menu.
   - Type **field** in the command bar, then press Enter.
   - Click the **Field** tool button ( ) on the **Text Formatting** toolbar when creating or editing Mtext (1.21.20) or a table (1.21.25) cell content.

   The **Field** dialog box displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10

2. Under **Field Names** , expand a group, then choose a field name. The settings for the selected field display.

3. Adjust the settings, then click the **OK** button. The **Field** dialog box closes. The field is created in the Mtext or table cell being edited **Or You are prompted:** Specify start point or [Height/Justify]: A context menu displays:



   - (option) Type **H** or choose **Height** in the context menu to define the text height.
   - (option- Type **J** or choose **Justify** in the context menu to set the text justification. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/08Fiel

Figure 1.923

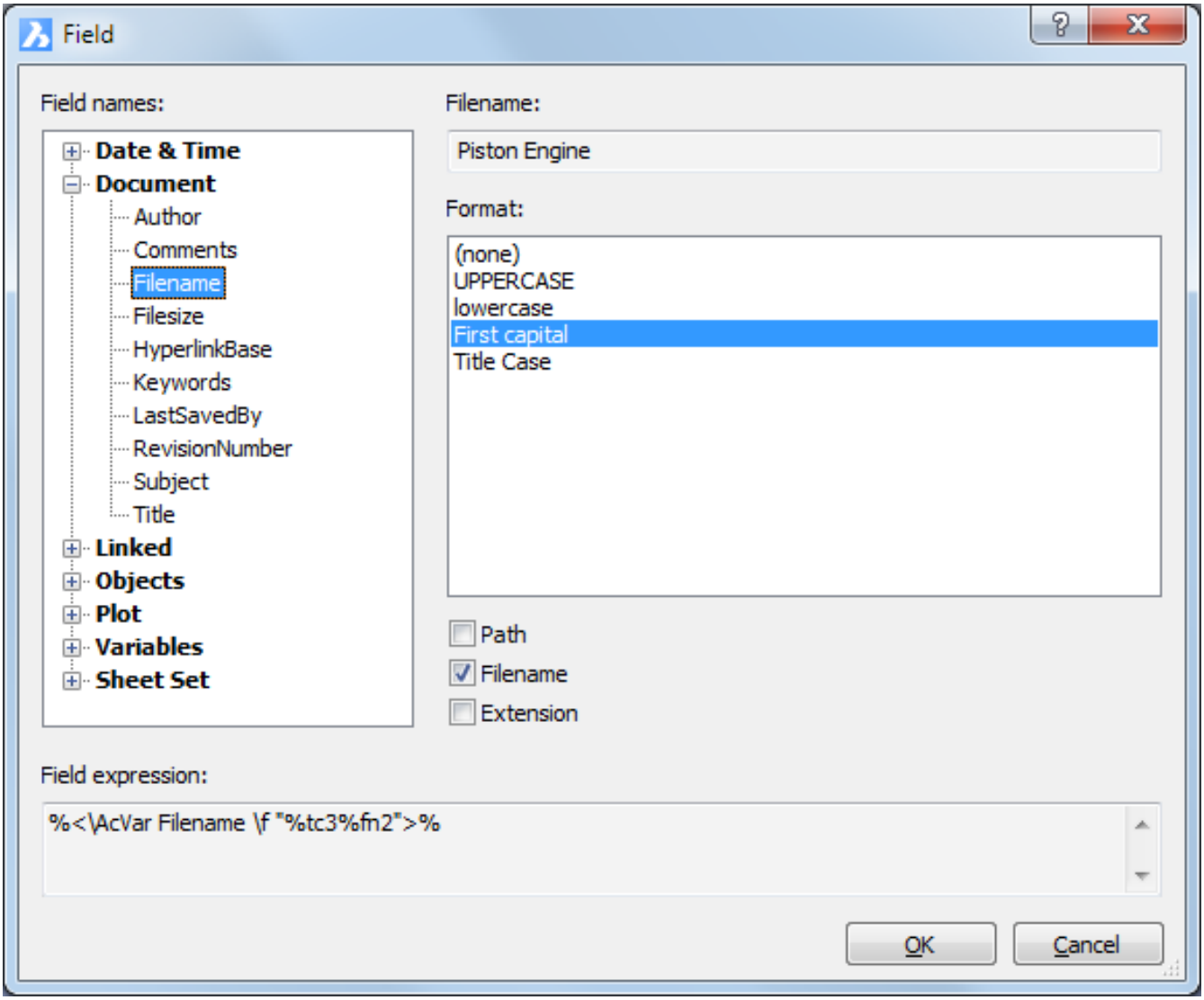


Figure 1.924

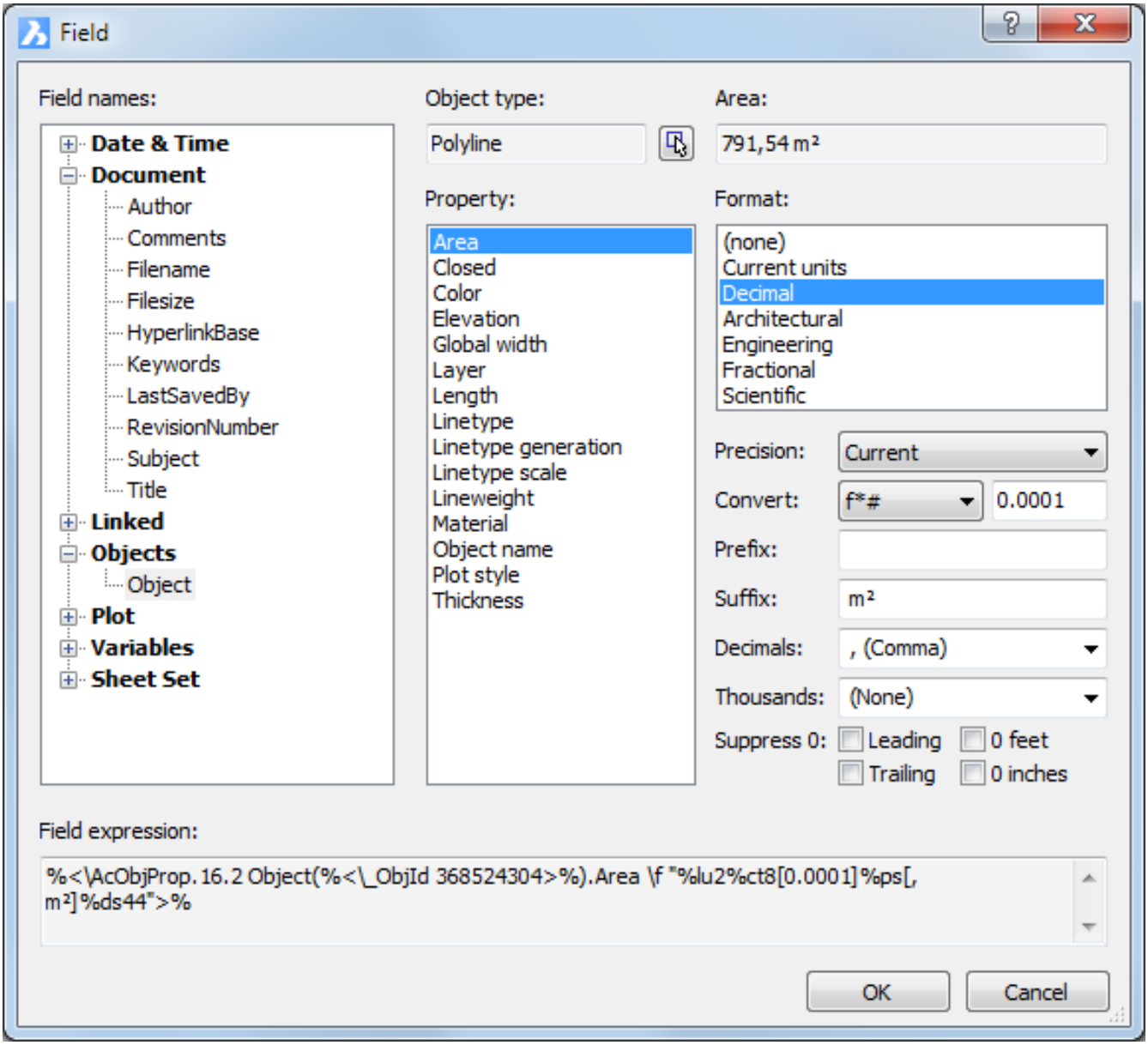


- Specify the insertion point of the field in the drawing. An **Mtext** entity, containing the field is created.

To display an entity property in a field

1. Launch the Field command (1.21.23).
2. Expand **Objects** in the **Field names** list and select **Object**.
3. Click the **Select Entity** button ( ) on the **Field** dialog box. The **Field** dialog box closes temporarily. **You are prompted:** Select Entity:
4. Click the entity you want to display a property of. The **Field** dialog box displays again.
5. Select the entity property in the **Property** list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us
6. Set the display layout:
   - **Format** : choose a unit format.
   - **Precision** : select a precision in the list button
   - **Convert** : this option allows to recalculate the field value.
     - Choose a formula in the list; e.g. f*#, where f is the factor you type in the Convert field and # is the field value.
     - Type a factor in the Convert field; e.g. 0.0001 to express the area of a polyline in square meters in a cm drawing.
   - **Prefix** : type a text string
   - **Suffix** : type a text string
   - **Decimals** : select a decimal separator character
   - **Thousands** : select a thousands separator character
   - **Suppress 0** : allows to suppress leading and/or trailing zeros
7. Click the **OK** button.
8. Specify a point to place the field in the drawing.

Figure 1.925


Field
Field names:
Date & Time
Document
Author
Comments
Filename
Filesize
HyperlinkBase
Keywords
LastSavedBy
RevisionNumber
Subject
Title
Linked
Objects
Object
Plot
Variables
Sheet Set
Object type:
Polyline
Area:
791,54 m²
Property:
Area
Closed
Color
Elevation
Global width
Layer
Length
Linetype
Linetype generation
Linetype scale
Lineweight
Material
Object name
Plot style
Thickness
Format:
(none)
Current units
Decimal
Architectural
Engineering
Fractional
Scientific
Precision:
Current
Convert:
f*#
0.0001
Prefix:
Suffix:
m²
Decimals:
, (Comma)
Thousands:
(None)
Suppress 0:
Leading
0 feet
Trailing
0 inches
Field expression:
%<\AcObjProp.16.2 Object(%<_ObjId 368524304>%).Area \f "%lu2%ct8[0.0001]%ps[, m²]%ds44">%
OK
Cancel

To edit a field

1. Edit the table cell or Mtext entity that contains the field.
2. Do one of the following:

   - Double click the field.
   - Right click the field, then choose **Edit Field ...** in the context menu.
   - Select the field, then press **Ctrl + F** .

   The **Field** dialog box displays.
3. Adjust the properties of the field.
4. Click the **OK** button to apply the changes.

To update a field

1. Do one of the following:

   - Click the **Update Field** tool button () on the **Redraw / Regen** toolbar.
   - Choose **Update Fields...** in the **Tools** menu.
   - Type **updatefield** in the command bar.

   You are prompted: Select field(s) to update:
2. Select the field(s) you want to update, then right click or press Enter. The selected fields are updated.

To convert a field to text

1. Edit the table cell or Mtext entity that contains the field.
2. Select the field.
3. Right click, then choose **Convert Field to Text** in the context menu.

1.21.24 Checking Spelling

Command: SPELL (6.27.63) The spelling checker checks the spelling of text in the current drawing. You can check the spelling of one or more selected text entities or check the entire drawing. During the spelling check, the **Spell** command matches the words in the drawing or the current selection set to the words in the current main dictionary and the current custom dictionary. If a word is not found in either dictionaries you can select the correct spelling in the suggestions field or you can add the word to the current custom dictionary. Custom dictionaries are used for discipline-specific words, such as medical or mechanical.

Dictionary settings

Which dictionaries are used by the **Spell** command is defined by the **DCTMAIN** (main dictionary) and **DCTCUST** (custom dictionary) system variables (1.13.11). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGu

NOTE	We only deliver the English dictionary because of the license type on the dictionary files : the English version can be distributed freely ; all other languages are not free. However, you can download other languages and use them for free (see To add (1.21.24) a main dictionary (1.21.24)).

Figure 1.926

Dictionaries	
Custom spelling dictionary	C:\Users\Louis\AppData\Roaming\Bricsys\Bricscad\V9\en_US\support\Sample.cus
Main spelling dictionary	en_US.dic

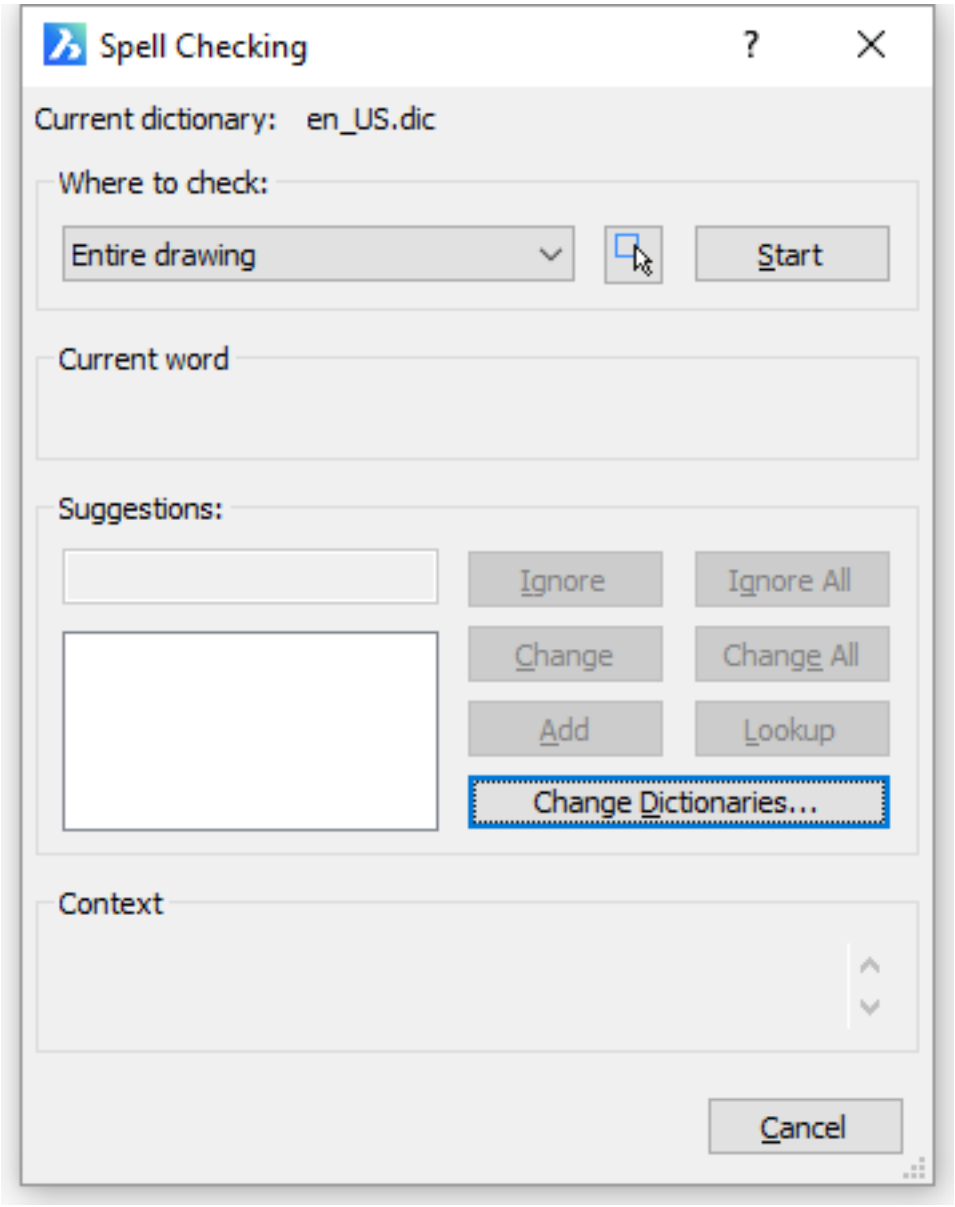
To open the Spell Checking dialog

Do one of the following:

- Choose **Check Spelling** in the **Tools** menu.
- Type **spell** in the command bar, then press Enter.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithTex

Figure 1.927

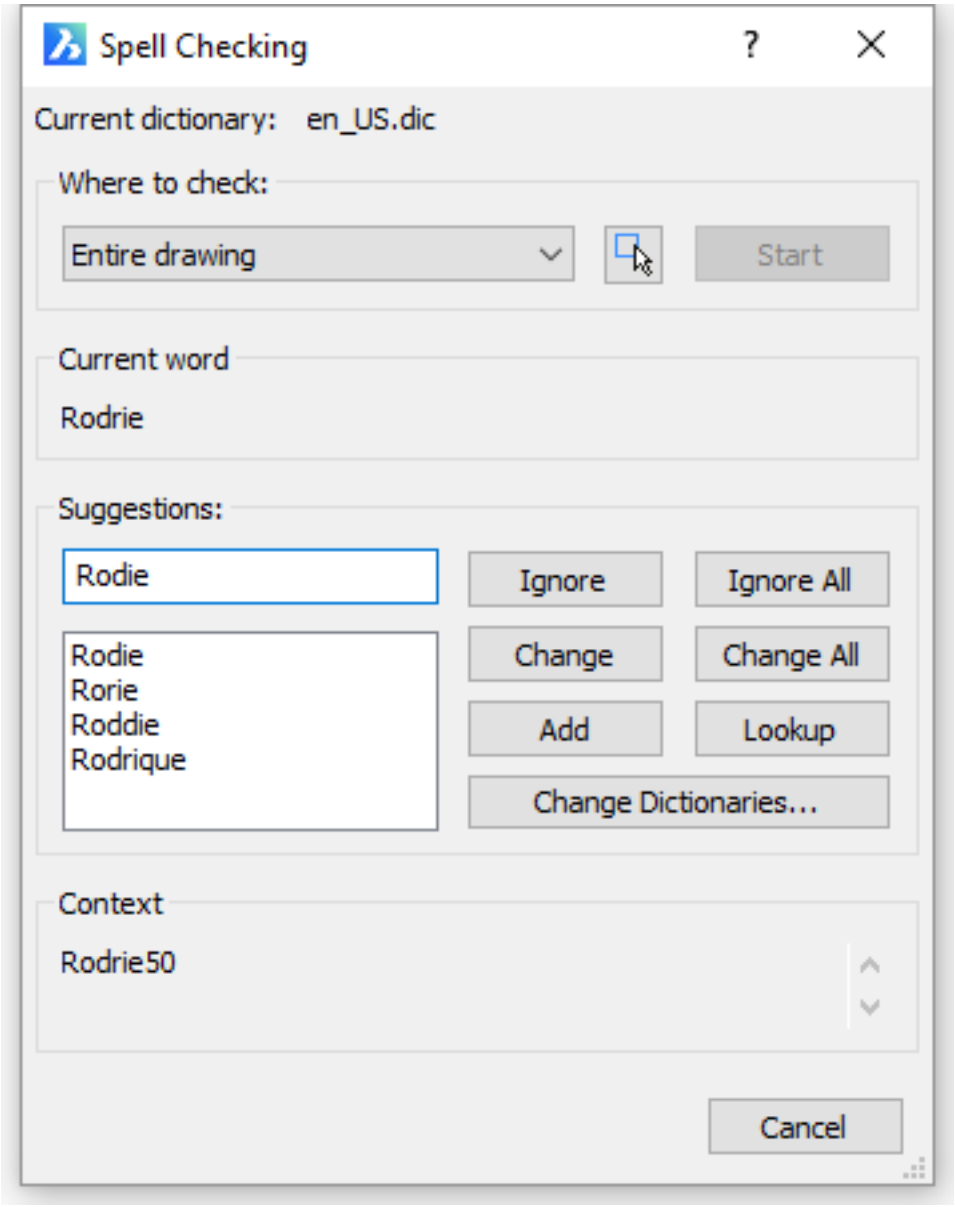

Spell Checking
Current dictionary: en_US.dic
Where to check:
Entire drawing
Start
Current word
Suggestions:
Ignore
Ignore All
Change
Change All
Add
Lookup
Change Dictionaries...
Context
Cancel

To check the spelling

1. Open the Spell Checking dialog (1.21.24).
2. (option) Click the **Select** button ( ) to compose a selection set (1.23.6).
3. (option) Change the current dictionaries (1.21.24).
4. Click the **Start** button. The first possibly misspelled word displays in the **Current word** section. The content of the text entity in which the **Current word** is found displays in the **Context** section. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/06SpellChe
5. Do one of the following:
   - Click the **Ignore** button to leave this instance of the **Current word** unchanged.
   - Click the **Ignore All** button to leave all instances of the **Current word** unchanged.
   - Accept the word in the **Suggestions** field or select a word in the **Suggestions** list, then click the **Change** button to replace this instance of the **Current word** by the word in the **Suggestions** field.
   - Accept the word in the **Suggestions** field or select a word in the **Suggestions** list, then click the **Change All** button to replace all instances of the **Current word** by the word in the **Suggestions** field.
   - Click the **Add** button to add the **Current word** to the current custom dictionary.
   - Click the **Lookup** button to list words similar to the word in the **Suggestions** field.
6. Repeat step 5 until the **Spelling check is complete** message displays.

NOTE	If a selection is active when you launch the **Spell** command, only text entities in the selection set are checked.

Figure 1.928


Spell Checking
?
Current dictionary: en_US.dic
Where to check:
Entire drawing
Start
Current word
Rodrie
Suggestions:
Rodie
Ignore
Ignore All
Rodie
Rorie
Roddie
Rodrique
Change
Change All
Add
Lookup
Change Dictionaries...
Context
Rodrie50
Cancel

To change dictionaries

1. Open the Spell Checking dialog (1.21.24).
2. Click the **Change Dictionaries...** button. The **Change Dictionaries** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithText/Art/06ChangeD
3. Click the **Main dictionary** list button to change the main dictionary. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscon
4. Select a new main dictionary.
5. (option) Click the **Browse...** button to select a new **Custom dictionary**.
6. (option) Type a word in the **Custom dictionary words** field, then click the **Add** button to add a word to the current custom dictionary.
7. Click the **Apply & Close** button.

NOTE

- Custom dictionaries are best saved in the **Support** folder of the Roamable root folder (1.13.2).
- If you want to share the custom dictionary with MS Word, point to the MS Word custom dictionary in step 5 in the above procedure. By default, **CUSTOM.DIC** is the custom dictionary in MS Word, which can be found under **C:\Users\<user name>\AppData\Roaming\Microsoft\Proof\CUSTOM.DIC**.

Figure 1.929

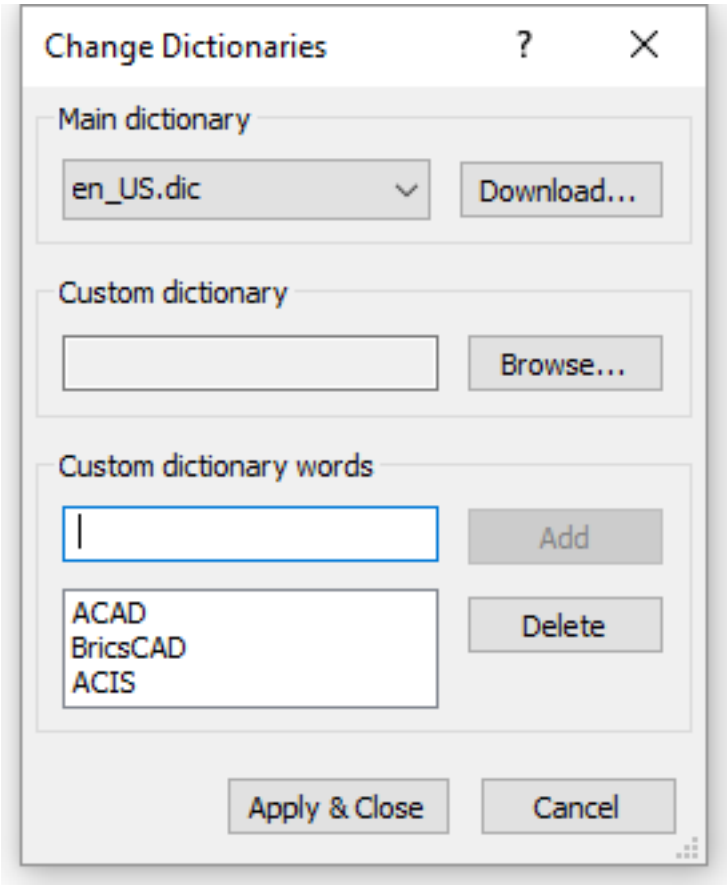
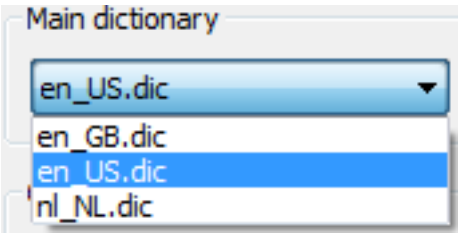


Figure 1.930



To add a new main dictionary

1. Open the Spell Checking dialog (1.21.24).
2. Click the **Change Dictionaries...** button.
3. Click the **Download...** button. Your browser opens: https://extensions.openoffice.org/dictionaries
4. Scroll down to the language of your choice, then click the link to open the page of the selected language.
5. Click the **Download extension** button to start downloading the ***.oxt** file. (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
6. Change the **oxt** extension to **zip** .
7. Extract the ***.aff** and ***.dic** files from the ZIP file to the folder **C:\Program Files\Bricsys\BricsCAD Vx\Support**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05Workingw
The dictionary is now available in the **Main Dictionary** list on the **Change Dictionaries** dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/05WorkingwithTex

Figure 1.931

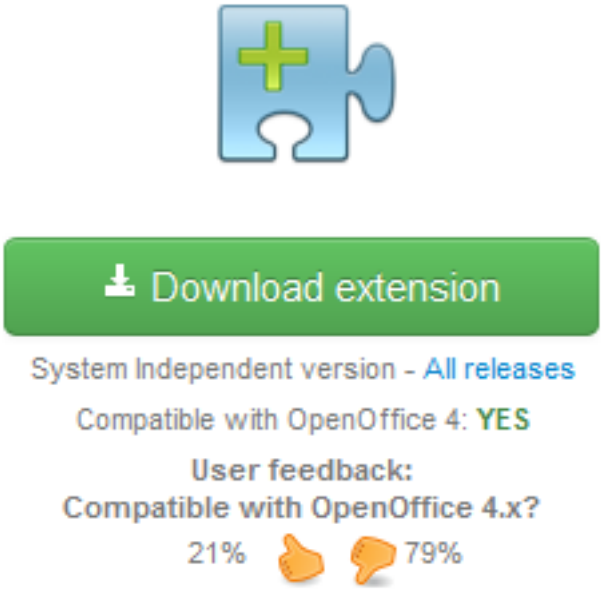


Figure 1.932

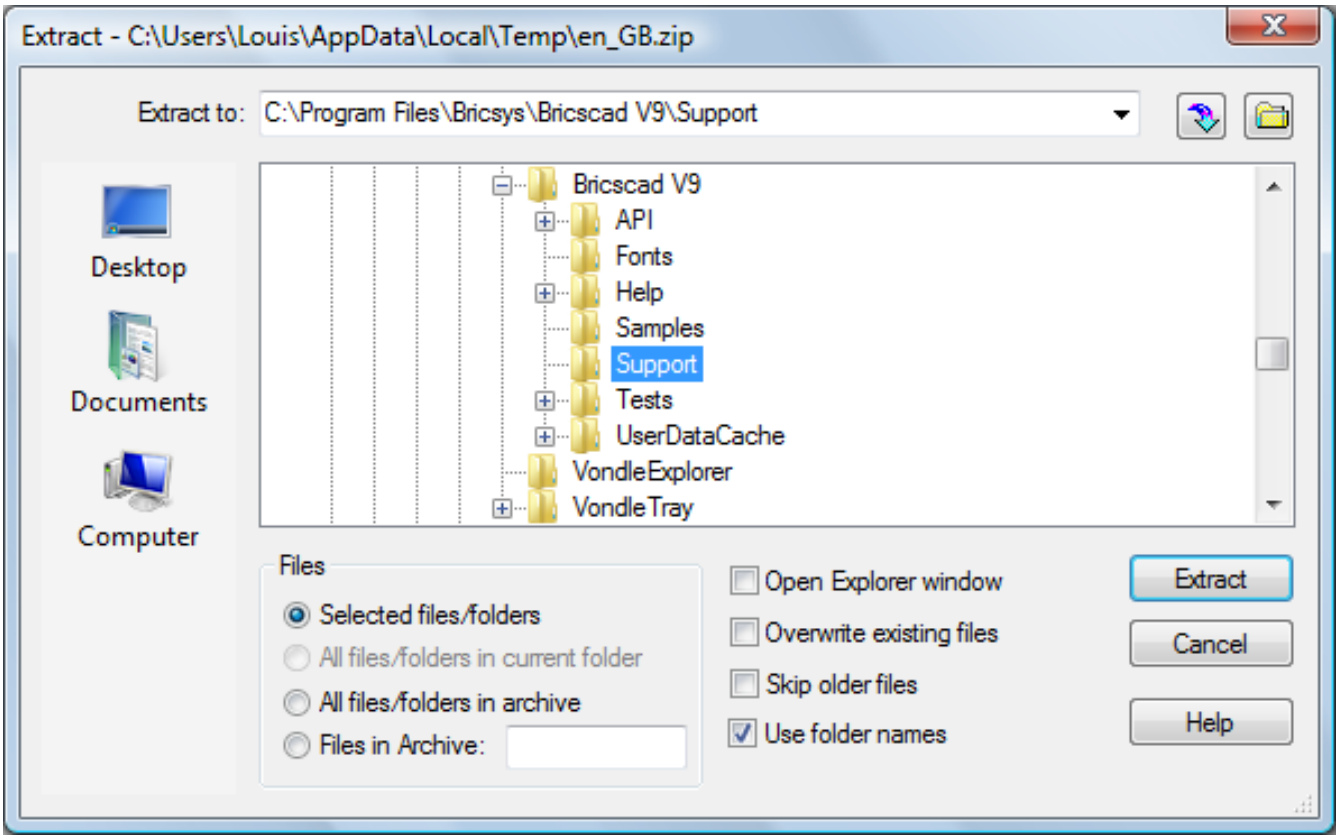
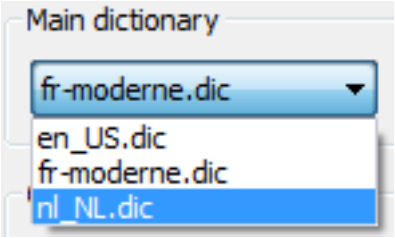


Figure 1.933



To add a new custom dictionary

1. Open a text editor, such as **Notepad**.
2. (option) Type the words you want to use in your custom dictionary. Each word must be on a separate line.
3. Save the file to the the **Support** folder of the Roamable root folder (1.13.2). Make sure the file extension is **.cus**.

Figure 1.934

A				
B	B	B	B	B
C	C	C	C	C
C	C	C	C	C
C	C	C	C	C

C	C	C	C	C
C	C	C	C	C
C	C	C	C	C
B	B	B	B	B
A				

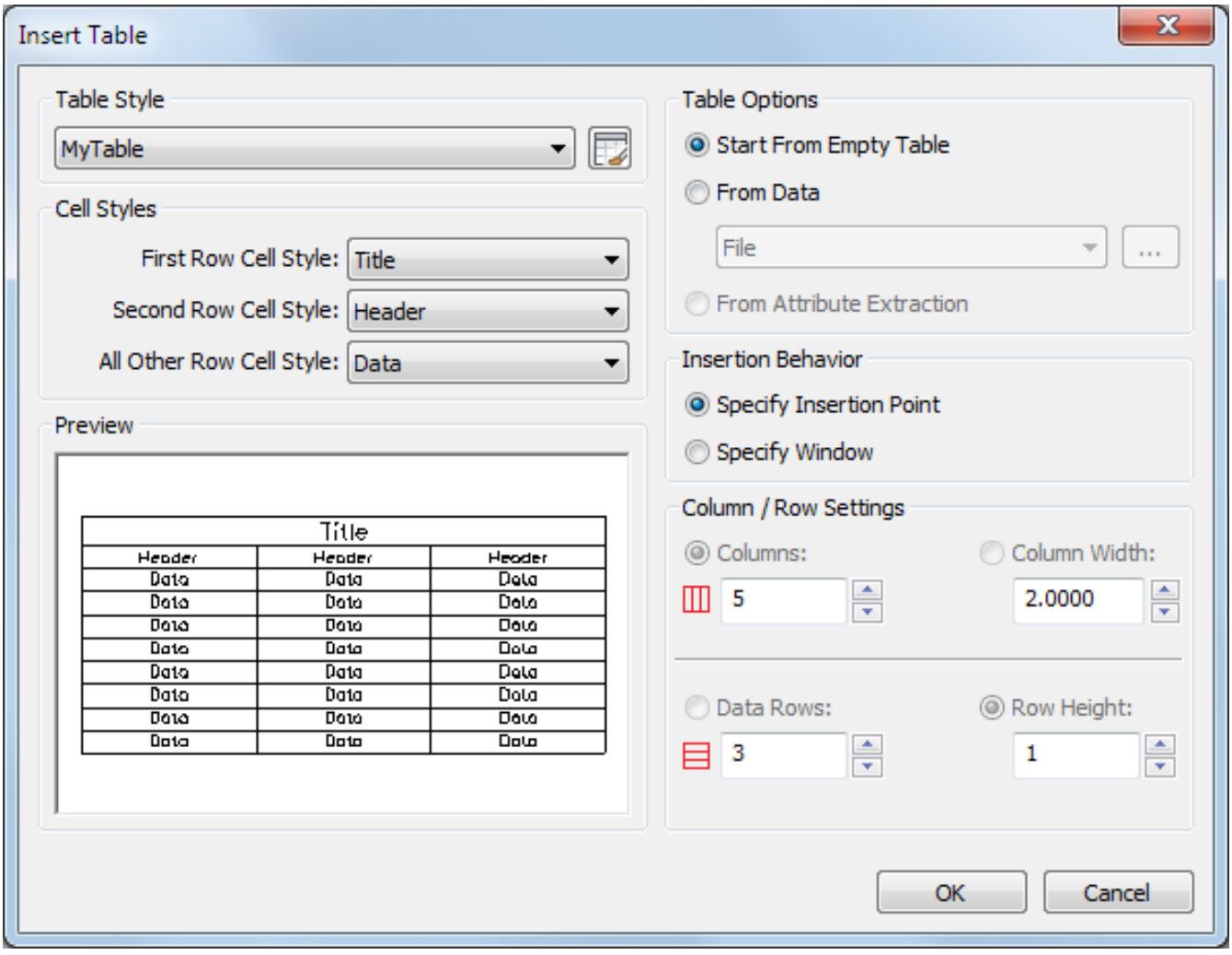
1.21.25 Tables

Commands: TABLE (6.29.1), -TABLE (6.29.2), TABLEDIT (6.29.3), TABLEEXPORT (6.29.4), TABLEMOD (6.29.5), TABLESTYLE (6.29.6) A table is an entity that is composed of rows and columns, much like a Microsoft Excel spreadsheet. A table entity can be created as an empty table or you can import data from a CSV (C omma S eparated V alues) file. The appearance of the table text and gridlines is controlled with a table style. Table styles are saved in the DWG file. A default table consists of a **Title** cell (A), a row of **Header** cells (B) and one or more rows of **Data** cells (C). A table can have one of two directions: **Down** (title row at the top) or **Up** (title row at the bottom). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/ART/07TableCells.png}) Table with a **Down** direction (left) and a **Up** direction (right).

To create an empty table

1. Do one of the following:
   - Click the **Table** tool button ( ) on the **Draw** toolbar.
   - Choose **Insert Table...** in the **Draw** menu.
   - Type **table** in the command bar, then press Enter.
2. The **Insert Table** dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source Choose a Table Style (1.21.26) from the **Table Style** list.
3. (option) Set the cell style for the first, second and other rows. By default the cell style for the first row is **Title**, the cell style for the second row is **Header** and the cell style for the other rows is **Data.**
4. Under **Table Options**, choose **Start from Empty Table**.
5. Under **Insertion Behavior** do one of the following:
   - Choose **Specify Insertion Point**, then specify the number of columns and the column width and number of data rows and the row height.
   - Choose Specify Window, then do one of the following:
     - Specify the number of columns and data rows.
     - Specify the column width and the row height.
     - Specify the column width and the number of data rows.
     - Specify the number of columns and the row height.
6. Click the **OK** button. Depending on your choice in step 5 **you are prompted** to specify an insertion point or a window. The table is created and the **Text Formatting** toolbar displays (see Create multiline text (1.21.20)).
7. Do one of the following:
   - Start adding text in the table (see Edit the table content (1.21.25)).
   - Click outside the table to close the **Text Formatting** toolbar.

Figure 1.935


Insert Table
Table Style
MyTable
Cell Styles
First Row Cell Style:
Title
Second Row Cell Style:
Header
All Other Row Cell Style:
Data
Preview
Title
Header
Data
Table Options
Start From Empty Table
From Data
File
From Attribute Extraction
Insertion Behavior
Specify Insertion Point
Specify Window
Column / Row Settings
Columns:
5
Column Width:
2.0000
Data Rows:
3
Row Height:
1
OK
Cancel

To create a table from data

When importing data from a CSV (Comma Separated Values) file, please make sure the list separator character on your system is the same as the one used in the CSV file (see Defining the list separator character).

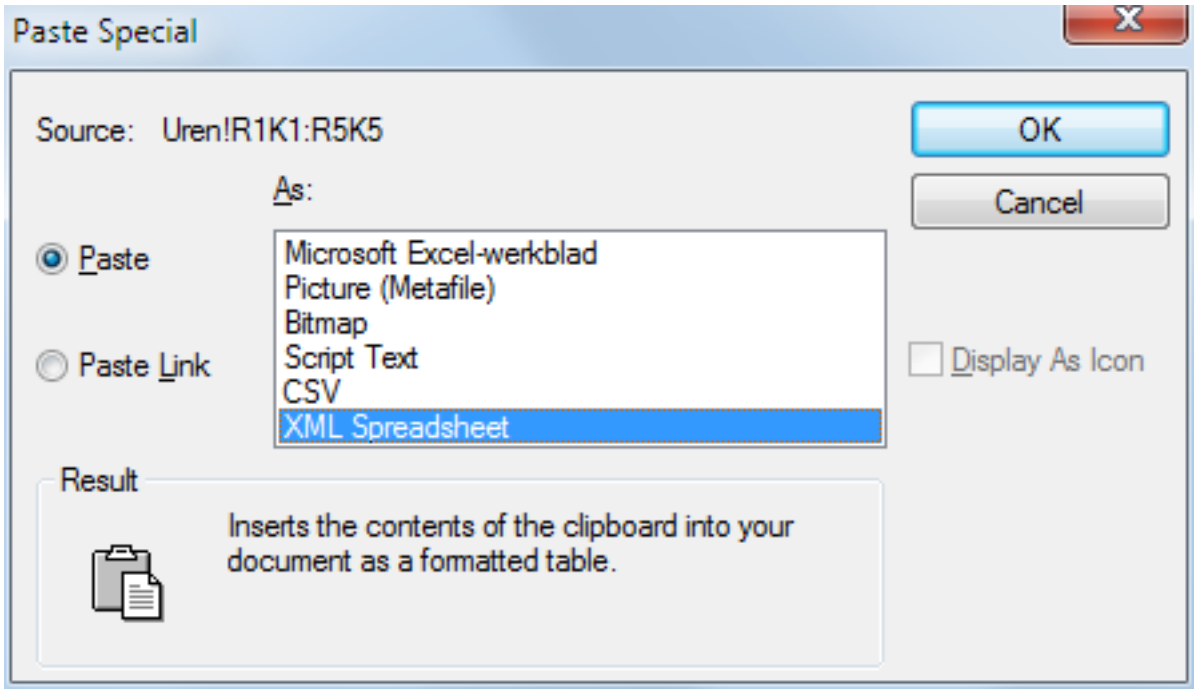
1. Do one of the following:
   - Click the **Table** tool button ( ) on the **Draw** toolbar.
   - Choose **Insert Table...** in the **Draw** menu.
   - Type **table** in the command bar, then press Enter.
2. The **Insert Table** dialog displays. Choose a Table Style (1.21.26) from the **Table Style** list.
3. Under **Table Options**, choose **From Data**.
4. Click the Browse button ( ).
5. The **Open** dialog displays. Choose a CSV file, then press the **Open** button.
6. Click the OK button on the **Insert Table** dialog. The dialog box closes and the table is attached to the cursor.
7. Click a point in the drawing to place the table. The table is created and the **Text Formatting** toolbar displays (see Create multiline text (1.21.20)).
8. Do one of the following:
   - Start editing the text in the table (see Edit the table content).
   - Click outside the table to close the **Text Formatting** toolbar.
9. (option) Select the table and drag the grips to adjust the dimensions of the table.

To create a table from an Excel spreadsheet

1. Open the spreadsheet in **Microsoft Office Excel**.
2. Select the cells you want to create a table from.
3. Copy the selection to the clipboard:
   - Press **Ctrl+C**.
   - Right click and choose **Copy** in the context menu.
4. In BricsCAD, choose **Paste special...** in the **Edit** menu.
5. The **Paste Special** dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/s
   Choose **XML Spreadsheet** in the **As** option list.
6. Click in the drawing to insert the table.

Tutorial: BricsCAD Spreadsheet Import Techniques Click here (1.2.9) to watch

Figure 1.936



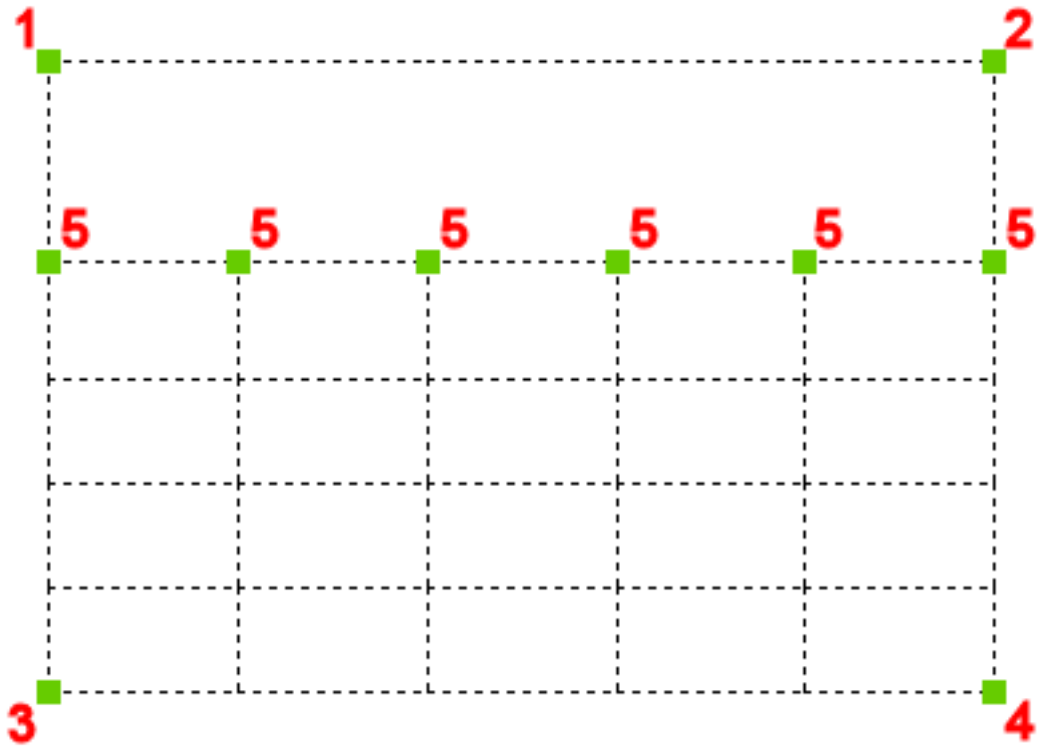
Editing a table using grips

When a table is selected, multiple grips display:

1. Moves the table.
2. Modifies the width uniformly.
3. Modifies the height uniformly.
4. Modifies the width and height uniformly.
5. Modifies the width of a column.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/ART/07TableGrips

Figure 1.937



Editing a table using the Table toolbar

1. To open the **Table** toolbar, do one of the following:
   - Click the cell you want to edit.
   - Press and hold the left mouse button then define a rectangle to select multiple cells
2. Column letters and row numbers display. The column(s) and row(s) of the selection hightlight. A rectangle with four handles is drawn around the selection. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/
   Do one of the following:
   - Click a cell to select it.
   - Press and hold the left mouse button, then drag the cursor to select multiple cells.
3. (option) Drag the grips of the selection to modify the dimensions.
4. (option) Click a tool in the Table toolbar.
5. (option) Right click to display a context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us
   - **Delete Cell Content** : deletes the text in the selected cells.
   - **Clear Cell Overrides** : resets all cell properties of the selected cells to the Cell Style settings as defined in the Table Style (1.21.26).
   - **Clear Table Overrides** : resets all cell properties of all cells to the Cell Style settings as defined in the Table Style (1.21.26).
6. (option) Repeat steps 3 and 4 to continue editing the table.
7. (option) Double click a cell to edit the cell text.
8. (option) Do one of the following:
   - Click another cell.
   - Press and hold the left mouse button, then drag the cursor to select multiple cells.
9. Click outside the table to close the **Table** toolbar.

Figure 1.938

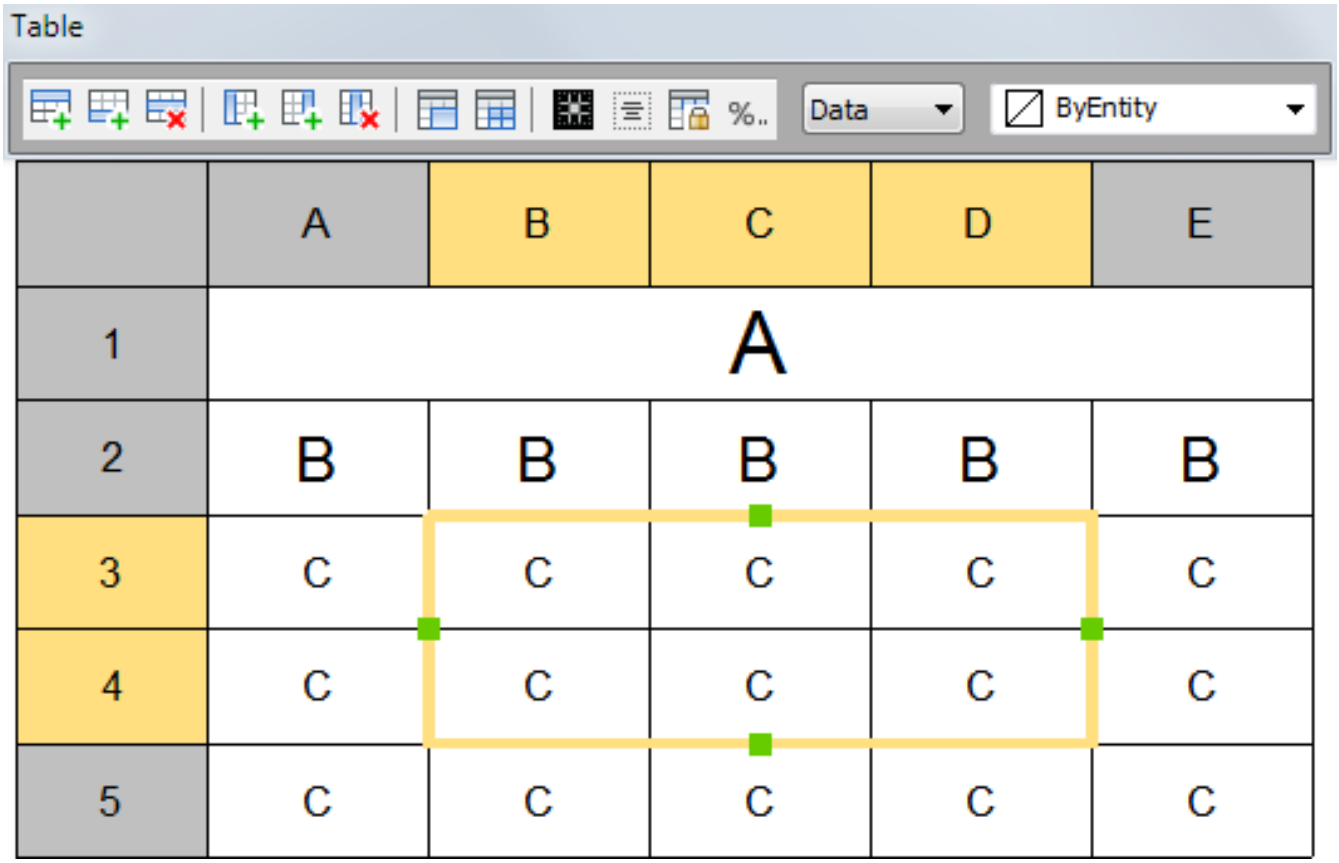
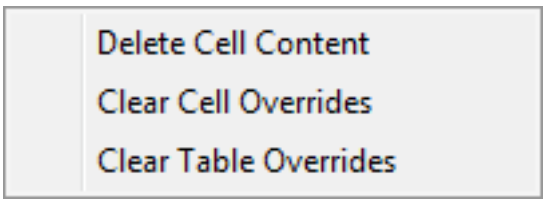


Figure 1.939



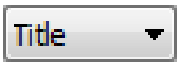
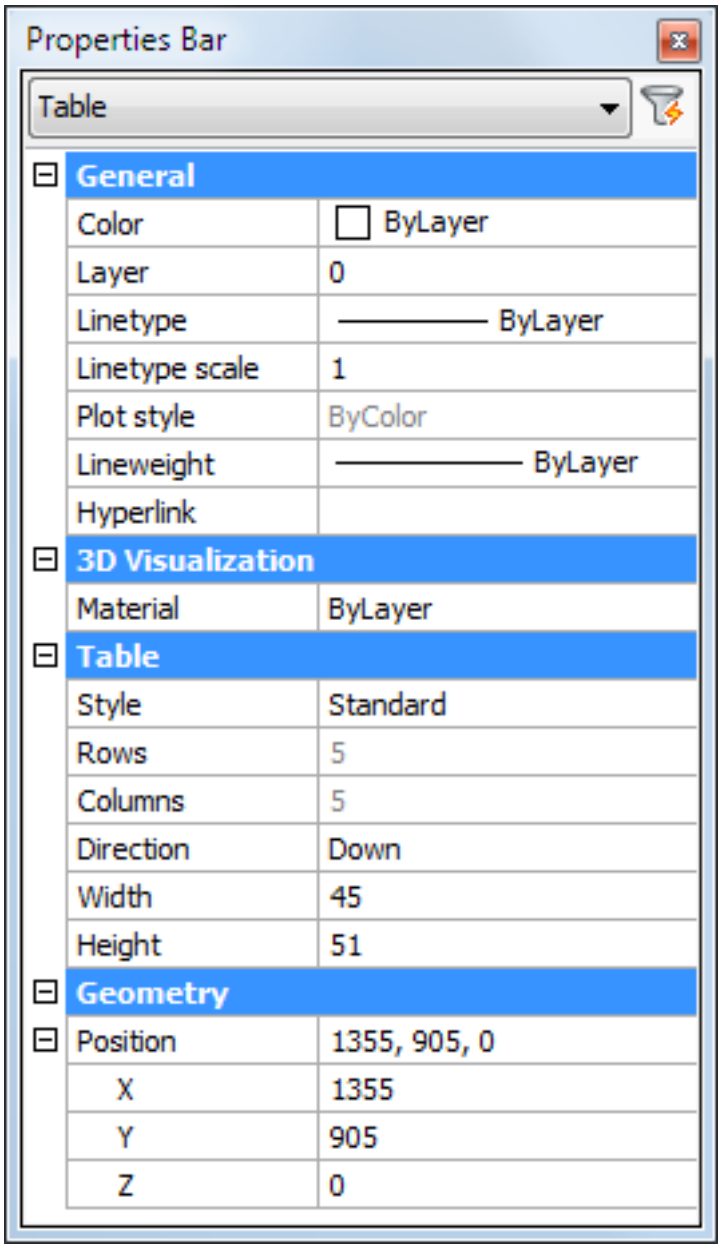
Icon	Tool name	Description
	Insert row above	Inserts an empty row above the selection.
	Insert row below	Inserts an empty row below the selection.
	Delete row	Deletes the row(s) of the selection.
	Insert column left	Inserts an empty column at the left of the selection.
	Insert column right	Inserts an empty column at the right of the selection.
	Delete column	Deletes the column(s) of the selection.
	Merge	Merges the selected cells.
	Unmerge	Splits previously merged cells.
	Border style	Sets the border style of the selection (not implemented yet).
	Text alignment	Sets the text alignment of the selection.
	Lock / Unlock	Locks / Unlocks the selection.
%..	Set cell format	Sets the cell format of the selection (not implemented yet)
	Set cell style	Sets the style of the selected cells: **Title** , **Header** or **Data** .
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/ART/07CellFillColor.png})	Set cell background	Sets the background color of the selection.

Table tools

Editing a table in the Properties bar

1. Click one of the grid lines. The properties of the table display in the Properties bar (1.12.4). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/ART/07EditTableProperties.png})
2. (option) Apply a differentTable Style (1.21.26).
3. (option) Change the table direction.
4. (option) Modify the width and/or height of the table.
5. (option) Modify the vertical and/or horizontal cell margins.
6. Press the Esc key to stop editing the table.

Figure 1.940


Properties Bar
Table
General
Color
ByLayer
Layer
0
Linetype
ByLayer
Linetype scale
1
Plot style
ByColor
Lineweight
ByLayer
Hyperlink
3D Visualization
Material
ByLayer
Table
Style
Standard
Rows
5
Columns
5
Direction
Down
Width
45
Height
51
Geometry
Position
1355, 905, 0
X
1355
Y
905
Z
0

Editing a cell in the Properties bar

1. Click the cell. The properties of the cell display in theProperties bar (1.12.4). (\vref {fig:C:/Dev/Task-2448/images/staticb
2. (option) Modify the width and/or height of the cell. The width and height of the column and/or row of the cell are changed.
3. (option) Modify the text rotation. The options are: 0°, 90°, 180° and 270°.
4. (option) Edit the cell content.
5. (option) Choose a different text alignment option. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1

Figure 1.941

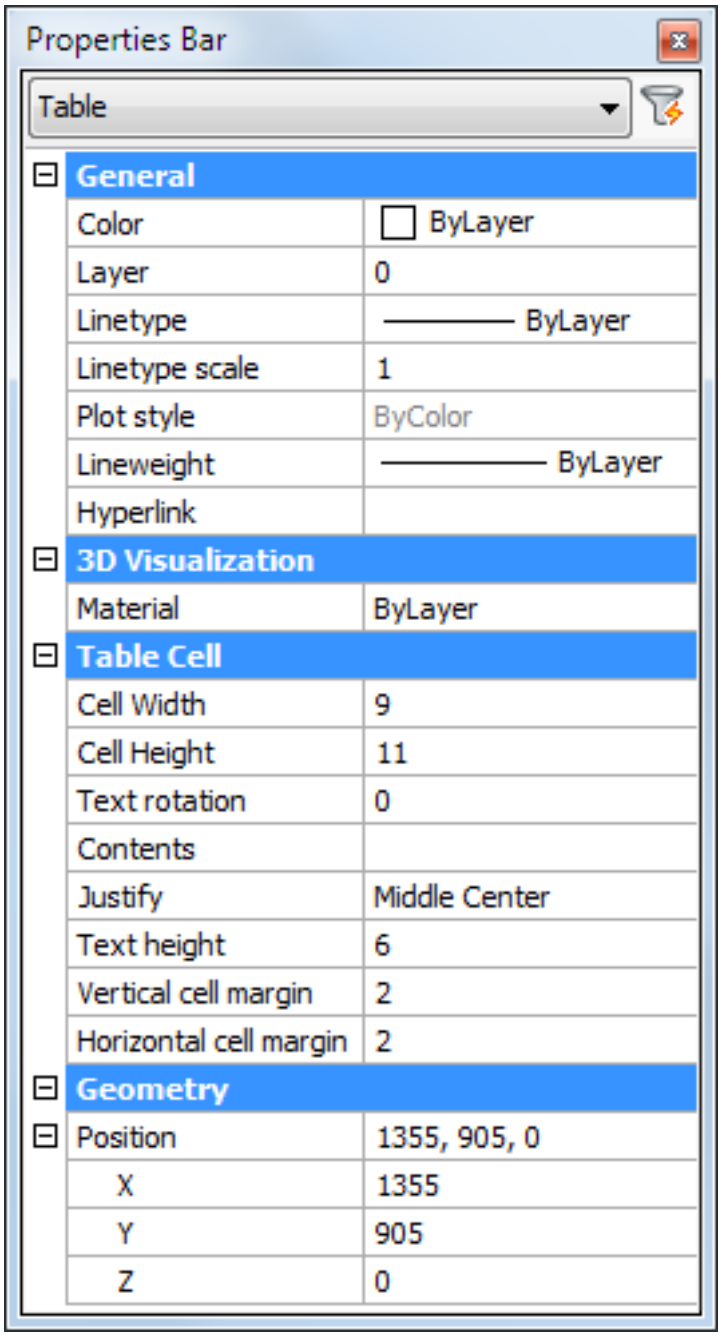
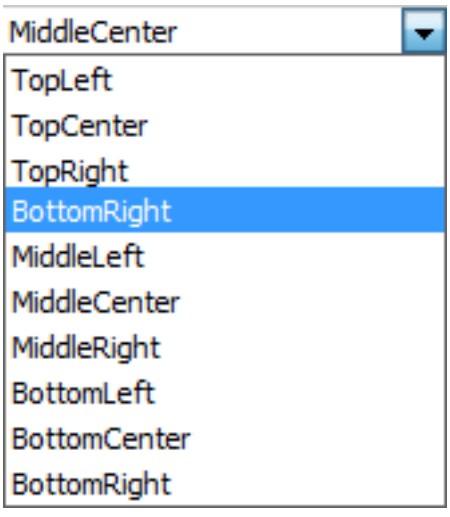


Figure 1.942



Evaluating formulas

Currently formulas are supported partially. There are more complex formulas which are not supported yet, but in most cases formulas in Excel sheets will remain functional when the Excel sheet is imported as a table. Editing and creating formulas through the field dialog is not supported yet, but in-place editing/creating is possible. E.g.: if you type =SUM(A2:A4) in a cell, it will be interpreted as a formula, and the sum of the values in the cells A2 through A4 is calculated.

To edit the table content

1. Do one of the following:
   - Double click the cell you want to edit.
   - Type **tabledit** in the command bar, then press Enter and click the cell you want to edit.
2. The **Text Formatting** toolbar displays above the selected cell (see Create multiline text (1.21.20)). Edit the text in thale cell. (option) Press Alt + Enter to create a second line in the same cell.
3. To edit another cell:
   - Press the TAB key to go to the next cell in the same row.
   - Press Shift + TAB to go to the previous cell in the same row.
   - Press Enter to go the next cell in the same column.
   - Press Shift + Enter to go to the previous cell in the same column.
   - Press one of the Arrow keys.
4. Click to stop editing the table content.

To export a table

When exporting data to a CSV (C omma S eparated V alues) file, the list separator character off your system is used in the CSV file (see Defining the list separator character).

1. Type **tableexport** in the command bar, then press Enter. **You are prompted:** Select table to export:
2. Click the table you want to export the data of. The Export Data dialog opens.
3. Select a folder to save the data in.
4. Type a name in the **File name** field.
5. Click the **Save** button. The data in the selected table are saved as a CSV (C omma S eparated V alues) file.

Defining the list separator character

1. Click the Start button in the Windows Taskbar, then choose Control Panel.
2. Do one of the following:
   - On the Control Panel Home window, choose: Clock, Language and Region > Regional and Language Options
   - On the Control Panel Classic View window, double click the Regional and Language Options icon.
3. On the Regional and Language Options dialog box click the Customize this format... button. The Customize Regional Options dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sc
4. On the Numbers page of the Customize Regional Options dialog box, go to List separator , then click the down arrow and select either comma (,) or semicolon (;).
5. Click the OK button on the Customize Regional Options and Regional and Language Options dialog boxes.

Figure 1.943

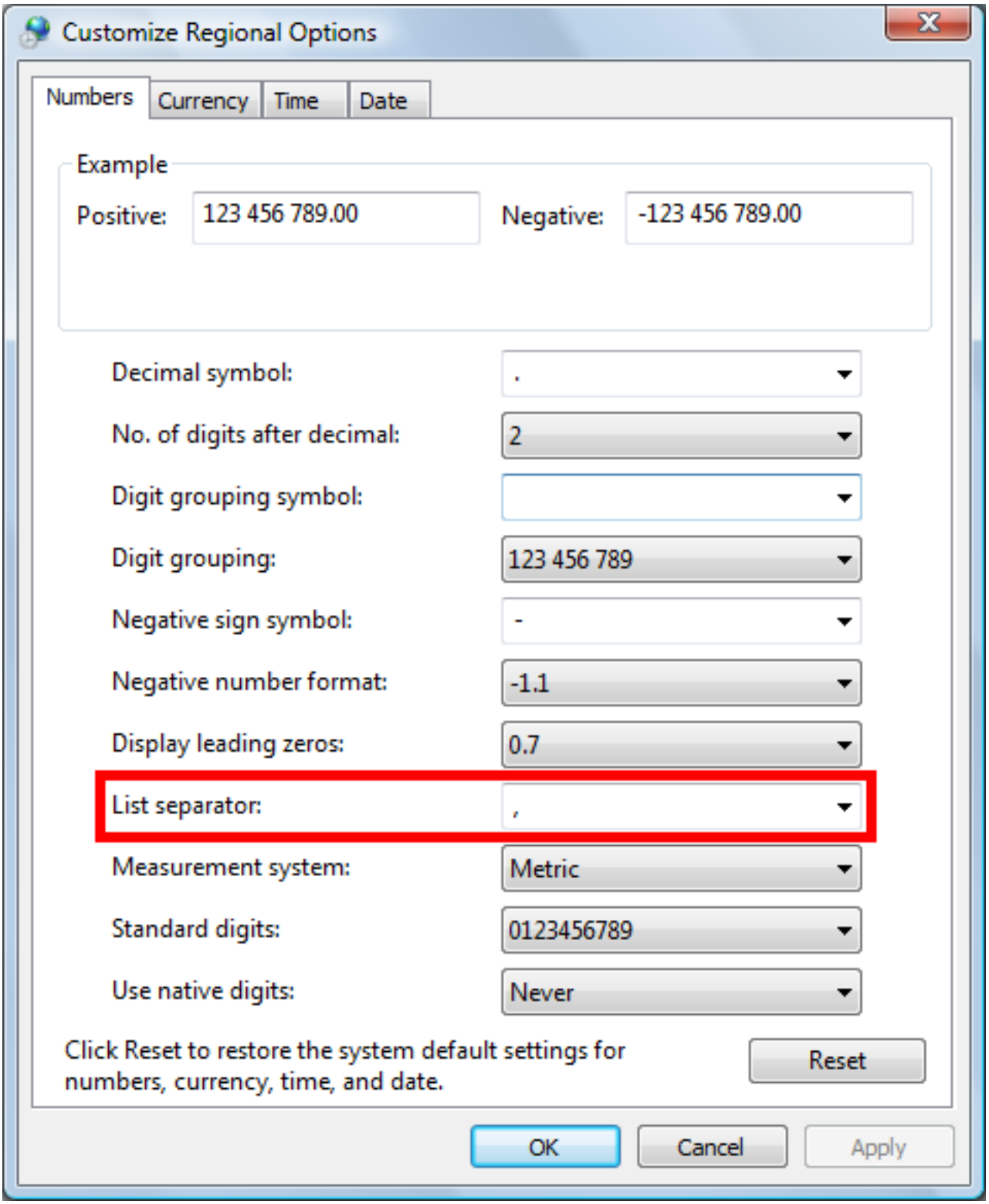


Figure 1.944



1.21.26 Table Styles

Command:TABLESTYLE (6.29.6) The appearance of atableis controlled by atablestyle. Each drawing contains theStandardtable style, which cannot be deleted. You can modify theStandardtable style to fit your needs or create your own table styles. Table styles are created and edited in theTable Stylescategory of theDrawing Explorer (1.14.2)dialog. In a table there are 3 cell types:Data,HeaderandTitle. Each cell type has its own cell style settings. By default all cells in a row share the same cell style settings, but you can edit the settings of each cell afterwards.

To open the Table Styles Explorer

Do one of the following:

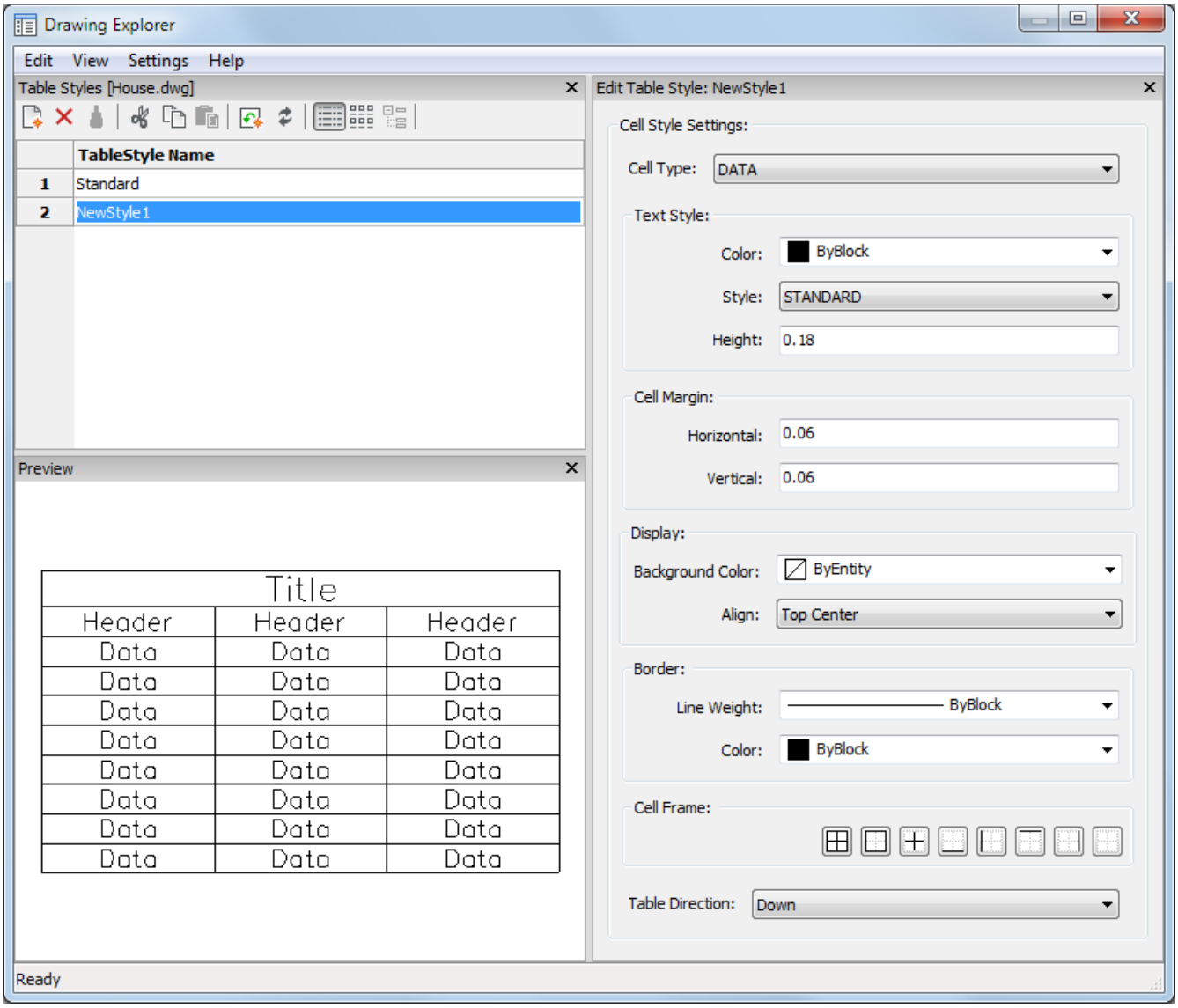
- ChooseDrawing Explorer > Table Styles...in theToolsmenu.
- Typetablestylein the command bar, then press Enter
- Click theExplore Table Stylestool button ( ) on theInsert Table (1.21.25)dialog.

To create a table style

1. Open theDrawing Explorer - Table Stylesdialog displays.
2. In theDrawing Explorer - Table Stylesdialog do one of the following:
   - ChooseNewin theEditmenu.
   - Click theNewtool button ( ).

   A new style is created using default settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us
3. Rename the style, replacing theNewStyle1default name.
4. Select a cell type:Data,HeaderorTitle.
5. Define the cell style settings for the selected cell type.
6. Repeat the previous steps for each cell type.

Figure 1.945


Drawing Explorer
Edit View Settings Help
Table Styles [House.dwg]
TableStyle Name
1 Standard
2 NewStyle1
Preview
Title
Header Header Header
Data Data Data
Edit Table Style: NewStyle1
Cell Style Settings:
Cell Type: DATA
Text Style:
Color: ByBlock
Style: STANDARD
Height: 0.18
Cell Margin:
Horizontal: 0.06
Vertical: 0.06
Display:
Background Color: ByEntity
Align: Top Center
Border:
Line Weight: ByBlock
Color: ByBlock
Cell Frame:
Table Direction: Down
Ready

To edit a table style

1. Open theDrawing Explorer - Table Stylesdialog displays.
2. In theDetailspane of theDrawing Explorer - Table Stylesdialog, select the table style you want to edit.
3. Select a cell type:Data,HeaderorTitle.
4. Edit the cell style settings for the selected cell type.
5. (option) Repeat the previous steps for another cell type.

To apply a style to a table

SeeTo edit a table using the Properties bar (1.21.25).

To delete a table style

1. Open theDrawing Explorer - Table Stylesdialog displays.
2. Select the table style you want to delete. Press and hold the Ctrl key to select multiple table styles.
3. Do one of the following:
   - Right click, then chooseDeletein the context menu.
   - Click theDeletebutton in theDetailsmenu on theDrawing Explorer - Table Stylesdialog

NOTE	It is not possible to delete a table style that is being used in the drawing.

Tutorial: BricsCAD Spreadsheet Import Techniques Click here (1.2.9) to watch

1.21.27 Annotative Entities

https://www.youtube-nocookie.com/embed/BKsyXQ0X1hQ Annotations are all entities in your drawing which are not a model or geometry, such as dimensions, texts, symbols, etc. A common problem with annotations is that they are often scale dependent. Some annotations must be scaled to the plot scale, or are meant to be shown at certain scales only. Annotative styles and entities change the way in which they are displayed (size, location, and orientation) depending on the current scale factor and the viewport. The following entities can have the annotative property:

- text and multiline text (1.21.30)
- hatches (1.21.29)
- blocks and attributes (1.21.31)
- dimensions and tolerances (1.21.32)
- leaders and multileaders (1.21.33)

The following styles can have the annotative property:

- text styles (see the STYLE (6.27.53) command)
- dimension styles (see the DIMSTYLE (6.11.20) command)
- multileader styles (see the MLEADERSTYLE (6.21.25) command)

NOTES

- Entities that are created using an annotative style, get the current annotation scale assigned automatically.
- When an annotative style is applied to a non-annotative entity, the annotative property is not assigned automatically.

Commands

ANNORESET: Resets all scale representations of the selected annotative entities to the location of their current scale representation, thus synchronizing all scale representations. ANNOUPDATE (6.4.9): Updates the selected annotative entities to match their current style definition. OBJECTSCALE (6.23.1): Adds or removes supported scales for annotative entities through a dialog box. -OBJECTSCALE (6.23.2): Adds or removes supported scales for annotative entities at the command line. SCALELISTEDIT (6.27.7): Edits the scale list of a drawing, used in **Plot Scale** section of the **Print** and **Page Setup** dialogs and the **Standard scale** property of a paper space viewport. Defines the available annotation scales for annotative entities. -SCALELISTEDIT (6.27.8): Edits the scale list of a drawing at the command line.

System variables and user preferences

ANNOALLVISIBLE: Toggles the display of annotative enitities that do not support the current annotation scale. ANNOTATIVEDWG: Specifies whether or not the drawing behaves as an annotative block when inserted into another drawing. CANNOSCALE: Sets the name of the current annotation scale for the current space. Only scales that exist in the current scale list are accepted. The CANNOSCALE setting is saved with model space and with each layout viewport in the various layouts. When you create an annotative entity the current annotative scale is applied automatically. CANNOSCALEVALUE: Displays the value of the current annotation scale. E.g. the value of the 1:2 scale is 0.5. HPANNOTATIVE: Controls whether new hatch entities are annotative. MSLTSCALE: If ON, linetypes in model space are scaled to the current annotation scale. A regen is needed when you toggle the value of MSLTSCALE. This variable is the equivalent of the PSLTSCALE (Paperspace linetype scale) variable in paper space. MSOLESCALE: Controls the size of an OLE object containing text when pasted in model space. The MSOLESCALE system variable only affects the initial scaling. Existing OLE objects are not affected if the variable is modified. SAVEFIDELITY: Controls whether the current display of the drawing is preserved when opened in a program that does not support annotative entities. SELECTIONANNODISPLAY: Controls whether all scale representations display when an annotative entity is selected.

Setting the current annotation scale

To set the current annotation scale, do one of the following:

- Click the **Annotation Scale** field under **Misc** in the Properties Bar (1.12.4), then press the down-arrow button and select a scale in the list.
- Right click the **Annotation Scale** field in the Status Bar (1.12.2), then select a scale in the list. If the **Annotation Scale** field is not available, click the down arrow at the right hand side of the Status Bar and choose **Annotation Scale** in the list.

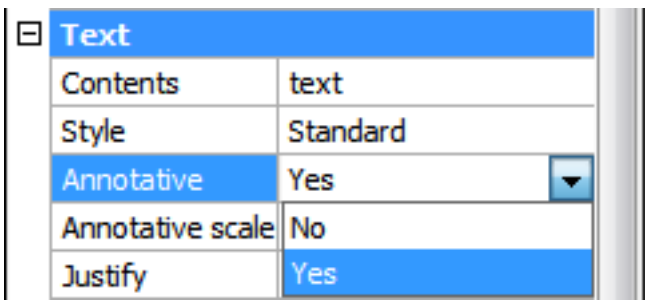
NOTE	Only scales that exist in the **Scale List** of the drawing are available. Use the SCALELISTEDIT (6.27.7) command to edit the **Scale List** .

To make an annotation entity annotative

Annotation entities which are created using a non-annotative style can be made annotative using the following procedure:

1. Select the annotation entity.
2. In the **Properties Bar**, click the **Annotative** property.
3. Select **Yes**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/A
The current annotation scale is applied to the selected entity.

Figure 1.946



1.21.28 Annotation Scales

Commands : OBJECTSCALE (6.23.1) and SCALELISTEDIT (6.27.7) Annotation scales are used to calculate the scaling factor for annotative entities. It defines the height of annotative text entities or the overall scaling of annotative blocks. In general the annotation scale allows to create annotative entities in model space at the correct paper space size automatically.

- Texts, MTexts, Dimensions, MLeaders: The current annotation scale is applied upon creation if the current style is annotative.
- Hatches and Gradient Fills: The current annotation scale is applied if the HPANNOTATIVE (1.13.11) system variable is ON.
- Blocks: The current annotation scale defines the scaling of annotative blocks upon insertion.

Annotative entities can support multiple annotation scales. When the SELECTIONPREVIEW (1.13.11) system variable is active or when using the Quad cursor menu (1.12.5), an icon displays next to the highlighted entity indicating whether a single annotation scale () applies to an annotative entity or multiple ()

To add or remove annotation scales to annotative entities

Use the following procedure to:

- Check which annotation scales currently apply.
- Add annotation scales.
- Remove annotations scales.

1. Select the annotative entity or entities.
2. In the Properties Bar (1.12.4), select the **Annotative Scale** field.
3. Click the **Browse** button () of the **Annotative Scale** field.

The **Annotation Object Scale** dialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/
4. (option) If multiple entities were selected, choose whether to list all scales for the selection or the common scales only. 5. (option) Do one of the following:

- Select one or more scales in the list, then click the **Delete** button. The selected scale(s) are removed from the list.
- Click the **Add...** button, then select one or more scales in the list and click the **OK** button. The selected scale(s) are added are added to the list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/1

6. Click the OK button to close the **Annotation Object Scale** dialog. The changes are applied to the selection set.

NOTE	Only scales that exist in the **Scale List** of the drawing are available. Use the SCALELISTEDIT (6.27.7) command to edit the **Scale List** .

Tutorial: Making Smarter Drawings with Annotative Scales Click here (1.4.14) to watch

Figure 1.947

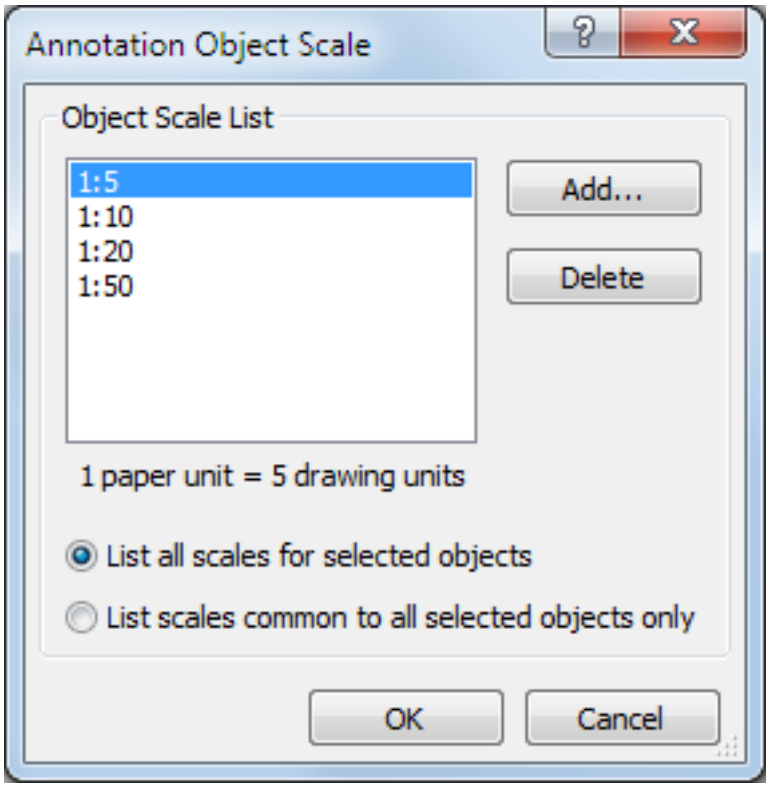
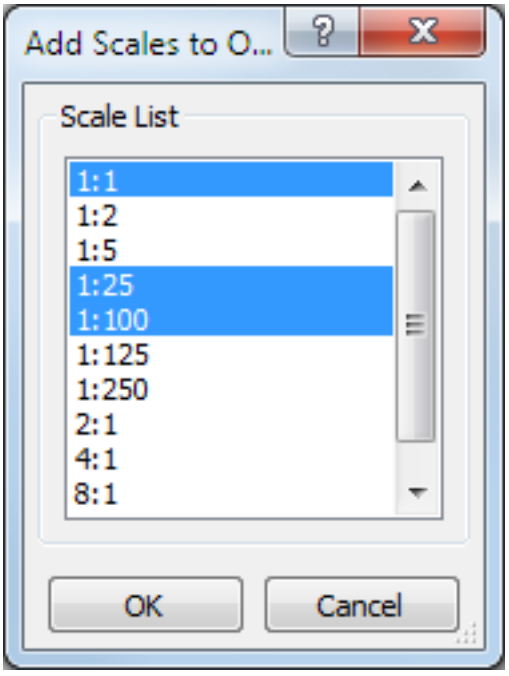


Figure 1.948



1.21.29 Annotative Hatches and Gradient Fills

Commands: HATCH (6.16.1), -HATCH (6.16.2), GRADIENT (6.15.17) The HPANNOTATIVE (1.13.11) system variable controls whether new hatch or gradient entities are annotative. HPANNOTATIVE is OFF by default. The system variable is not saved and is reset to its default value when BricsCAD closes.

To set the HPANNOTATIVE system variable

Do one of the following:

- In the **Settings** dialog, under **Drafting / Entity Creation / Hatches**.
- Check the **Annotative** option on the **Hatch and Gradient** dialog box of theHATCH (6.16.1)or GRADIENT (6.15.17) command.
- Choose the **Annotative** option in the command prompt of the-HATCH (6.16.2)command.

To make a hatch or gradient entity annotative

1. Select the hatch or gradient fill. The properties of the selected entity display in the Properties Bar (1.12.4).
2. In the Properties Bar, select the **Annotative** property.
3. Choose **Yes** from in the options list. The current annotation scale is applied to the selected hatch or gradient entity and the hatch is scaled accordingly.

To remove the annotative property

1. Select the hatch or gradient fill. The properties of the selected entity display in the Properties Bar (1.12.4).
2. In the Properties Bar, select the **Annotative** property.
3. Choose **No** from in the options list.

1.21.30 Annotative Texts

Commands: TEXT (6.29.12), MTEXT (6.21.35) and STYLE (6.27.53) Text entities are created using the current text style:

- If the text style is annotative, the text entity gets the annotative property automatically and the current annotation scale is assigned.
- If the text style is not annotative, you need to make the text entity annotative manually.

Annotative texts have two height properties:

- **Model text height** : the display height on the screen in model space, with respect to the current annotation scale.
- **Paper text height** : the height on paper if printed on the supported annotation scale.

To make a text entity annotative

1. Select the text entity. The properties of the selected text entity display in the Properties Bar (1.12.4).
2. In theProperties Bar (1.12.4), select the **Annotative** property.
3. Choose **Yes** from in the options list. The current annotation scale is assigned. The text is not scaled: the current text height becomes the **Model text height**.

To remove the annotative property

1. Select the text entity. The properties of the selected text entity display in theProperties Bar (1.12.4).
2. In the Properties Bar, select the **Annotative** property.
3. Choose **No** from in the options list. If the text entity supports multiple annotation scales, the text is scaled back to its initial scale.

1.21.31 Annotative Blocks

Commands: BLOCK (6.6.11) and EXPBLOCKS (6.13.11) The annotative property of a block is either set when creating the block definition (1.22.5) or afterwards, in the **Drawing Exlorer / Blocks** dialog. Annotative blocks can have an optional property to always match the orientation of a layout, independent from the rotation of the viewport content.

To make a block definition annotative

1. Open the Drawing Explorer / Blocks (1.22.8) dialog.
2. If necessary, choose Detail View (1.22.8). Annotative block definitions have the **Annotative** icon () displayed.
3. Click in the **Annotative** field of a block definition to toggle the annotative property on/off.

To match the orientation of a block to a layout

1. Open the Drawing Explorer / Blocks (1.22.8) dialog.
2. If necessary, choose Detail View (1.22.8).
3. Click the checkbox in the **Match Orientation** field of a block definition to toggle the **Match Orientation** property on/off.

1.21.32 Annotative Dimensions

Dimension entities are created using the current dimension style (1.21.6):

- If the dimension style is annotative, the dimensions get the annotative property automatically and the current annotation scale is assigned.
- If the dimension style is not annotative, you need to make the dimensions annotative manually.

NOTE

- The **Dim Scale Overall** (DIMSCALE (1.13.11)) setting is not editable for annotative dimension styles, and is neglected for annotative dimensions.
- It is recommended to set **Dim Scale Overall** = 1 for annotative dimension styles.

To make a dimension entity annotative

1. Select the dimension entity. The properties of the selected dimension entity display in the Properties Bar (1.12.4).
2. In the Properties Bar (1.12.4), select the **Annotative** property.
3. Choose **Yes** from in the options list. The current annotation scale is assigned. The dimension entity is scaled with respect to the current annotation scale.

To remove the annotative property

1. Select the dimension entity. The properties of the selected dimension entity display in the Properties Bar (1.12.4).
2. In the Properties Bar, select the **Annotative** property.
3. Choose **No** from in the options list. The dimension entity is scaled with respect to the current value of the **Dim Scale Overall** setting.

1.21.33 Annotative Leaders

Under Construction.

1.22 Blocks, Attributes, External References and Components

1.22.1 Working with attributes

An attribute is a particular entity that you can save as part of a block definition. Attributes consist of text-based data. You can use attributes to track such things as part numbers and prices. Attributes have either fixed or variable values. When you insert a block containing attributes, the program adds the fixed values to the drawing along with the block, and you are prompted to supply any variable values. After you insert blocks containing attributes, you can extract the attribute information to a separate file and then use that information in a spreadsheet or database to produce a parts list or bill of materials. You can also use attribute information to track the number of times a particular block is inserted into a drawing. Attributes can be visible or hidden. Hidden attributes are neither displayed nor plotted, but the information is still stored in the drawing and written to a file when you extract it.

General procedure to work with attributes

1. Define the attributes.
2. Create a block.

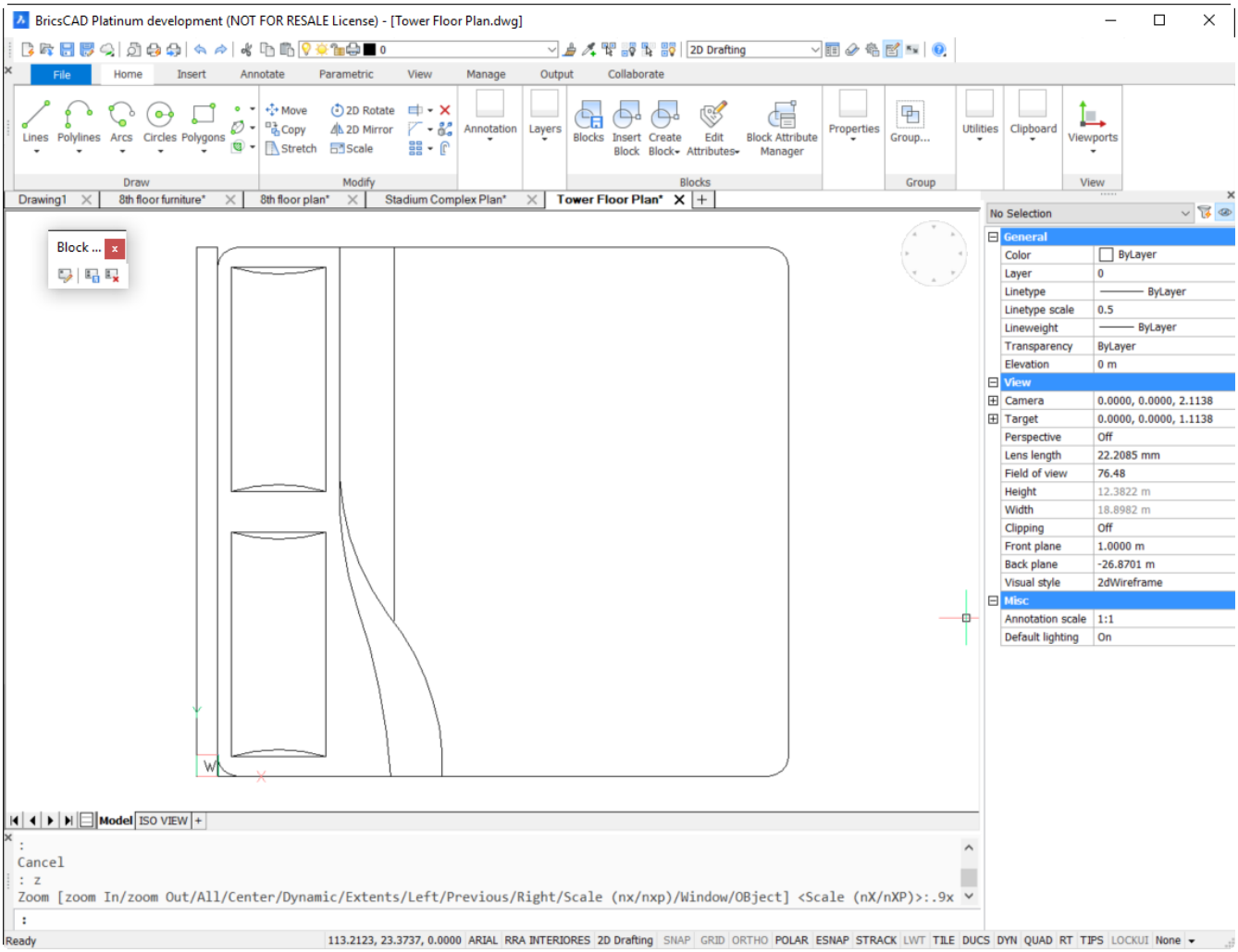
You can attach attributes to a block. Include the attributes when the program prompts you to select the entities to be included in the block definition. If a block contains attributes, the program prompts you each time you insert the block, so you can specify different values for the attributes each time you insert it into a new drawing.

1. Insert the block in a drawing.
2. (option)Edit the attributes in a block.
3. Extract attribute information.

1.22.2 Blockify

Command: BLOCKIFY (6.6.13) BricsCAD includes two Blockify tools, one for 2D entities and one for 3D solids. The Blockify tools enable you to easily convert repetitive sets of entities to block definitions. You simply select a set of 3D solids. BricsCAD creates a block definition, searches the drawing for identical sets of entities and automatically replaces them with the block reference. Youtube

Figure 1.949



1.22.3 Block editor

Commands: BEDIT (6.6.7) and BCLOSE (6.6.4) You can access the Block editor with the BEDIT command or by double-clicking on the block you wish to edit. When using the block editor, BricsCAD displays the block entities in a temporary drawing session. It includes all the tools you need to create and edit a block definition. To exit the block editing session, use the BCLOSE command and choose whether to save or discard the changes. If you select the Save tool while in the Block editing session, BricsCAD saves the changes to the block definition and exits the block editor. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Blocks-Attributes-ExternalReferences/Blockeditor.png})

1.22.4 Working with Blocks

Define Blocks

The process to define a block using the BLOCK command is the same in BricsCAD as in AutoCAD®. It displays the Block Definition dialog box with the necessary controls to create a new block definition in the current drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Blocks-Attributes-ExternalReferences/HHBricsCAD-Working-with-Blocks-01.png}) Note: The Block Definition dialog box in BricsCAD doesn't have options for Hyperlink, Quick select or Block Editor. You can, however, attach hyperlinks, select objects, and edit blocks using alternate methods. The familiar WBLOCK command allows you to write out a block definition as a separate drawing file, external to the current drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Blocks-Attributes-ExternalReferences/HHBricsCAD-Working-with-Blocks-02.png}) BricsCAD doesn't have a Block Editor. Instead, you create and edit block geometry using the drawing editor with all the tools and functionality it has to offer available.

Figure 1.950

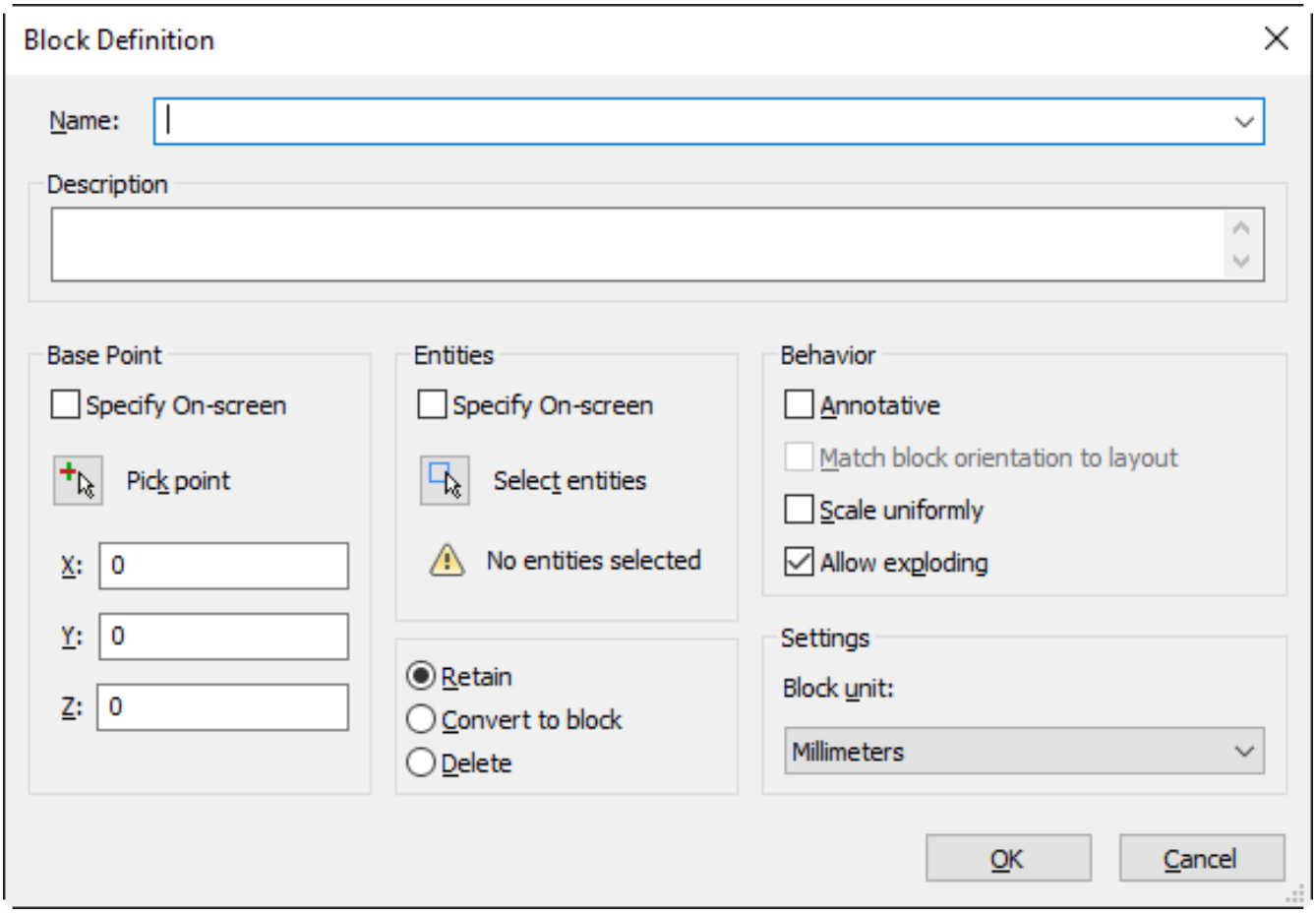
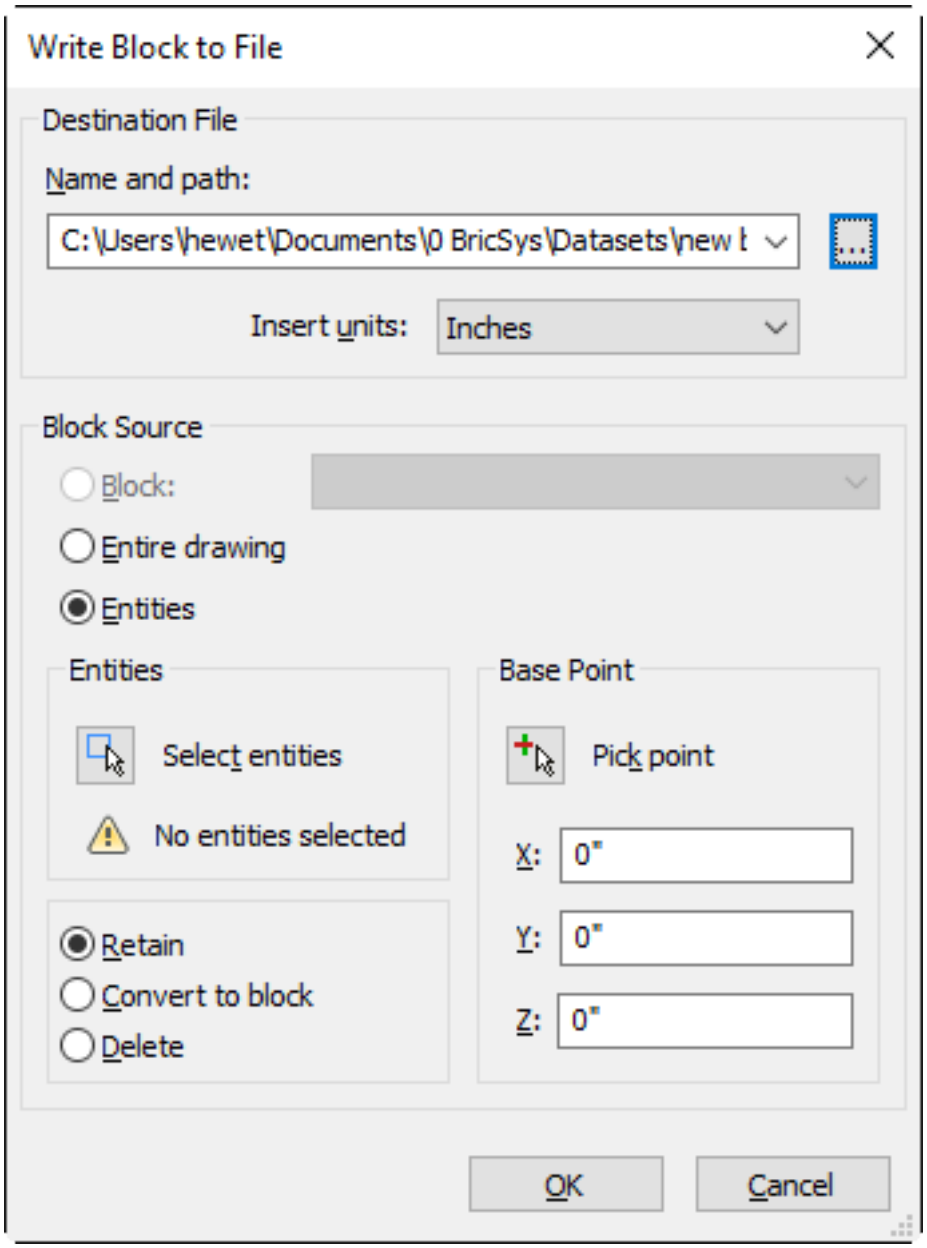

Block Definition
Name:
Description
Base Point
Specify On-screen
Pick point
X: 0
Y: 0
Z: 0
Entities
Specify On-screen
Select entities
No entities selected
Retain
Convert to block
Delete
Behavior
Annotative
Match block orientation to layout
Scale uniformly
Allow exploding
Settings
Block unit:
Millimeters
OK
Cancel

Figure 1.951


Write Block to File
Destination File
Name and path:
C:\Users\hewet\Documents\0 BricSys\Datasets\new b
Insert units:
Inches
Block Source
Block:
Entire drawing
Entities
Entities
Select entities
No entities selected
Base Point
Pick point
X:
0"
Y:
0"
Z:
0"
Retain
Convert to block
Delete
OK
Cancel

Insert Blocks

You can select a block to insert in your drawing from the Insert Block dialog box. It's easily accessible from the Insert tab on the Quad. You can also access it via other methods including the INSERT command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Blocks-Attributes-ExternalReferences/HHBricsCAD-Working-with-Blocks-03.png}) The Insert Block dialog box will look familiar to you as a former AutoCAD user. It includes typical options to specify the insertion point, scale and rotation angle for the block. (\vref {fig:C:/Dev/Task-2448/images/staticbrics
Attributes-ExternalReferences/HHBricsCAD-Working-with-Blocks-04.png}) Note: The Insert Block dialog box doesn't include an option to insert a block using geographic data. The Insert Block dialog box doesn't include a preview image. Instead, BrisCAD offers a more intuitive way to preview, insert, and manage your blocks!

Figure 1.952

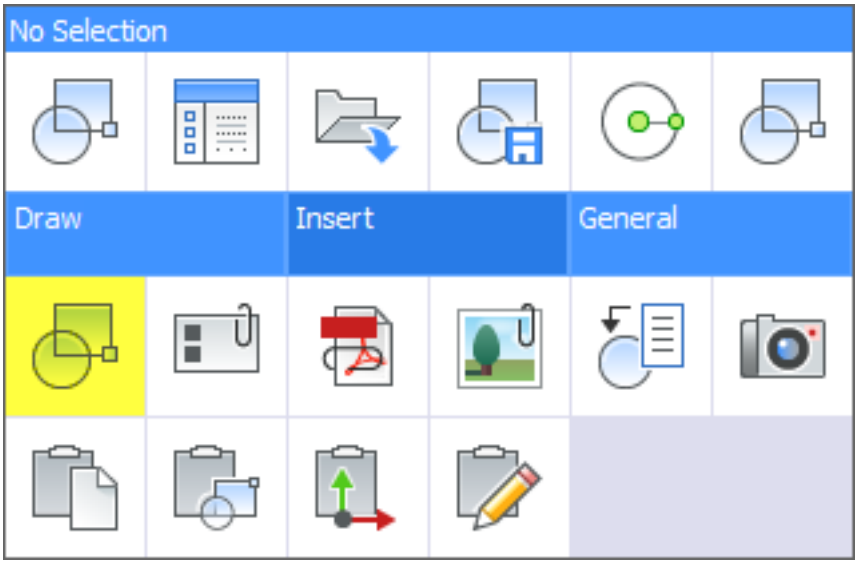
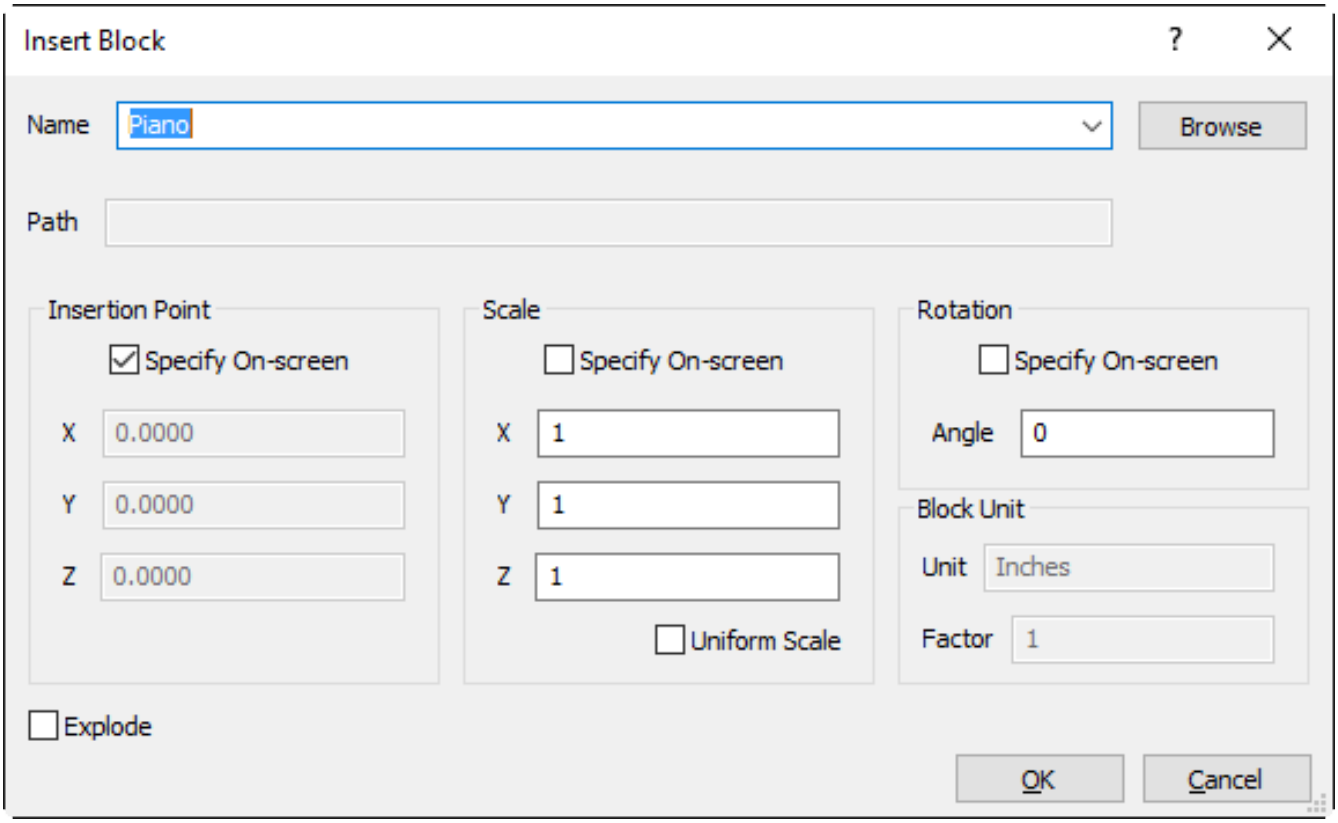


Figure 1.953



Manage Blocks in Drawing Explorer

You may remember that Drawing Explorer offers a central location to manage all the style definitions in your drawing. In fact, Drawing Explorer does much more than that. It centralizes all named drawing content, including blocks, in a single dialog box. Access Drawing Explorer from the General menu on the Quad. It's also accessible via other methods including the EXPLORER command. If you launch Drawing Explorer with the EXPBLOCKS command (XB alias), it automatically displays the panels for Blocks. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/
Attributes-ExternalReferences/HHBricsCAD-Drawing-Style09.png}) Drawing Explorer lists all block definitions in the drawing and includes relevant information and controls. You can see how many instances of a block are in the drawing and you can modify basic block properties. For example, you can specify whether a block is Annotative or Explodable. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Blocks-Attributes-ExternalReferen
Working-with-Blocks-05.png}) Another option, unique to BricsCAD, ensures the block always faces the camera. This can be particularly useful (and fun) for 3D visualization. For example, I created a block with an image of my photo and inserted it in a 3D model. As I orbit around the model, the block/image always faces forward. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Blocks-Attributes-ExternalReferences/HHBricsCAD-
Working-with-Blocks-06-1024x257.png}) The right-click menu and toolbar offer additional controls. You can easily create a new block definition, delete a block definition and all its insertions, and purge an unused block definition. Insert blocks and even add them to the current tool palette. Anything you want to do with blocks is easily accessible from the Drawing Explorer! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Blocks-
Attributes-ExternalReferences/HHBricsCAD-Working-with-Blocks-07.png})

Tutorial: Manage blocks with the Drawing Explorer Click here (1.4.11) to watch

Figure 1.954

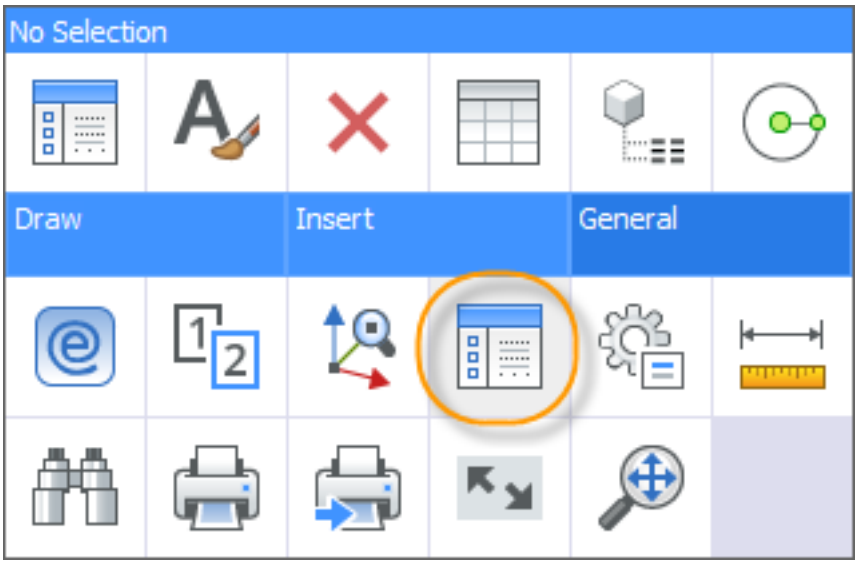


Figure 1.955

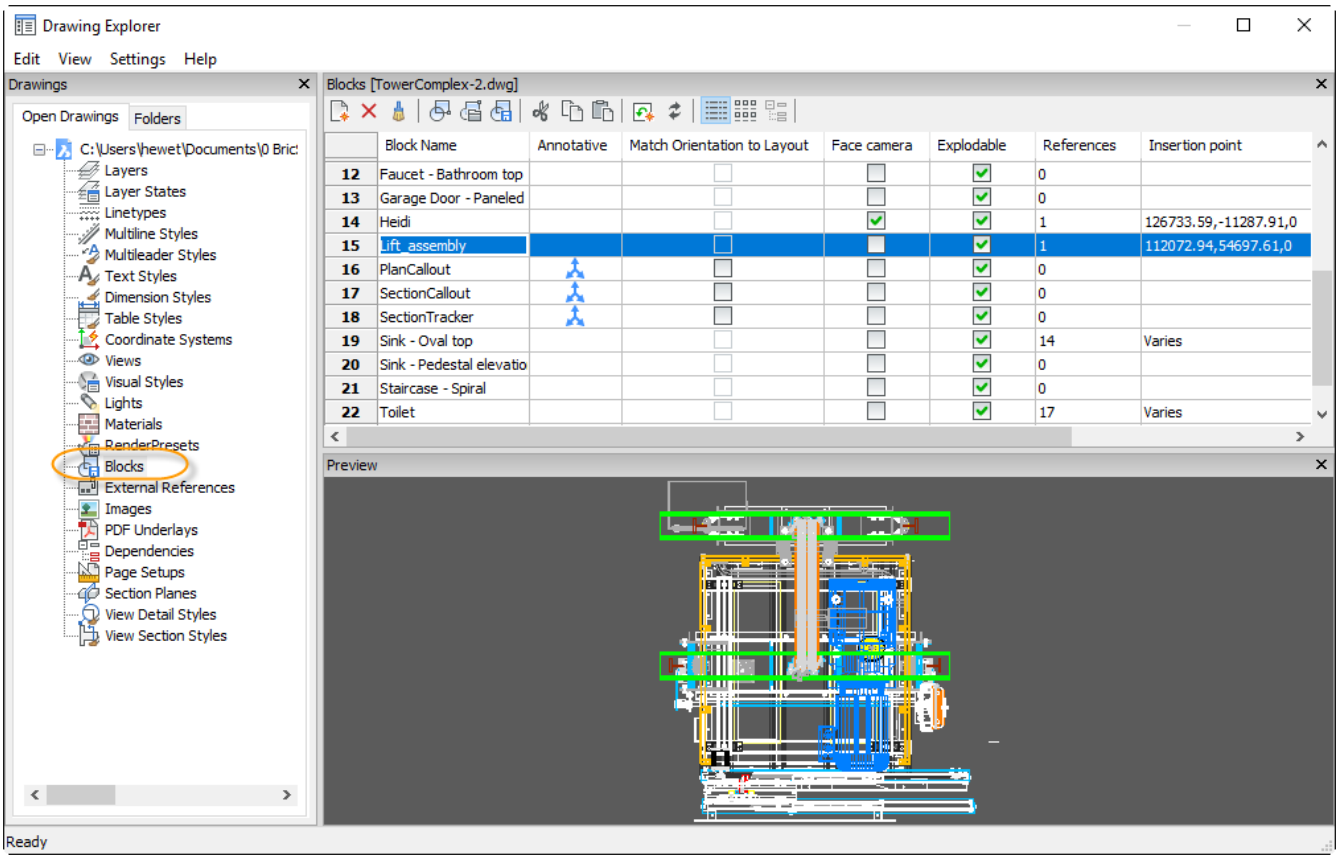


Figure 1.956



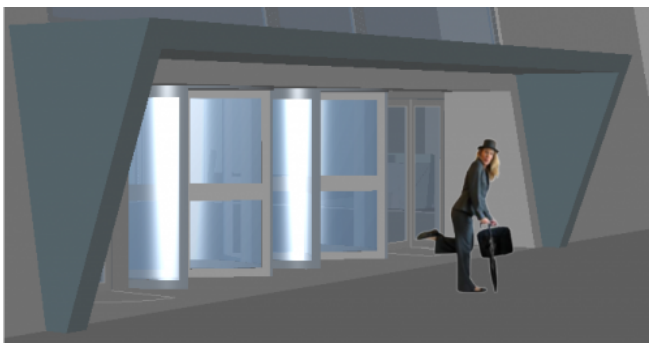
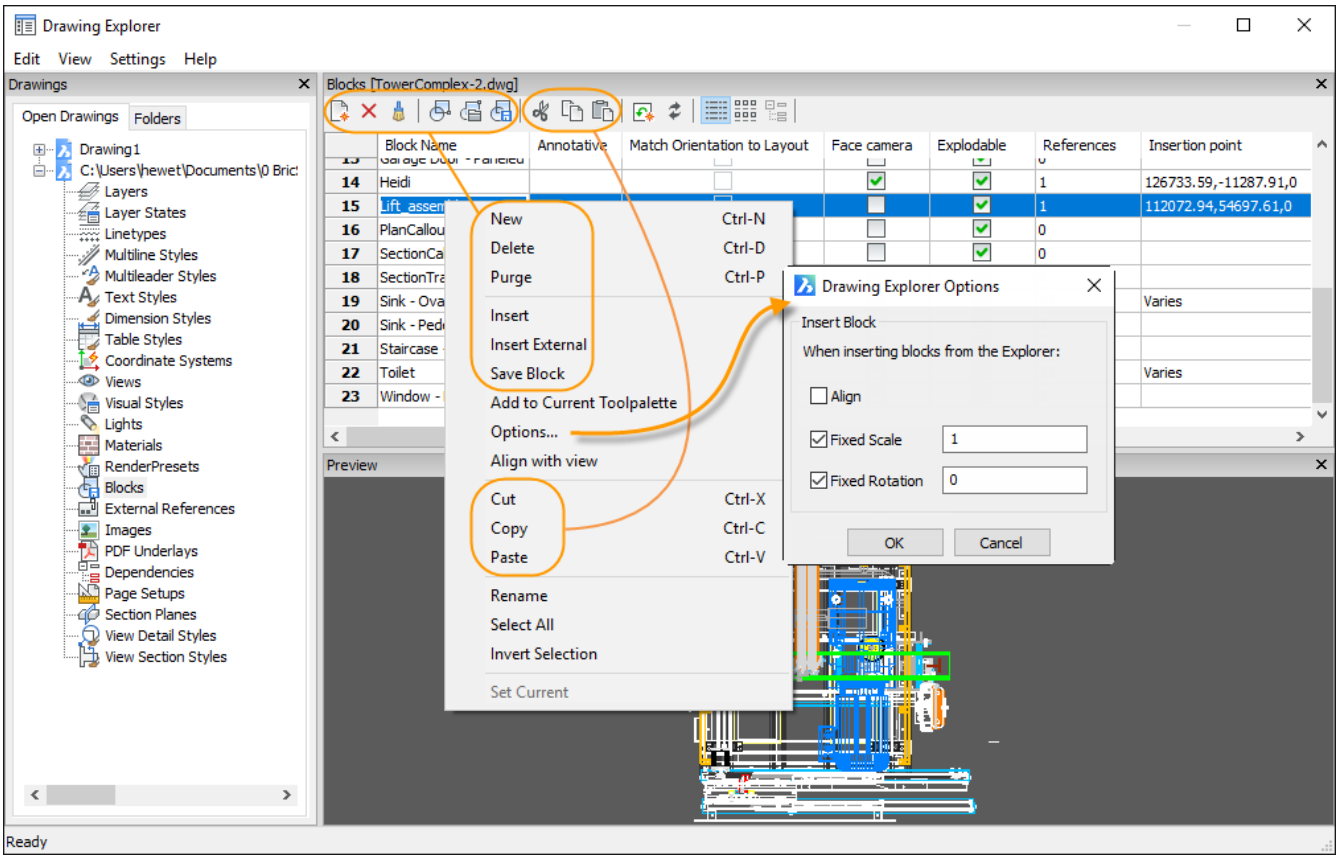


Figure 1.957


Drawing Explorer
Edit View Settings Help
Drawings
Open Drawings Folders
Drawing1
Layers
Layer States
Linetypes
Multiline Styles
Multileader Styles
Text Styles
Dimension Styles
Table Styles
Coordinate Systems
Views
Visual Styles
Lights
Materials
RenderPresets
Blocks
External References
Images
PDF Underlays
Dependencies
Page Setups
Section Planes
View Detail Styles
View Section Styles
Blocks [TowerComplex-2.dwg]
Block Name
Annotative
Match Orientation to Layout
Face camera
Explodable
References
Insertion point
14 Heidi 1 126733.59,-11287.91,0
15 Lift_assem 1 112072.94,54697.61,0
16 PlanCallou 0
17 SectionCa 0
18 SectionTra
19 Sink - Ova Varies
20 Sink - Ped
21 Staircase
22 Toilet Varies
23 Window -
New Ctrl-N
Delete Ctrl-D
Purge Ctrl-P
Insert
Insert External
Save Block
Add to Current Toolpalette
Options...
Align with view
Cut Ctrl-X
Copy Ctrl-C
Paste Ctrl-V
Rename
Select All
Invert Selection
Set Current
Drawing Explorer Options
Insert Block
When inserting blocks from the Explorer:
Align
Fixed Scale 1
Fixed Rotation 0
OK
Cancel
Preview
Ready

1.22.5 Creating Blocks

Commands: BLOCK (6.6.11), -BLOCK (6.6.12), WBLOCK (6.33.1), PASTEBLOCK (6.24.9)and SECTIONPLANETOBLOCK (6.27.14) The **Block** command groups entities into a block using a dialog box. The **-Block** commands prompts you in the command bar. The **Wblock** command writes blocks and other drawing parts to a separate drawing file (short for "write block"). The **Pasteblock** command pastes entities from the **Clipboard** as a block in the current drawing (see Paste a selection as a block (1.23.16)). The **SectionPlaneToBlock** command saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

General procedure to create a block

- Specify the name.
- Specify the insertion point. The insertion point is the base point for the block and serves as the reference point when you later insert the block into a drawing.
- Select entities.

To create a block in a drawing

1. Do one of the following:

   - Click the **Create Block** tool button ( ) on the Tools toolbar.
   - Choose **Create Block** in the Tools menu.
   - Type **block** in the command bar, then press Enter.

   The Block Definition dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sou

2. Type a name for the new block in the **Name** field.

3. Do one of the following to specify the **Base Point**:

   - Check the **Specify On-screen** option. You will be prompted to specify the base point after clicking the **OK** button.
   - Click the **Pick point** button ( ). The Block Definition dialog closes temporarily to let you pick a point in the drawing.
   - Type the coordinates of the base point in the X-, Y- and Z-field.

4. Do one of the following to select the entities to be included in the block definition:

   - Check the **Specify On-screen** option. You will be prompted to select the entities after clicking the **OK** button.
   - Click the **Select entities** button ( ). The Block Definition dialog closes temporarily to let select the entities in the drawing.

5. Set the block properties:

   - **Annotative**: creates an annotative block (1.21.31).
   - **Match block orientation to layout**: if checked, the orientation of an annotative block (1.21.31) is preserved in a rotated viewport.
   - **Scale uniformly**: if checked, the X-, Y- and Z-scaling cannot be set differently.
   - **Allow exploding**: if not checked, prevents the block from being exploded.
   - **Block unit**: set the insertion unit for the block (see also: To insert a block (1.22.6)).

6. Set the behavior of the selected entities after the block is created:

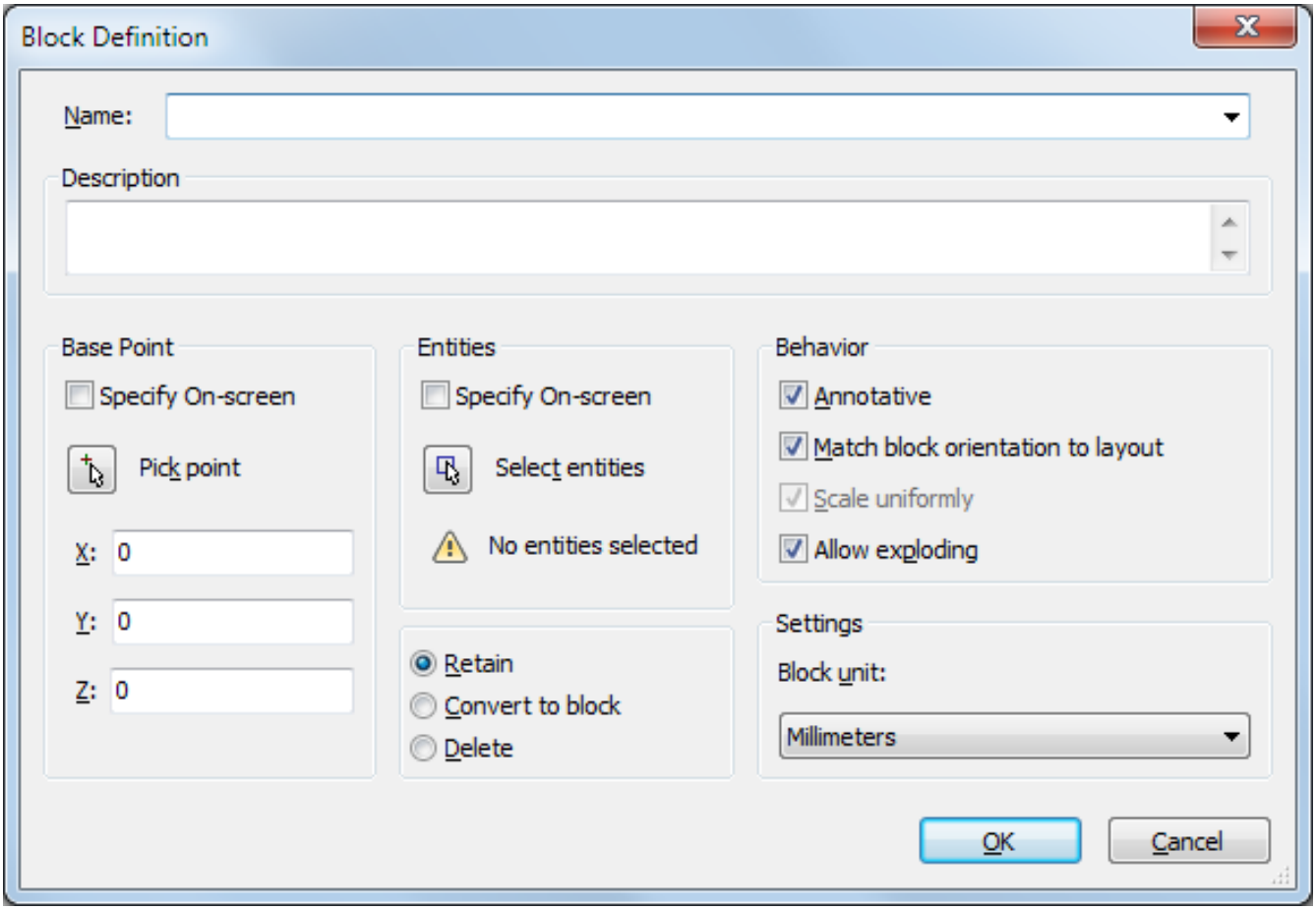
   - **Retain**: The selected entities remain in the drawing.
   - **Convert to block**:The selected entities are replaced by the newly created block.
   - **Delete**: The selected entities are deleted.

7. Click the **OK** button to create the block.

Note: If the **Delete** option in step 6 was set, the OOPS (6.23.7) command brings back the erased block entities.

Tutorial: Tip on how to create blocks quickly on the fly Click here (1.4.4) to watch

Figure 1.958



To create a block using the Blocks Explorer

1. Do one of the following:

   - Choose **Blocks...** in the Settings menu.
   - Type **explblocks** in the command bar, then press **Enter**.

   The Drawing Explorer - Blocks (1.22.8) dialog opens.

2. In the **Drawing Explorer** dialog do one of the following:

   - Click the **New** button ( ) on the Details toolbar.
   - Choose **New** in the Edit menu.

   The Drawing Explorer dialog closes. **You are prompted:** Insertion point for new block:

3. Specify the insertion point of the new block. You are prompted: Select entities for block.

4. Select the entities (1.23.6) for the new block.

5. Right click to create the block. The selected entities are removed from the drawing. The Drawing Explorer dialog reopens. The **Name** field of the new block is selected.

6. Type a name in the **Name** field, replacing the default **NewBlock1** name.

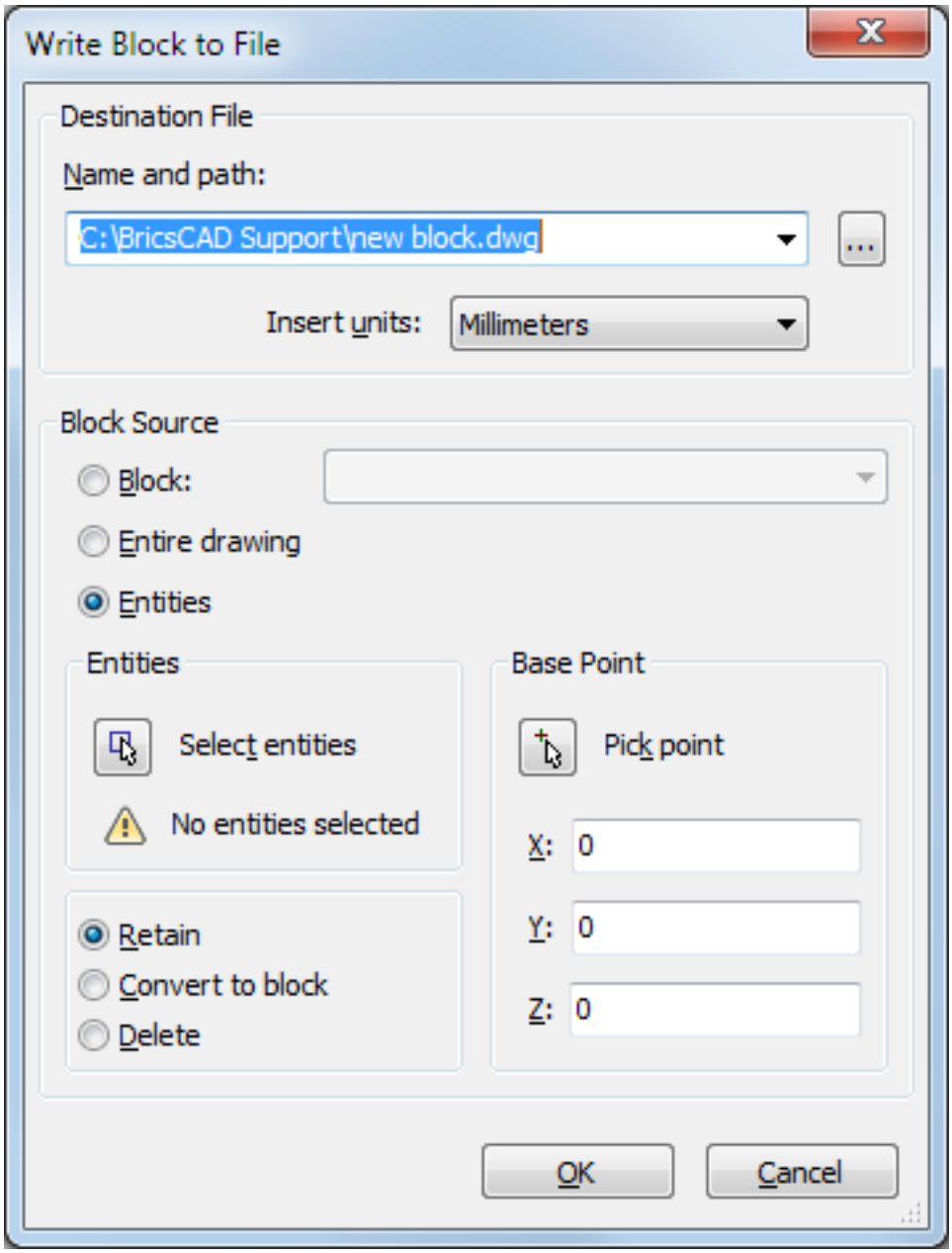
7. Close the Drawing Explorer dialog.

8. (option) Type **oops** in the command bar, then press Enter to bring back the selected block-entities.

To save a block to a separate file

1. Do one of the following:

Figure 1.959



- Click the **Save Block...** tool button ( ) on the Tools toolbar.
- Choose **Save Block...** in the Tools menu.
- Type **wblock** in the command bar, then press Enter.

The **Write Block to File** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/so

2. Click the browse button ( ) next to the **Name and path** field. The Save Block dialog opens.
3. On the Save Block dialog, select a folder and type a name in the **File name** field, then click the **Save** button. The Save Block dialog closes.
4. **Insert units**: set the insertion unit for the block (see also: To insert a block (1.22.6)).
5. To select the source entities for the new block, do one of the following:
   - Click the **Block** radio button, then select a block from the drop-down list.
   - Click the **Model** space radio button, to select all the entities in the model space of the current drawing.
   - Click the **Entities** radio button, then click the**Select Entities** button ( ). The **Write Block to File** dialog temporarily closes to let you select entities in the drawing. Right click to stop selecting entities. Click the **Pick point** button ( ) to define the **Base point** of the new drawing **Or** Type the coordinates of the base point in the X-, Y- and Z-field. Set the behavior of the selected entities after the block is created:

- **Retain**: The selected entities remain in the drawing.
- **Convert to block**: The selected entities are replaced by the newly created block.
- **Delete**: The selected entities are deleted.

6. Click the **OK** button to create the drawing file.

Note: If the **Delete** option in step 5 was set, the OOPS (6.23.7) command brings back the erased block entities.

1.22.6 Inserting Blocks

Commands: INSERT (6.17.13), -INSERT (6.17.14), INSERTALIGNED (6.17.15) and MINSERT (6.21.17) The **Insert** command lets you insert blocks through a dialog box. The **-Insert** command inserts blocks by prompting in the command bar. BricsCAD will lookup the block name in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the **SRCHPATH** system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>' The **Insertaligned** command inserts a block entity, with easy alignment on existing entities. The **Insertaligned** command is similar to the **Insert** command, but during the placement the block will be dynamically aligned with existing lines, polylines, arcs or circles. The block is aligned with the entity to which the cursor snaps. If no entity is snapped, the block is aligned with the X-axis of the WCS. The **Minsert** command Inserts a block as a rectangular array; combines the **-Insert** and **Array** commands (short for "multiple insertion"). You can choose to select an existing block definition or to insert an entire drawing as a block. When you insert a drawing, a new block definition is created in the current drawing. If you change the original drawing file, those changes have no effect on the current drawing unless you redefine the block by reinserting the changed drawing. When a block is inserted in a drawing, it is treated as a single entity.

Tutorial: Manage blocks with the Drawing Explorer Click here (1.4.11) to watch

General procedure to insert a block:

- Specify a block definition or drawing file.
- Specify the insertion point.
- Specify the scale.
- Specify the rotation angle.

NOTES

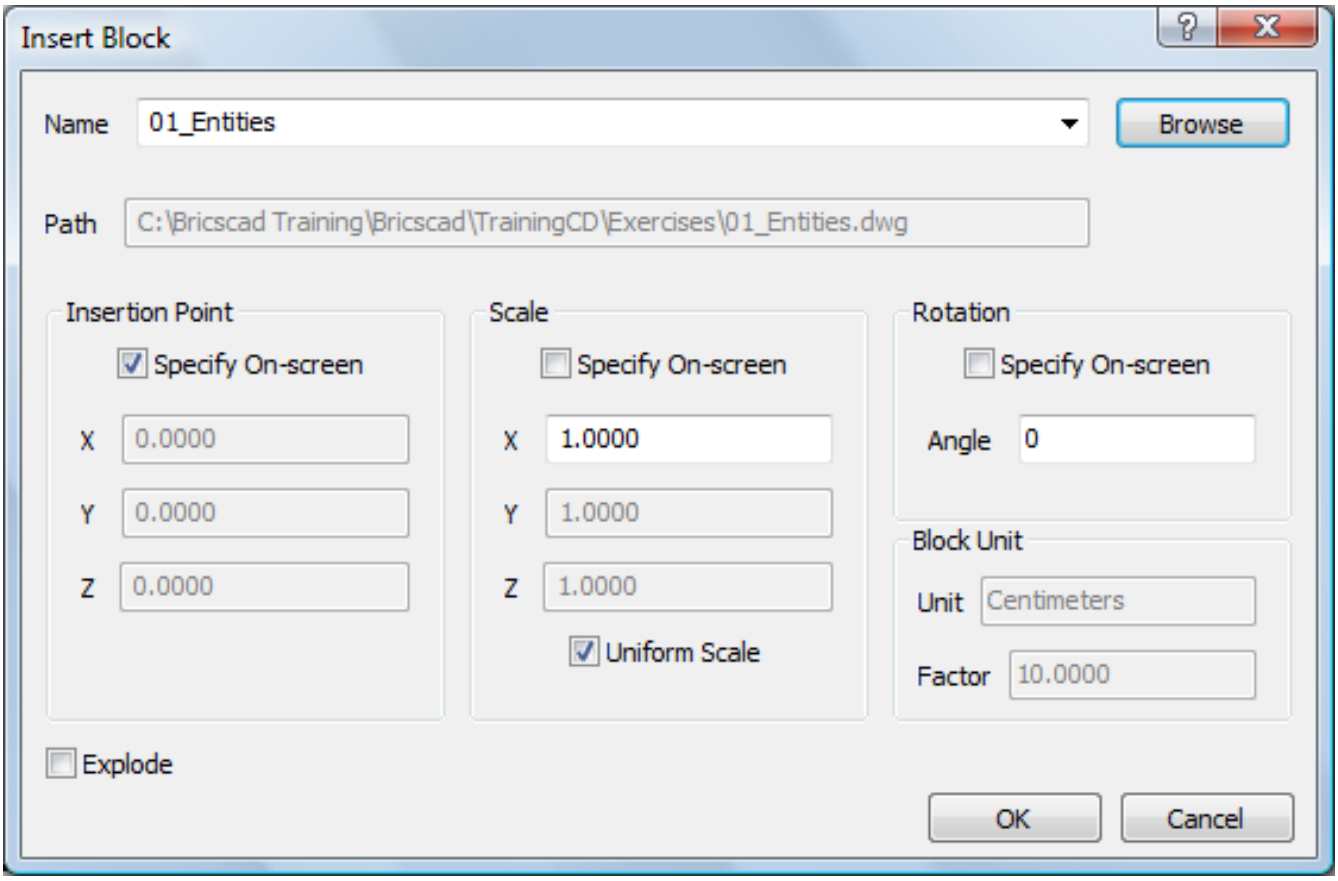
- When inserting a drawing as a block, the base point, as defined by the **INSBASE** system variable, of the inserted drawing is the origin point of the block. The **INSBASE** system variable is set by the BASE (6.6.3) command.
- If a block contains attributes you will be prompted to fill out the text for each attribute in the command bar.
- If the **DRAGOPEN** system variable is set to zero (OFF), drawings can be inserted as a block by dragging them from the **Windows Explorer** dialog into the current drawing; if **DRAGOPEN** = 1 (ON), the drawing will be opened instead.
- The **INSUNITS** system variable defines a drawing units value for automatic scaling when inserting blocks or attaching Xrefs (1.22.16).

In the **Insert Block** dialog:

- The **Unit** field under **Block Unit** is set by the **INSUNITS** (Insertion Units) system variable in the source drawing or the **Block Unit** property of a block definition (see To create a block in a drawing (1.22.5)).
- The **Factor** field expresses the relation between the value of the **INSUNITS** variable in the source drawing and the target drawing. E.g. if **INSUNITS** is Millimeters in the source and the target, the value of the **Factor** field is 0.1.
- If **INSUNITS** in the source drawing is **Unspecified** , **INSUNITSDEFSOURCE** is used instead.
- If **INSUNITS** in the target drawing is **Unspecified** , **INSUNITSDEFTARGET** is used instead.

The values of **INSUNITSDEFSOURCE** and **INSUNITSDEFTARGET** are saved in the registry and therefore apply to all drawings in which **INSUNITS** is unspecified (unitless).

Figure 1.960



To insert a block

1. Do one of the following

   - Click the **Insert Block...** tool button ( ) on the Insert toolbar.
   - Choose **Insert Block...** in the Insert menu.
   - Type **insert** in the command bar, then press Enter.

   The **Insert Block** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/1

2. In the **Insert Block** dialog, do one of the following:

   - Select an existing block definition in the **Name** list.
   - Click the **Browse** button and select a drawing file.

3. (option) Check the Explode (1.23.36) option to explode the block after insertion.

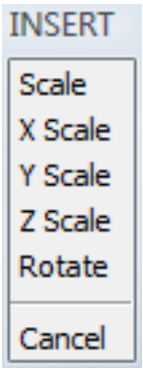
4. (option) Uncheck the **Specify On-Screen** option, then specify the **Insertion Point** by keying in the coordinates in the **X** , **Y** and **Z** fields.

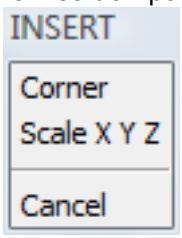
5. (option) Uncheck the **Specify On-Screen** option, then specify the **Scale** by keying in the scaling factors in the **X** , **Y** and **Z** fields. Check the **Uniform Scale** option to equal the **X** , **Y** and **Z** scale factors.

6. (option) Check the **Specify On-Screen** option to specify the Rotation angle when inserting the block.

7. Click the **OK** button. **You are prompted:** Insertion point for block. A prompt menu displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalReferences/ART/0102Inser

Figure 1.961



8. Specify the insertion point. **You are prompted:** Corner/XYZ/X scale factor <1.000000>: A prompt menu displays. 

9. Right click or press Enter to accept the default **X scale factor** . You are prompted: Y scale factor: <Equal to X scale (1.000000)>:
10. Right click or press Enter to set the **Y scale factor** equal to the **X scale factor** . The block is inserted.

To insert a block using the Drawing Explorer

Inserting internal blocks

1. Open the Drawing Explorer - Blocks (1.22.8) dialog.
2. (optional) Choose either **Detail View** ( ) or **Icon View** ( ).
3. (option) Edit the insert options:
   - Choose **Options...** in the Settings menu or select a block, then right click and choose **Options ...** in the context menu.
   - Set the insert options in the Drawing Explorer Option**s** dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys
4. Do one of the following: Select a block, then do one of the following:
   - Click the **Insert Block** button ( ) on the **Details** toolbar.
   - Right click and select **Insert** from the context menu
   - When in **Icon View**, double click the block.

   To select an external drawing, do one of the following:
   - Click the **Insert External** tool button ( ) on the **Details** toolbar.
   - Choose **Insert External** in the **Edit** menu.
   - Right click, then choose **Insert External** in the context menu.

   The Drawing Explorer - Blocks window closes. **You are prompted:** Multiple blocks/Scale/<Insertion point for block>: A prompt menu displays (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/
5. Follow the instructions in the command bar to insert the block. The Drawing Explorer - Blocks window re-opens.
6. (option) Insert more blocks.
7. Close the Drawing Explorer - Blocks window.

Figure 1.962

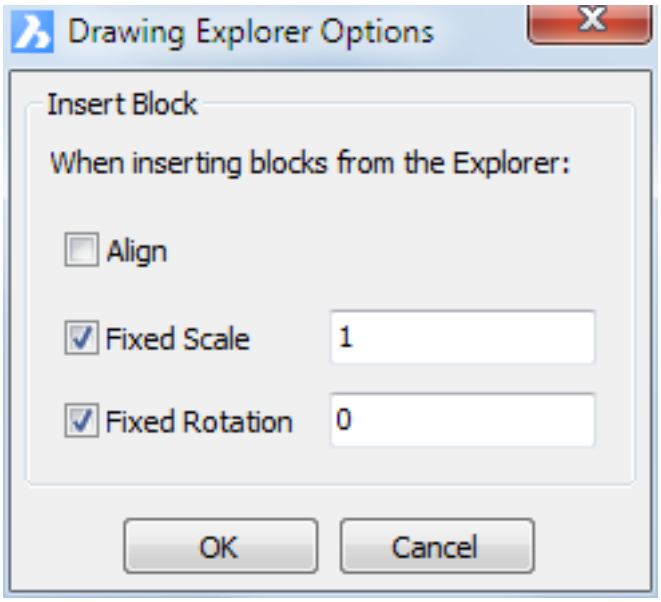
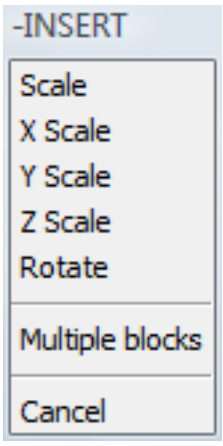


Figure 1.963



Inserting blocks from another drawing

1. Open the Drawings Explorer (1.14.2).
2. In the **Drawings** pane, click the Folders tab.
3. (option) If not yet available in the **Local Folders** tree, add the folder (1.14.2) of the drawing.
4. Expand the folder.
5. Click the **Expand** icon (+) in front of the drawing. The **Blocks** icon of the drawing displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalReferences/ART/0102Bloc
6. Click the **Blocks** icon. The blocks in the drawing display in the **Details** pane. If necessary, click the **Icon View** button ( ) in the Details toolbar to see thumbnail images of the blocks.
7. (option) Edit the insert options:
   - Choose **Options...** in the **Settings** menu or select a block, then right click and choose **Options ...** in the context menu.
   - Set the insert options in the Drawing Explorer Options dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys

   The available options are:

   - **Align**: If checked, launches the InsertAligned (6.17.15)command to insert the block. When the **Align** option is checked, the **Fixed Scale** and **Fixed Rotation** options are not available.
   - **Fixed Scale**: If checked, allows to insert the block at a fixed scale.
   - **Fixed Rotation**: If checked, allows to insert the block at a fixed rotation.
8. To insert a block, do one of the following:
   - Click the **Insert Block** button ( ) on the Details toolbar.
   - When in Icon View (1.22.8), double click the thumbnail image of the block.
   - When in Detail view (1.22.8), double click the number in front of the block name.

   The Drawing Explorer dialog closes temporarily to let you insert the block in the drawing. **You are prompted:** Multiple blocks/<Insertion point for block>: A prompt menu displays (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc
9. Follow the instructions in the command bar to insert the block. The Drawing Explorer window reopens.
10. (option) Insert more blocks.
11. Close the Drawing Explorer - Blocks window.

Figure 1.964

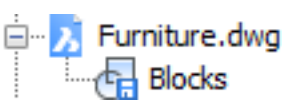


Figure 1.965

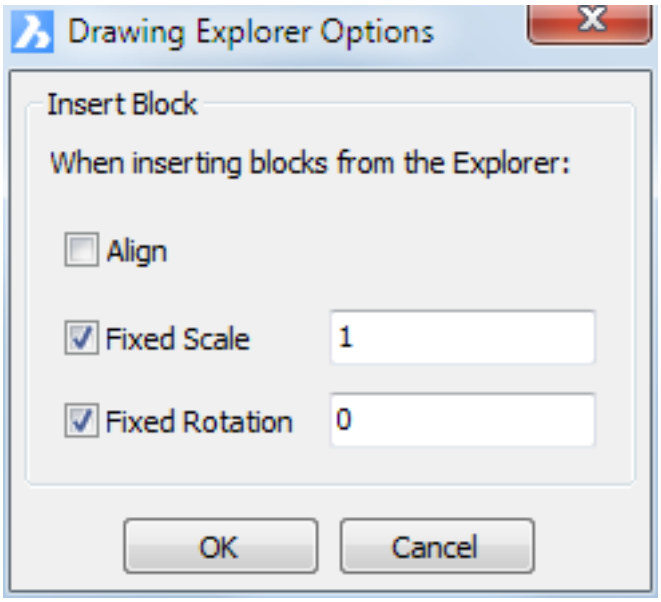
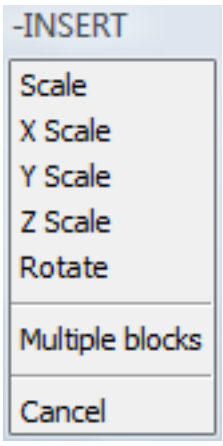


Figure 1.966



To insert a block aligned with an entity

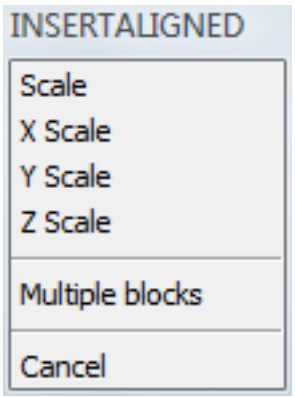
1. Make sure the appropriate Entity Snaps are active.
2. Type **insertaligned** or **insal** in the command bar, then press Enter. You are prompted: ? to list blocks in drawing/~ to open the file dialog/<Block to insert> <name>:
3. Do one of the following:
   - Press Enter or right click to insert the most recently placed block again.
   - Type the name of an existing block definition in the command bar, then press Enter.
   - Type the name of an external block file in the command bar, then press Enter.
   - Type **~** (tilde) then press Enter to open the file dialog.

   You are prompted: Multiple blocks/<Insertion point for block>: A prompt menu displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalReferences/ART/0102Inser
4. Snap to an entity. The block is aligned with the entity.
5. Click to define the insertion point. When you move the crosshairs, the block is mirrored about the selected entity or a about aline tangent to the selected entity.
6. Click to specify the mirroring of the block.

NOTES

- If you type the name of block in step 3, BricsCAD will lookup the block in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the **SRCHPATH** system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'
- The block is aligned as follows: The block is rotated so that its local Z-axis is parallel with the Z-axis of the active UCS, and its local X-axis should be parallel with the tangent to the entity at the position of the snap point. When you click a point on an entity the block is aligned with the entity. Then, by moving the mouse pointer around the insertion point, you can choose how the block should be mirrored. The origin point of the block lies on the entity.

Figure 1.967



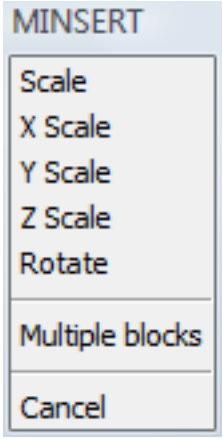
To insert multiple instances of a block in a rectangular array

1. Type **minsert** in the command bar, then press Enter. **You are prompted:** ? to list blocks in drawing/~ to open the file dialog/<Block to insert>:
2. Do one of the following:
   - Type **~** (press and hold the **alt gr** key, then press the **~** key), then press Enter. The **Insert Block** file dialog displays. Select the drawing file you want to insert and press the **Open** button (or double click the file name).
   - Press Enter to accept the <Block to insert>.
   - Type a block name, then press Enter. Optionally, type ? to display search for existing block definitions.
3. The block is attached to the cursor. **You are prompted:** Multiple blocks/<Insertion point for block>: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalReference
4. Specify the insertion point. **You are prompted:** Corner/XYZ/X scale factor <1.00>:
5. To define the X scale factor, do one of the following:
   - Press Enter to accept the default .
   - Type the X scale factor, then press Enter.
   - Click to define the X scale factor graphically.

   You are prompted: Y scale factor: <Equal to X scale (current X scale)>:
6. Repeat step 5 to define the Y scale factor. **You are prompted:** Rotation angle for block <0>:
7. Do one of the following:
   - Press Enter to set the rotation angle to 0°.
   - Type a rotation angle, then press Enter.

   Click to define the rotation angle graphically. **You are prompted:** Number of rows in the array <1>:
8. Type the number of rows, then press Enter or press Enter for one row. **You are prompted:** Number of columns <1>:
9. Type the number of columns, then press Enter or press Enter for one column. **You are prompted:** Vertical distance between rows, or spacing rectangle:
10. Type the row spacing distance, then press Enter. **You are prompted:** Horizontal distance between columns:
11. Type the column spacing distance, then press Enter. The block array is created as a single entity.

Figure 1.968



NOTES

- If you choose **Multiple blocks** in step 3, you are prompted to create multiple instances of the block array.
- It is not possible to explode a **MInsert Block** entity..
- If you type the name of block in step 2, BricsCAD will lookup the block in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the **SRCHPATH** system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'

1.22.7 Redefining Blocks

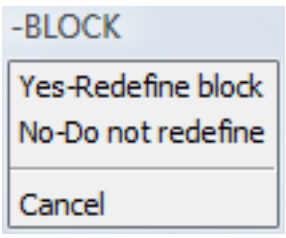
Commands: BLOCK (6.6.11), -BLOCK (6.6.12) and INSERT (6.17.13) To redefine a block that was created in the current drawing, you create a new block using the same name. If the block was inserted from a separate drawing, reinsert that block. When a block definition is redefined all instances of that block in the current drawing are updated automatically. See also: Editing blocks and external references (1.22.19).

To redefine an internal block definition

1. Type **-block** in the command bar, then press Enter. **You are prompted:** Name for new block, or ? to list existing blocks:
2. Type the name of the block in the command bar and press Enter. **You are prompted:** The block <name> already exists. Do you want to redefine it? <N>. A prompt menu displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
3. To redefine the block definition, do one of the following:
   - Type **Y** , then press Enter.
   - Choose **Yes-Redefine block** in the prompt menu.

   You are prompted: Insertion point for new block:
4. Specify the insertion point of the new block. You are prompted: Select entities for block.
5. Select the entities (1.23.6) for the new block.
6. Right click to create the block. The selected entities are removed from the drawing. All existing instances of the block are updated.
7. (option) To bring back the selected block entities, type **oops** in the command bar, then press Enter.

Figure 1.969



To reload an external drawing as a block

1. Do one of the following:

   - Click the **Insert Block...** tool button ( ) on the **Draw** toolbar.
   - Choose **Insert Block...** in the **Draw** menu.
   - Type **insert** in the command bar, then press Enter.

   The **Insert Block** dialog opens.

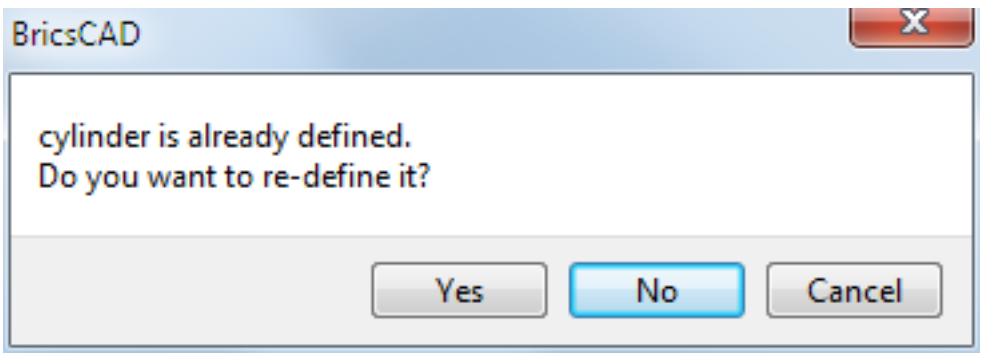
2. On the **Insert Block** dialog click the **Browse** button, then select the external drawing file.

3. Click the **Insert** button. An alert box displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us

4. Click the **Yes** button to redefine the existing block definition. You are prompted: Insertion point for block:

5. Continue with step 3 of the Inserting Blocks (1.22.6)procedure.

Figure 1.970



1.22.8 Exploring Blocks

Command: EXPBLOCKS (6.13.11) The **Explblocks** command opens the **Blocks** section of the Drawing Explorer dialog box (short for "explorer blocks"). In the **Blocks Explorer** you can:

- **Create** new blocks ( )
- **Delete** blocks ( )
- **Purge** unused block definitions ( )
- **Insert** blocks ( ) in the current drawing
- **Insert** external block files ( )
- **Save** a block as a (new) drawing ( )
- **Cut** ( ) or **Copy** ( ) a block definition, then **Paste** ( ) the block definition in another drawing
- **Rename** a block.

Open the Blocks Explorer

To open the **Blocks Explorer** do one of the following:

- Choose **Drawing Explorer > Blocks...** in the Tools menu.
- Type **expblocks** in the command bar.

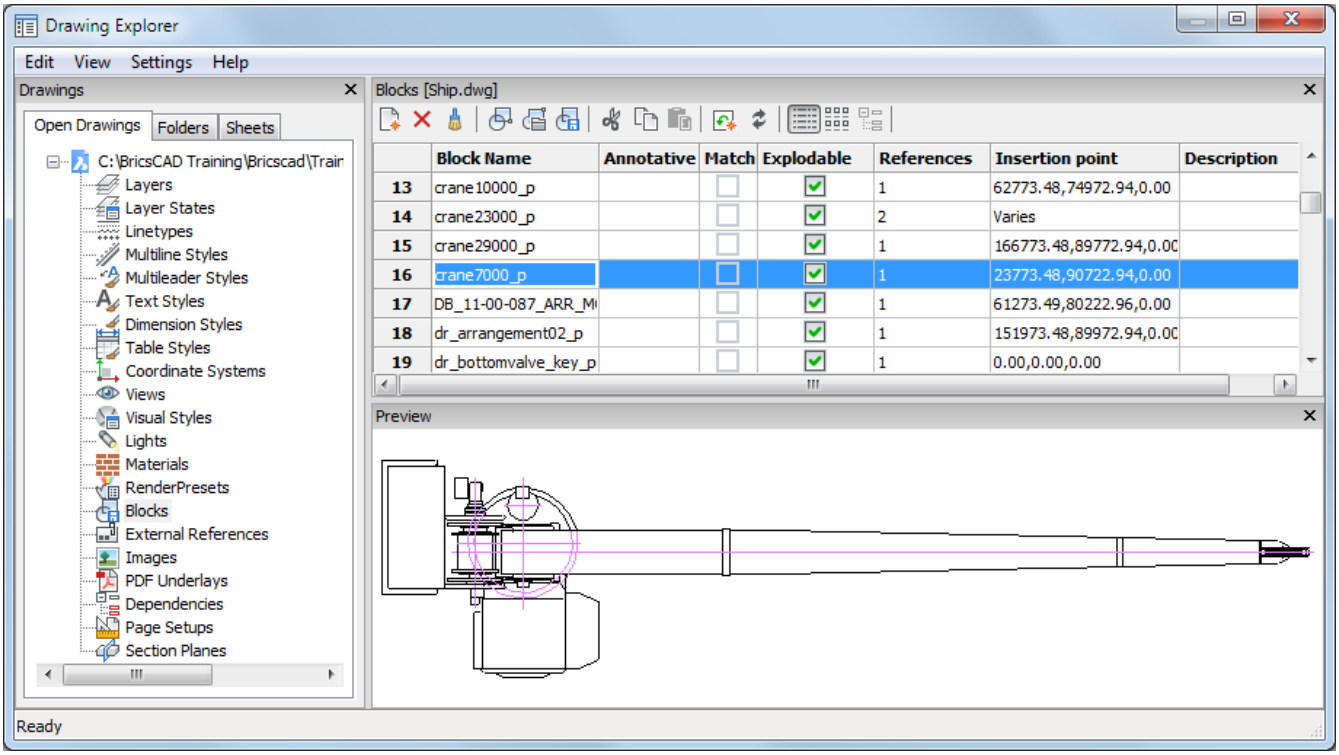
The Blocks Explorer display options

You can choose between **Detail View** (list) and **Icon View** (thumbnails) to see the blocks in the current drawing.

Open the Detail View of the Blocks Explorer

Click the **Detail View** button ( ) in the **Drawing Explorer** toolbar. The **Detail View** button is now pressed indicating the block details are displayed. A preview of the selected block displays in the **Preview** sub-window of the **Drawing Explorer**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternal
Detail View of the blocks in the current drawing.

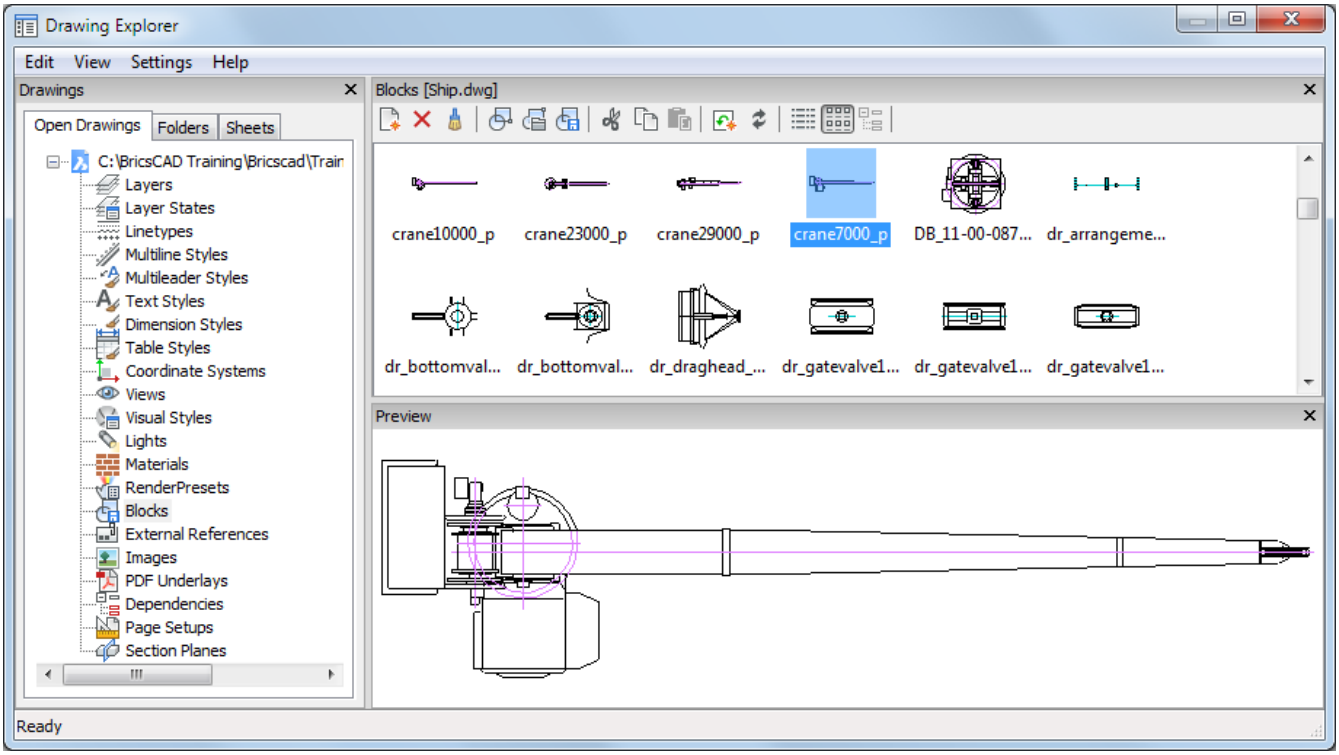
Figure 1.971



Open the Icon View of the Blocks Explorer

Click the **Icon View** button ( ) in the **Drawing Explorer** toolbar. The **Icon View** button is now pressed indicating the block icons are displayed. A preview of the selected block displays in the **Preview** sub-window of the **Drawing Explorer**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalRe Icon View of the blocks in the current drawing.

Figure 1.972



To rename a block

1. Open the Blocks Explorer (1.22.8) dialog.
2. Select the block, then click the **Block Name** field.
3. Type a new name in the **Block Name** field and press Enter.

1.22.9 Exploding Blocks

Command: EXPLODE (6.13.13) If the **Explodable** property of a block is set, you can explode an inserted block to its original component entities. Exploding a block affects that single instance of the block only. The original block definition remains in the drawing and you can still insert additional copies of the original block. If you explode a block that contains attributes, the attributes are lost, but the original attribute definitions remain.

To explode a block

1. Do one of the following:

   - Click the **Explode** tool button ( ) on the Modify toolbar.
   - Choose **Explode** in the Modify menu.
   - Type **explode** in the command bar, then press Enter.

   You are prompted: Select entities to explode.

2. Select (1.23.6) the block(s).

3. Press Enter or right click to explode the blocks.

Note: Exploding dissociates component entities to their next simplest level of complexity: nested blocks or polylines in a block become blocks or polylines again.

Setting the Explodable property of a block

1. Do one of the following:

   - Choose **Drawing Explorer > Blocks...** in the Tools menu.
   - Type **expblocks** in the command bar, then press Enter

   The Drawing Explorer - Blocks dialog displays.

2. If necessary, click the **Detail View** tool button ( ) in the Drawing Explorer - Blocks toolbar.

3. Click the checkbox in the **Explodable** field of a block to toggle the property.

1.22.10 Dynamic Blocks

BricsCAD partly supports dynamic blocks, you can:

- insert dynamic blocks from your existing dynamic blocks libraries
- edit the custom properties in the Properties Bar (1.12.4)
- grip edit dynamic blocks

The creation of new dynamic blocks is not possible yet.Instead of offering dynamic block functionality,BricsCAD focuses entirely on parametrics. (1.18.6) Youtube

Editing dynamic blocks

1. Select the dynamic block in the drawing. The properties of the selected block display in the Properties Bar. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalRefe

2. Under **Attributes > Custom** : edit the properties needed. The dynamic block is updated automatically. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalReference

Figure 1.973

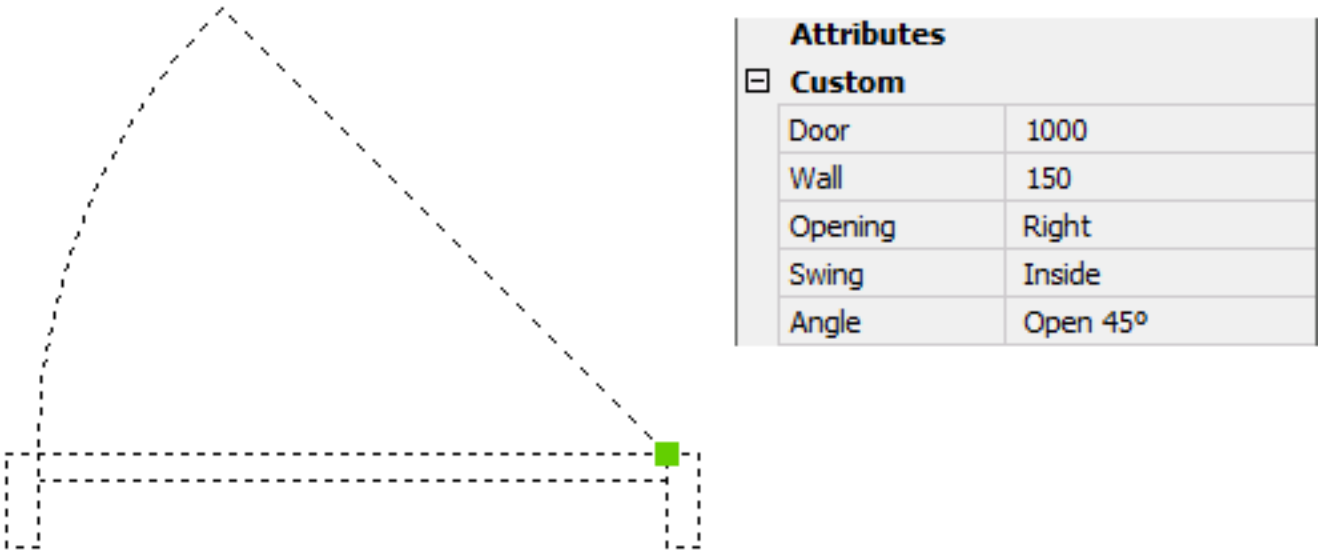
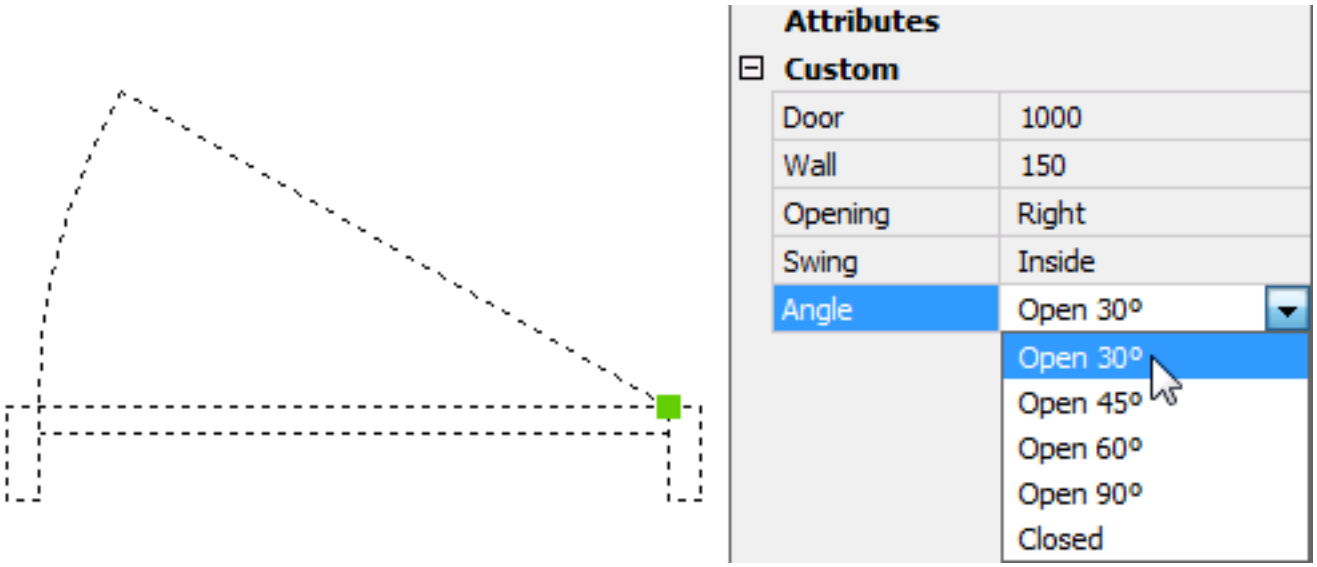


Figure 1.974

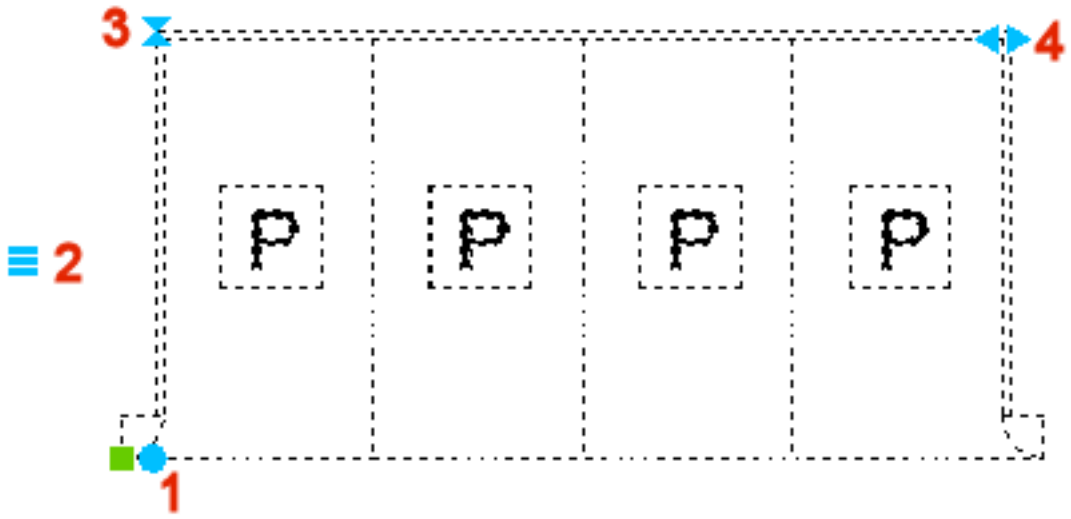


Grip editing dynamic blocks

1. Select the dynamic block in the drawing. The following grip types display:
   - Point
   - XY
   - Linear
   - Polar
   - Rotation
   - Flip
   - Visibility
2. Drag a grip to edit the block. In the example below:
   - Rotate the block (1): drag the grip to rotate the block dynamically or type a value in the dynamic entry field.
   - Show/hide entities (2): click to select an option.
   - Flip (3): click to mirror the block
   - Edit the length (4): select the grip to dynamically edit the length or type a value in the dynamic entry field.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalReference

Figure 1.975



1.22.11 Defining Attributes

Command: ATTDEF (6.4.27) and -ATTDEF (6.4.34) The **Attdef** command defines how a block's attributes will be created through a dialog box. The **-Attdef** command creates block's attribute definitions at the command prompt.

General procedure to define attributes

- Specify the characteristics of the attribute, including its name, prompt, and default value;
- Specify the location;
- Define the text formatting;
- Optionally: set the attribute flags (hidden, fixed, validate or predefined).

To define an attribute

1. Do one of the following:

   - Click the **Define Attributes...** tool button ( ) on the **Tools** toolbar.
   - Choose **Define Attributes...** in the **Tools** menu.
   - Type **attdef** in the command bar, then press Enter.

   The **Define Attribute** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sour

2. Define the Attribute:

   - Type a name in the **Tag** field. The **Tag** Identifies each occurrence of an attribute in the drawing. The name you type in the **Tag** field can be any combination of characters except spaces. Lowercase letters are automatically converted to uppercase.
   - Type a prompt in the **Prompt** field. The attribute prompt displays when you insert a block containing the attribute. If a prompt is omitted, the **Tag** is used as the prompt instead.
   - Do one of the following:
     - Type a default value in theDefault textfield. For variable attributes, the default value is replaced by the actual value when you insert a block containing the attribute.
     - Click theInsert Fieldbutton ( ) to assign afield valueto the attribute (seeUsing sheet set and sheet properties in a title block).
     - If theMultiple linesflag is checked, theDefault textfield is dimmed.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalRefe
   Click the button next to the **Default text** field. The **Define Attribute** dialog temporarily closes to let you **create** a Mtext (1.21.20) entity.

   - Do one of the following:
     - Specify the x-, y-, and z-coordinates for the attribute insertion point.
     - Click thePick Pointbutton ( ) to specify the insertion point in the drawing. You are prompted: Select insertion point. TheDefine Attributedialog closes temporarily.
   - (option) Set the **Invisible** attribute flag. Attributes with the **Invisible** flag set do not display nor print. If the **ATTMODE** (Attribute Display Mode) variable is set to **2** , all attributes display, including **Hidden** attributes.
   - (option) Set the **Constant** attribute flag. Attributes with the **Constant** flag set cannot be edited.
   - (option) Set the **Verify** attribute flag. Attributes with the **Verify** flag set must be explicitly validated when a block containing such attributes is inserted.
   - (option) Set the **Preset** attribute flag. When inserting a block you are not prompted to define attributes of which the **Preset** flag is set. You can edit attribute afterwards though.
   - (option) Set the **Lock Position** attribute flag to lock the position of the attribute within the block reference. Unlocked attributes can be moved relative to the rest of the block using grip editing and multiline attributes can be resized.
   - Define the **Text** properties: **Text Style** , **Justification** , **Annotative** , **Text height** and **Rotation** .

3. Click the **OK** button to create the attribute in the drawing. The **Define Attribute** dialog closes.

4. (option) Right click to reopen the **Define Attribute** dialog, then repeat steps 2 and 3 to create another attribute.

Figure 1.976

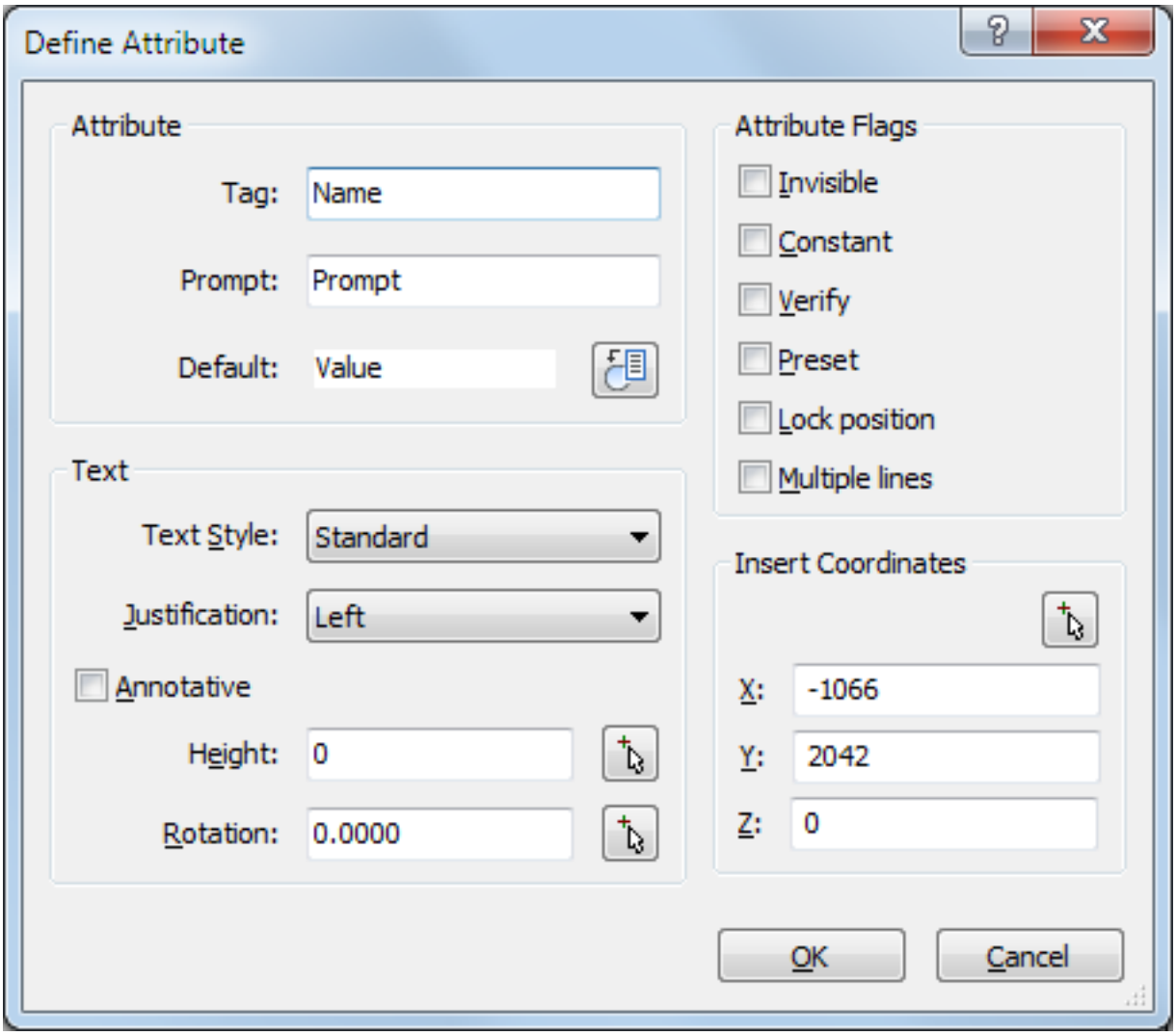
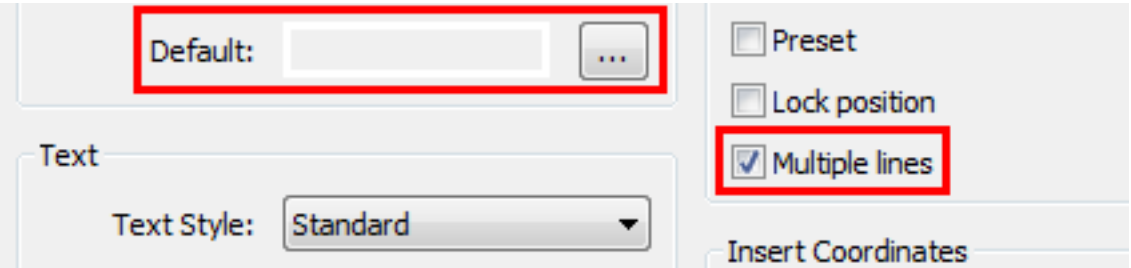


Figure 1.977



Using sheet set properties in a title block

1. Create the sheet set.
2. Define the custom sheet set and sheet properties.
3. Open a sheet (layout) from the sheet set.
4. Create the title block source entities in the paper space layout:
   - Create the borders, line work and fixed texts.
   - Create attributes with fields (1.21.23) that refer to sheet set, subset and sheet properties (see To define an attribute (1.22.11)).

   See the SHEETSET (6.27.31) command to learn more about default and custom sheet set, subset and sheet properties. Custom sheet set properties: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGu
   Sheet properties and sheet custom properties: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/U
5. Create a block (1.22.5) from the title block source entities.
6. Copy the block to drawing in one of your block folders (see To insert a block form another drawing (1.22.6)).
7. Delete the title block source entities.
8. The title block can be inserted in the other sheets of the sheet set and in any other sheet set where the same custom sheet set and sheet properties exist. All fields that refer to default or custom sheet set, subset or sheet properties are filled out automatically. If no value is available, a field displays hyphens (----) (see also Placing blocks with attributes (1.22.14)).

Figure 1.978

Sheet Set Custom Properties	
Architect	Bricsys & Partners
ArchitectAddress	Bellevue 5/201
ArchitectCity	9050 Gent
ArchitectPhone	09 244 01 91
Contractor	Robert Stone ltd.
ContractorAddress	123 Main Street
ContractorCity	New Town
Owner	Carpenter
OwnerAddress	1 Church Road
OwnerCity	New Town
OwnerTitle	Mr. & Mrs.
SiteAddress	48 Brick Lane
SiteCity	New Town
SiteReference	Section A no. 123

Figure 1.979

Sheet	
Number	08
Title	Elevation South
Description	South elevation drawing
Publish	☑ Include for Plot/Publish
Layout	LAYOUT1 (C:\BricsCAD Trainir
Revision Number	1
Revision Date	15/08/2013
Issue Purpose	Final Design
Category	Elevation
Sheet Custom Properties	
Date	12/08/2013
PaperSize	A2
Printer	pdf
Remark	none
Scale	1/50

1.22.12 Editing attribute definitions

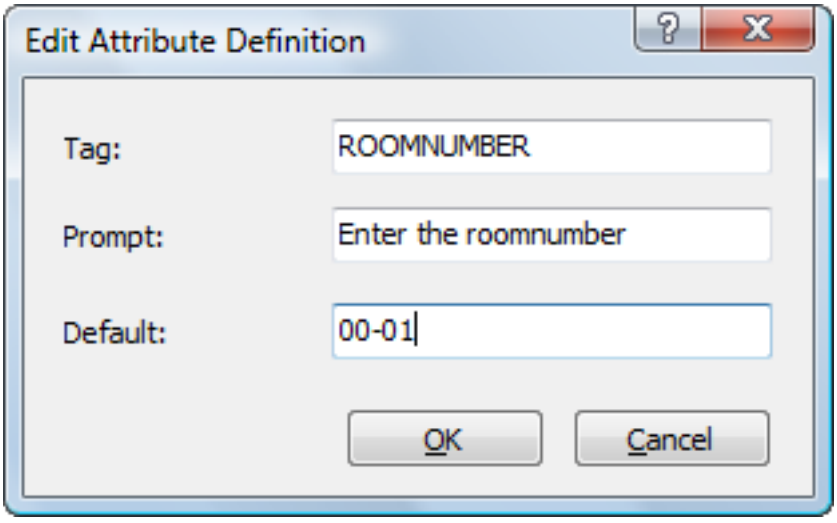
Command: DDEDIT (6.10.14) The **Ddedit** command edits single-line text, multi-line text, attribute definitions, and attribute text (short for "dynamic dialog editor"). You can edit an attribute definition only before it is saved as part of a block definition.

To edit tag, prompt or default text of an attribute

1. Do one of the following:
   - Type **ddedit** in the command bar, then press Enter and select the attribute.
   - Double click the attribute.

   The **Edit attribute definition** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG
2. (option) Type a new name in the **Tag** field. Lowercase letters are converted to uppercase automatically.
3. (option) Type a new prompt in the **Prompt** field. If you leave the **Prompt** field empty, the attribute **Tag** will be used as the prompt instead.
4. (option) Type a new text in the **Default** field.
5. Press the **OK** button to save your changes.

Figure 1.980



To edit an attribute definition

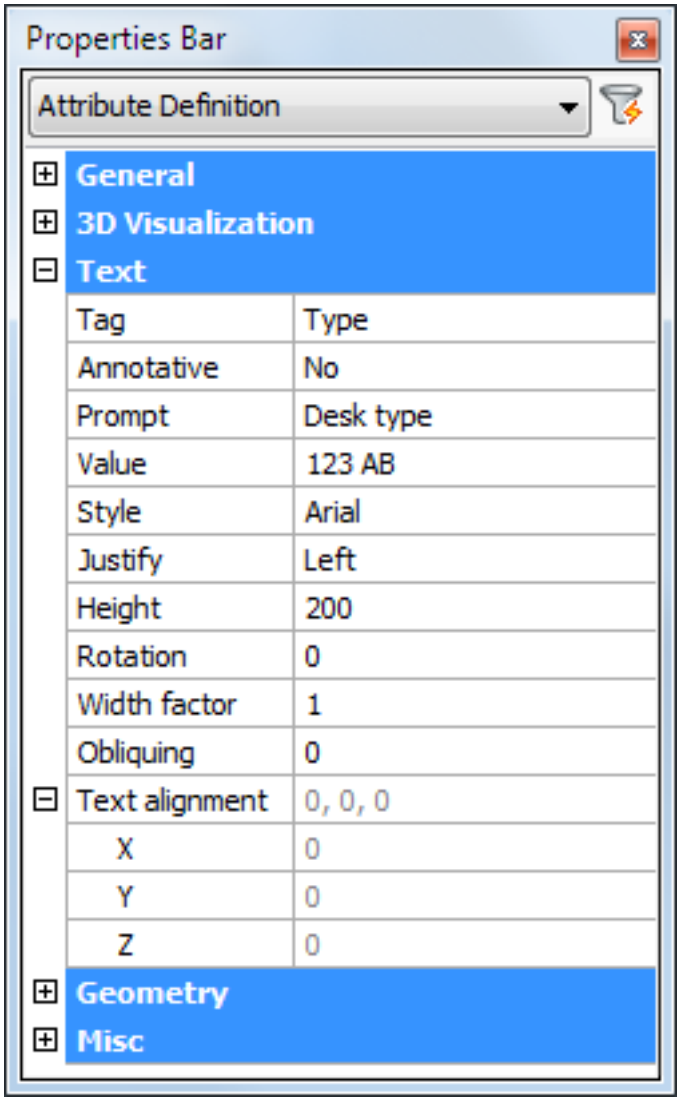
1. Click the attribute. The attribute properties display in the Properties Bar (1.12.4). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalReferences/ART/0202EditA

2. Modify the attribute properties in the **Properties Bar** fields.

3. Press the Esc key to stop editing.

Note: Make sure the Properties Bar (1.12.4) is open before starting this procedure.

Figure 1.981


Properties Bar
Attribute Definition
General
3D Visualization
Text
Tag
Type
Annotative
No
Prompt
Desk type
Value
123 AB
Style
Arial
Justify
Left
Height
200
Rotation
0
Width factor
1
Obliquing
0
Text alignment
0, 0, 0
X
0
Y
0
Z
0
Geometry
Misc

1.22.13 Editing attributes attached to blocks

Command: EATTEDIT (6.13.1) The **Eattedit** command edits the attributes in blocks. You are prompted to select a block with editable attributes. When you select such block, the Attribute Editor dialog displays. If the selected block does not contain attributes or if the selected entity is not a block you are prompted to select another block.

Using the Attribute Editor

1. Do one of the following:

   - Click the **Edit Block Attributes...** tool button ( ) on the Attributes toolbar.
   - Choose **Edit Block Attributes...** in the Tools | Attributes menu.
   - Type **eattedit** in the command bar, then press Enter.

   You are prompted: Select a block. **Or** double click a block containing attributes, then go to step 3.

2. Select a block with attributes. The Attribute Editor dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/

3. Select an attribute in the list. The current value of the selected attribute displays in the **Value** field.

4. (option) Type a new value in the **Value** field.

5. (option) Click the **Properties** tab to modify **Layer** , **Color** , **Lineweigth** and/or **Linetype** .

6. (option) Click the **Text Options** tab to edit the text properties. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help

7. (option) Click the **Apply** button to apply the changes.

8. (option) Repeat steps 3 through 6 to edit another attribute.

9. Do one of the following:
   - Click the **OK** button to close the Attribute Editor dialog and apply the changes.
   - Click the Apply button to apply the changes and keep the Attribute Editor dialog open.
   - Click the **Cancel** button to close the Attribute Editor dialog. Changes that are not applied yet are not saved.
   - Click the **Select** button ( ) to select another block. The Attribute Editor dialog temporarily closes to let you select another block in the drawing.

Figure 1.982

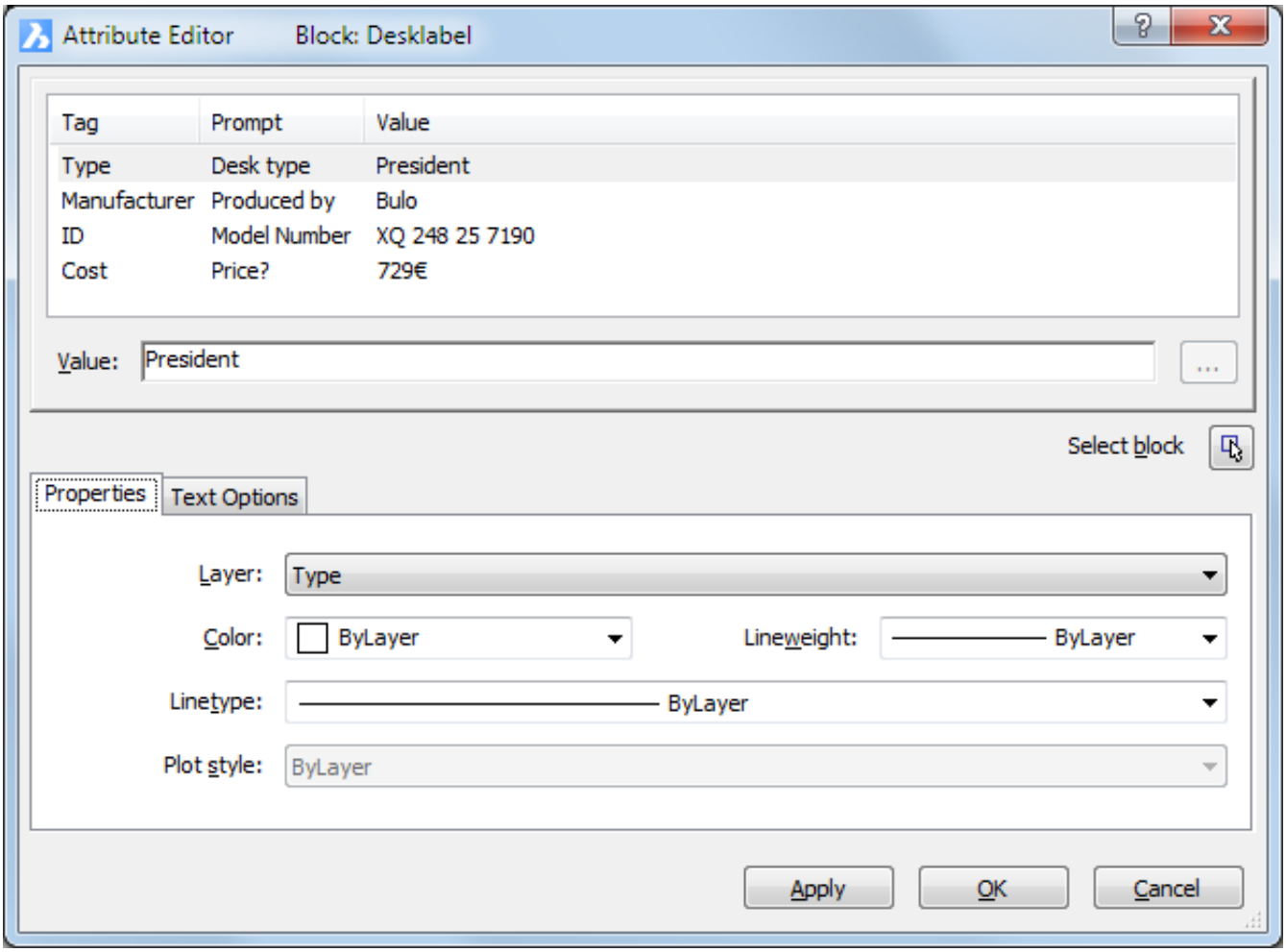
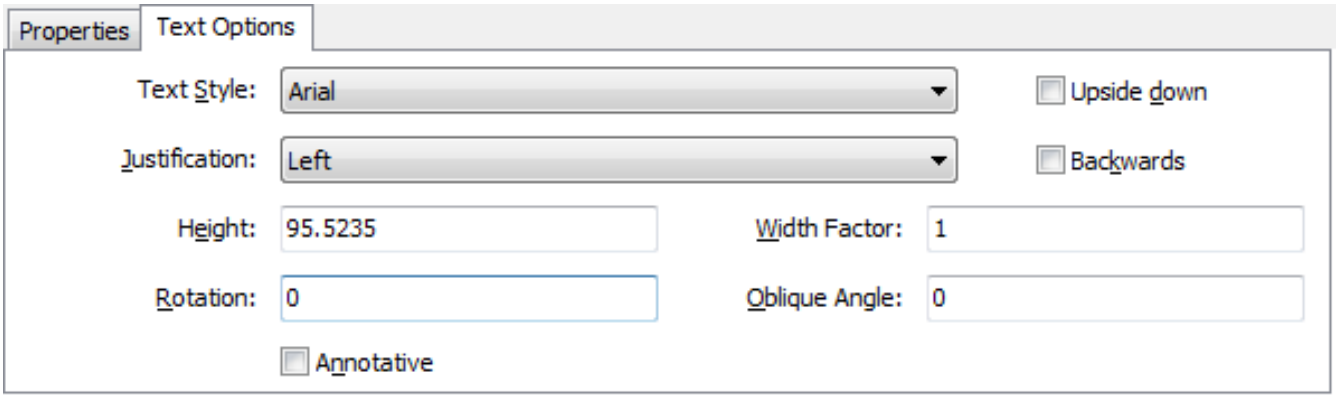


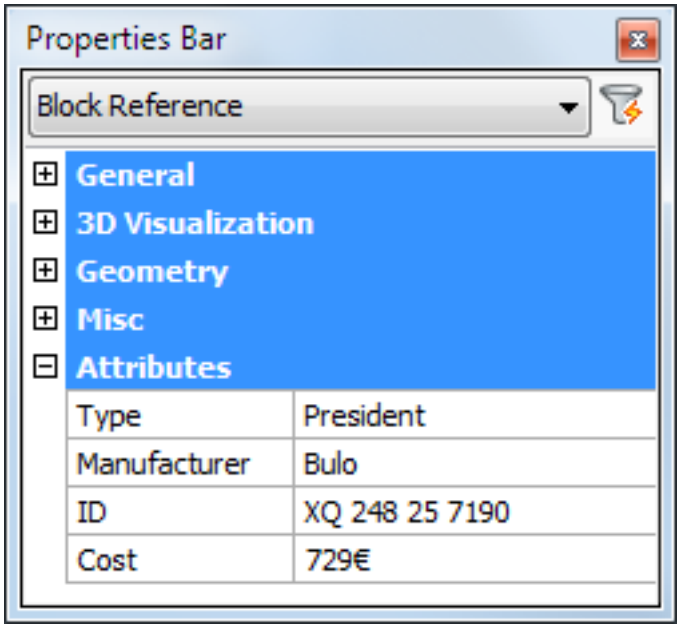
Figure 1.983



Editing attributes in the Properties bar

1. Select a block containing attributes. The current value of the attributes display in the Properties bar (1.12.4).
2. Click the attribute you want to edit. The settings field of the selected attribute is active. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalReferences/ART/0204EditA
3. Type a new value, then press enter or select another attribute.

Figure 1.984


Properties Bar
Block Reference
General
3D Visualization
Geometry
Misc
Attributes
Type
President
Manufacturer
Bulo
ID
XQ 248 25 7190
Cost
729€

1.22.14 Placing blocks with attributes

Commands:INSERT (6.17.13) and-INSERT (6.17.14) When you place a block which contains attributes, you will be prompted to define the attributes in the command bar or through a dialog box. If theATTDIA (1.13.11) system variable is ON, attributes display in a dialog box. If theATTDIA (1.13.11) system variable is OFF, you are prompted in the command bar to define the attributes.

To place a block with attributes

1. Launch theInsert Block... (1.22.6) tool.
2. Position the block in the drawing. All block entities are placed in the drawing, except the attributes. **You are prompted:** <Attribute Prompt> <Default Text>: **Or** a dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
3. Do one of the following:
   - In the command bar: The command bar reads: <Attribute Prompt> <Default Text>:
     - Press Enter or right click to accept the default text.
     - Type a new text in the command bar, then press Enter and define the next attribute.
   - In the**Edit Attributes** dialog box:
     - Select an attribute in the list. The current value displays in the**Value** field.
     - Type a new value in the**Value** field.
     - Press**OK** when all attributes are properly defined.

Figure 1.985

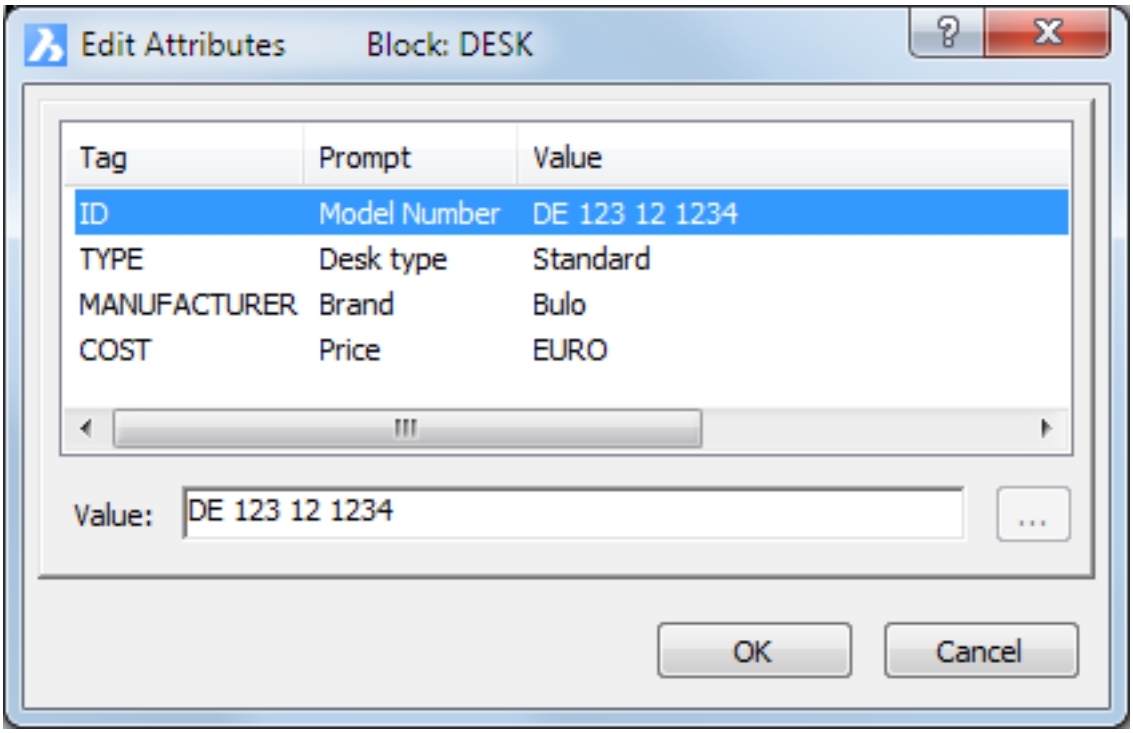
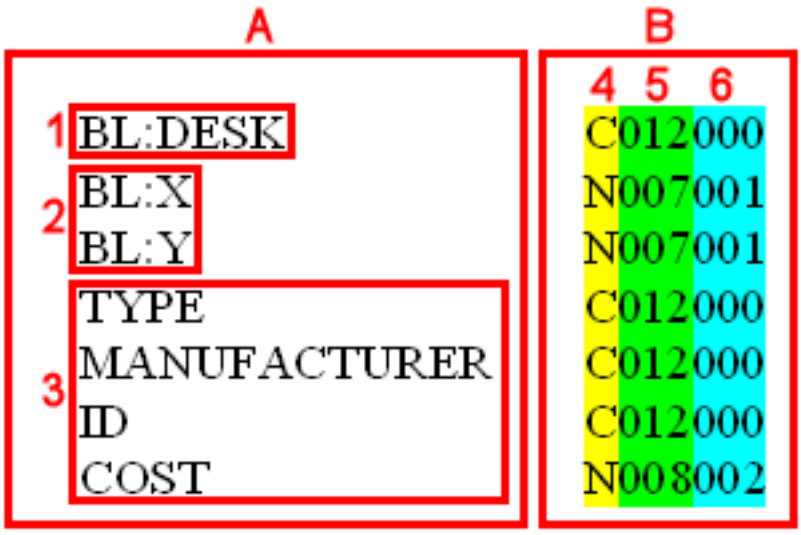

Edit Attributes
Block: DESK
Tag
Prompt
Value
ID
Model Number
DE 123 12 1234
TYPE
Desk type
Standard
MANUFACTURER
Brand
Bulo
COST
Price
EURO
Value:
DE 123 12 1234
...
OK
Cancel

Figure 1.986



1.22.15 Extracting attribute information

Command: ATTEXT (6.4.28) and -ATTEXT (6.4.29), DATAEXTRACTION (6.10.1) The **Attext** command copies data from attributes to a text file via a dialog box. The **-Attext** command copies data from attributes to a text file via the command line. The text file can the be imported in a database or a spreadsheet. You can save the file in any of the following formats:

- **Comma Delimited Format** (CDF): Contains one line for each instance of a block, with individual attribute fields separated by commas. Character string fields are enclosed with single quotation marks. You must specify a template file when extracting to a CDF file.
- **Space Delimited Format** (SDF): Contains one line for each instance of a block. Each attribute field has a fixed length; there are no separators or character string delimiters. You must specify a template file when extracting to a SDF file.
- **Drawing Exchange Format** (DXF): Creates a subset of a standard DXF file (a *.dxx file) containing all the information about each block, including the insertion points, rotation angles, and attribute values. No template file is required.

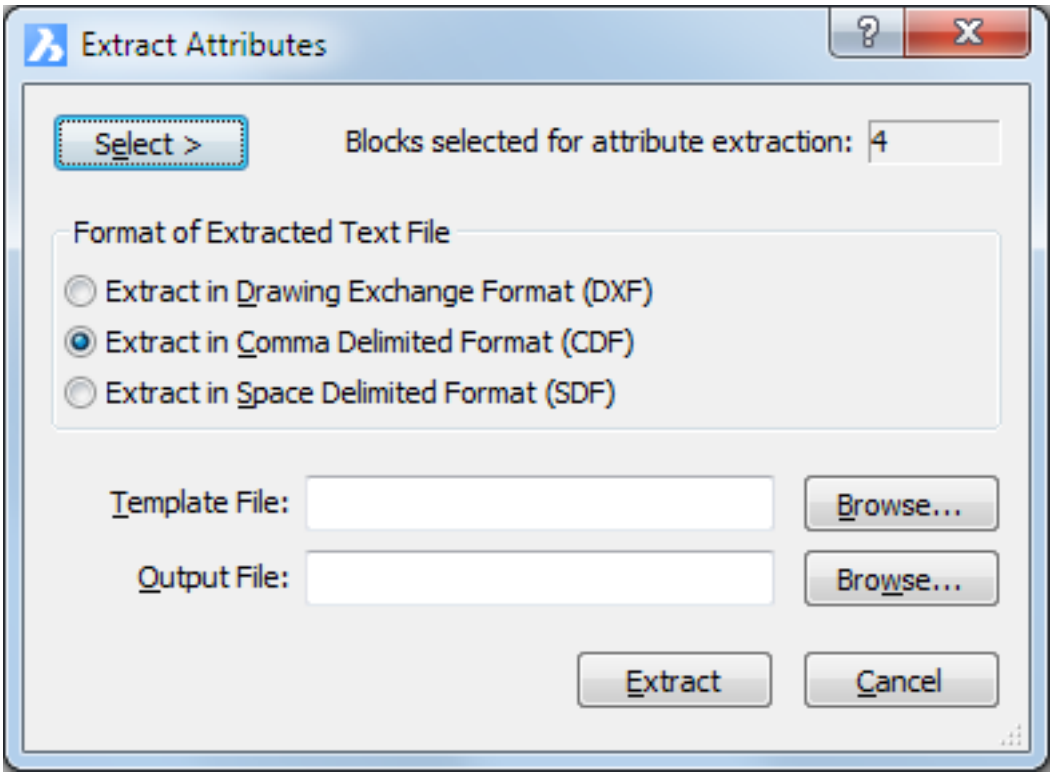
Before extracting attributes to a CDF or SDF file, you must create a template file. The template file is an ASCII text file that specifies the attribute data fields to be written in the extract file. Each line of the template file specifies one attribute field. BricsCAD recognizes 15 different fields, which contain elements such as the block name, the x-, y-, and z-coordinates of its insertion point, the layer on which it is inserted, etc. You can include any of these fields. The template file must include at least one attribute name. Each line in the template file must start with the field name. Block name and insertion-point values must begin with BL:. The next nonblank character must be either a C (indicating a character string field) or an N (indicating a numeric field). This character is then followed by three digits indicating the width of the field (in characters). The final three digits indicate the number of decimal places (for numeric fields). In the case of character fields, the last three digits must be zeros (000). A typical template file is similar to the one shown here: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExtern
A. Field name B. Field format 1. Block name (must be preceded by **BL:**) 2. Coordinate fields (must be preceded by **BL:**) 3. Attribute tags 4. Indicates the field data type: **C** for character fields, **N** for numeric fields 5. Field width (number of characters) 6. Number of decimal places for numeric fields, **000** for character fields

To extract attribute information

1. Do one of the following:
   - Click the **Extract Attributes** tool button on the Tools toolbar.
   - Choose **Extract Attributes** in the tools menu.
   - Type **attext** in the command bar, then press Enter.

   The Extract Attributes dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sou
2. Click the **Select** button. The **Extract Attributes** dialog closes temporarily.

Figure 1.987



3. Select the blocks, then press Enter or right click. The **Extract Attributes** dialog reopens.
4. Select the text file format: **DXF** (drawing exchange format), **CDF** (comma delimited format) **or SDF** (space delimited format).
5. Select the **Template file** .
6. Specify the **Output file** name.
7. Click the **Extract** button. **You are prompted:** <number> records extracted.

1.22.16 Attaching an external reference

Command: XATTACH (6.34.1) The **Xattach** command attaches a drawing as an external reference (xref). The attached xref appears as an insert in the current drawing. Like blocks, attached xrefs can be nested. The attached drawing is based on the most recently saved version. The origin point used when attaching an external reference is controlled by the **INSBASE** (insertion base point) settings variable. Use the BASE (6.6.3) command to define the **INSBASE** variable in the external reference drawing.

To attach an external reference

1. Do one of the following:

   - Click the **Attach Xref...** tool button ( ) on the Insert toolbar.
   - Choose **Attach Xref...** in the **Insert** menu.
   - Type **xattach** in the command bar, then press Enter.
   - Click the **Attach Xref** tool button ( ) on the Details toolbar on the Drawing Explorer / Xrefs (1.14.2) window.

   The **Attach External Reference** dialog window displays.

2. Select the drawing you want to attach, then press the **Open** button on the Open dialog window. The **Attach External Reference** dialog window opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/

3. The full path to the selected drawing displays in the **Path** field. (option) Select the **Path Type**:
   - (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExterna
   - (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExterna
   - (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExterna

4. Choose the **External Reference Type**:
   - **Attachment** : If the current drawing is attached as an xref to another drawing, an attachment becomes a nested xref.
   - **Overlay** : An overlay is ignored when the drawing to which it is attached is then attached as an xref to another drawing.

5. Choose whether to specify the **Insertion Point** on screen or in the X-, Y-, Z-fields.

6. Choose whether to specify the **Scale** on screen or in the X-, Y-, Z-fields. Check the **Uniform Scale** option to specify the scale in the X-field.

7. Choose whether to specify the **Rotation** angle on screen or in the **Angle** field.

8. Click the **OK** button to attach the Xref. Depending on the options in steps 5, 6 and 7 you are prompted to specify the insertion point, scale and/or rotation angle.

Note: The **Unit field** under **Block Unit** is set by the INSUNIT (Insertion Units) system variable (1.22.6) in the external reference. The Factor field expresses the relation between the value of the INSUNIT system variable in the xref and the parent drawing.

Figure 1.988

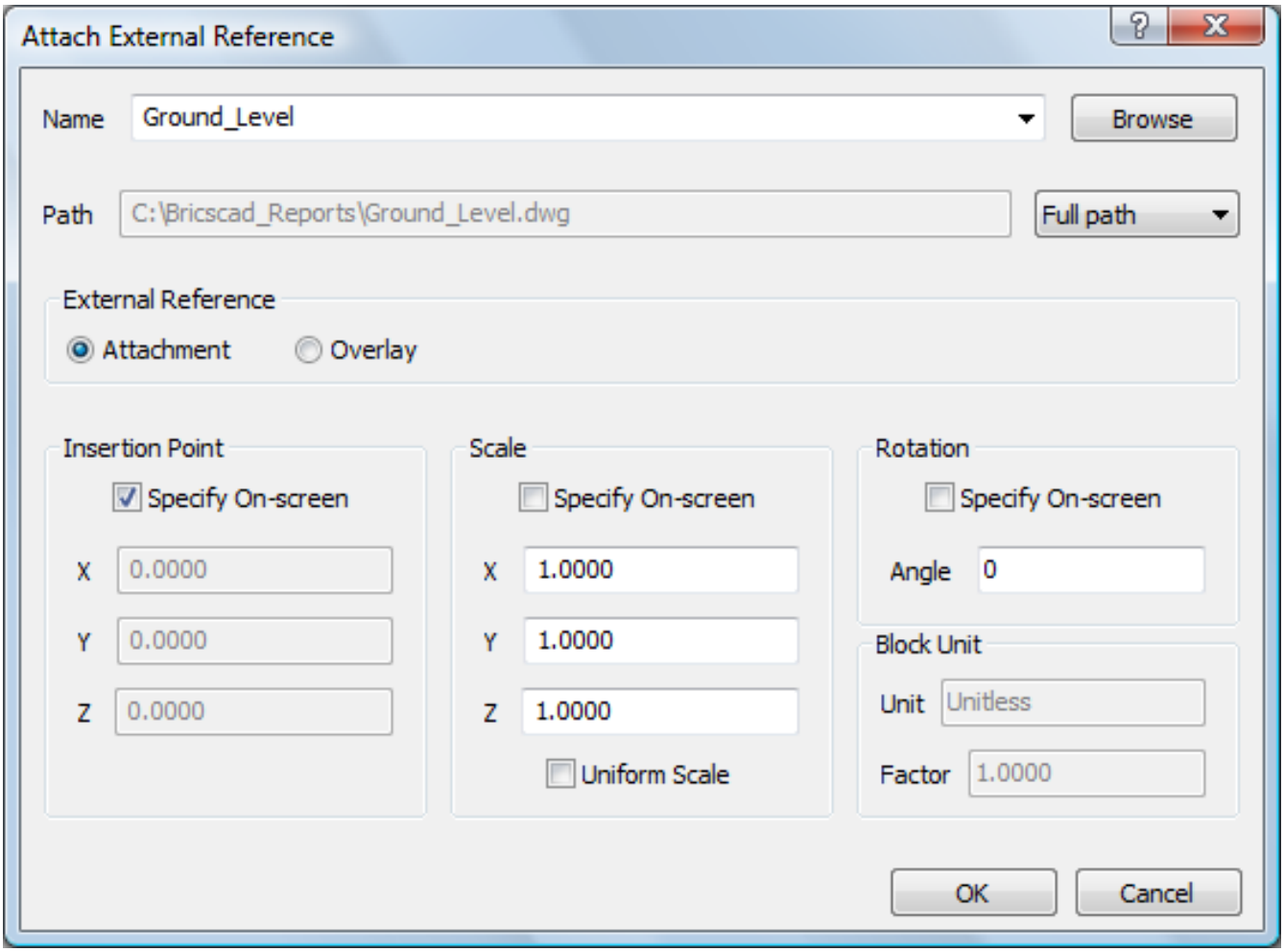


Figure 1.989



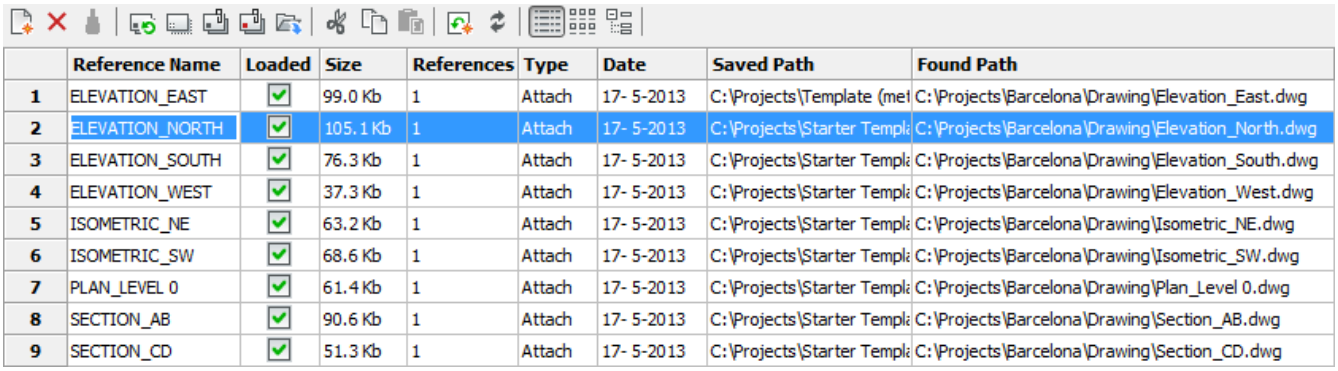
Figure 1.990



Figure 1.991



Figure 1.992



	Reference Name	Loaded	Size	References	Type	Date	Saved Path	Found Path
1	ELEVATION_EAST	☑	99.0 Kb	1	Attach	17- 5-2013	C:\Projects\Template (met	C:\Projects\Barcelona\Drawing\Elevation_East.dwg
2	ELEVATION_NORTH	☑	105.1 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Elevation_North.dwg
3	ELEVATION_SOUTH	☑	76.3 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Elevation_South.dwg
4	ELEVATION_WEST	☑	37.3 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Elevation_West.dwg
5	ISOMETRIC_NE	☑	63.2 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Isometric_NE.dwg
6	ISOMETRIC_SW	☑	68.6 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Isometric_SW.dwg
7	PLAN_LEVEL 0	☑	61.4 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Plan_Level 0.dwg
8	SECTION_AB	☑	90.6 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Section_AB.dwg
9	SECTION_CD	☑	51.3 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Section_CD.dwg

1.22.17 Managing Xrefs

Command:XREF (6.34.8) TheXref command opens theExternal Referencessection of theDrawing Explorerdialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalRefer TheDrawing Explorer - External Referencesdialog allows to:

- Attach (1.22.16)a drawing as an external reference ( ).
- Detach an external reference ( ).
- Reload an external reference ( ).
- Unload an external reference ( ).
- Bind an external reference ( ).
- Insert an external reference ( ).
- Modify the attachment type:AttachorOverlay.
- Modify theSaved Pathof an xref.
- Modify theFound Pathof an overlay.

To detach external references

1. Select the xref(s) you want to detach. Press and hold the Ctrl key to select multiple xrefs or to unselect a selected xref.
2. Click theDetach Xreftool button ( ) on theDetailstoolbar. The selected xref(s) are detached. You are not prompted to confirm the detachment.

To unload external references

Do one of the following:

- Click theLoadedcolumn of a loaded Xref.
- Select the Xref(s), then either click theUnload Xreftool button ( ) on theDetailstoolbar or click theLoaded-column of one of the selected Xrefs.

To reload external references

Do one of the following:

- Click theLoadedcolumn of an unloaded Xref.
- Select the Xref(s), then either click theReload Xreftool button ( ) on theDetailstoolbar or click theLoaded-column of one of the selected Xrefs.

1.22.18 Working with References

Attach Reference Files

BricsCAD allows you to attach externally referenced drawings, images, and pdf files to the current drawing. Easily access the attach tools from the Insert tab on the Quad. You can also access them via other methods including the Command line (XATTACH, IMAGEATTACH, PDFATTACH). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
Attributes-ExternalReferences/HHBricsCAD-Working-with-ExternalFiles-01.png}) You attach reference files in BricsCAD the same way you do in AutoCAD®. First select a valid file type: DWG, Raster image, or PDF. Then, specify attach options in the relevant Attach dialog box.Supported raster images include: BMP, JPG, JPEG, PCX, PNG, GIF, TGA, TIF, TIFF, JP2, J2K, ECW, and SID. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Blocks-Attributes-ExternalReferences/HHBricsCAD-Working-with-ExternalFiles-02.png}) Note: The dialog boxes do not include preview images. However, when attaching a PDF file, you can select which page to attach from the page list.

Figure 1.993

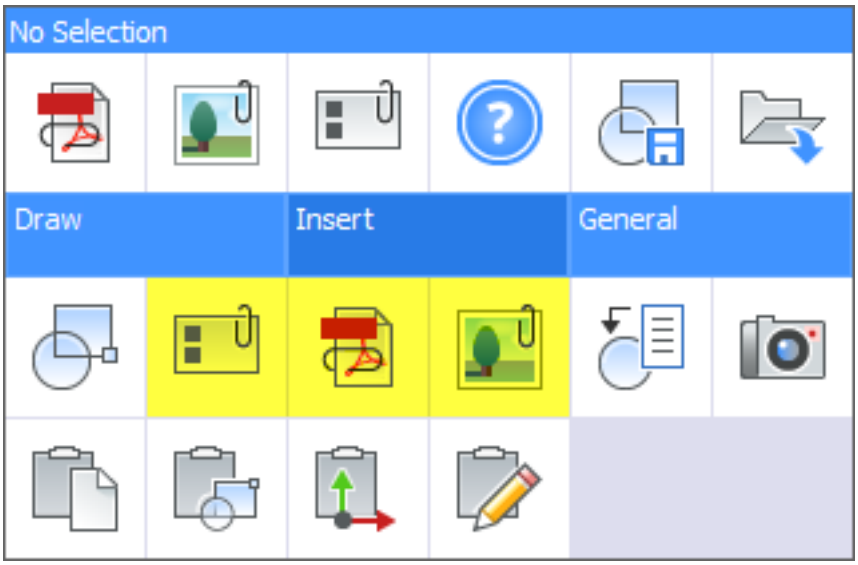
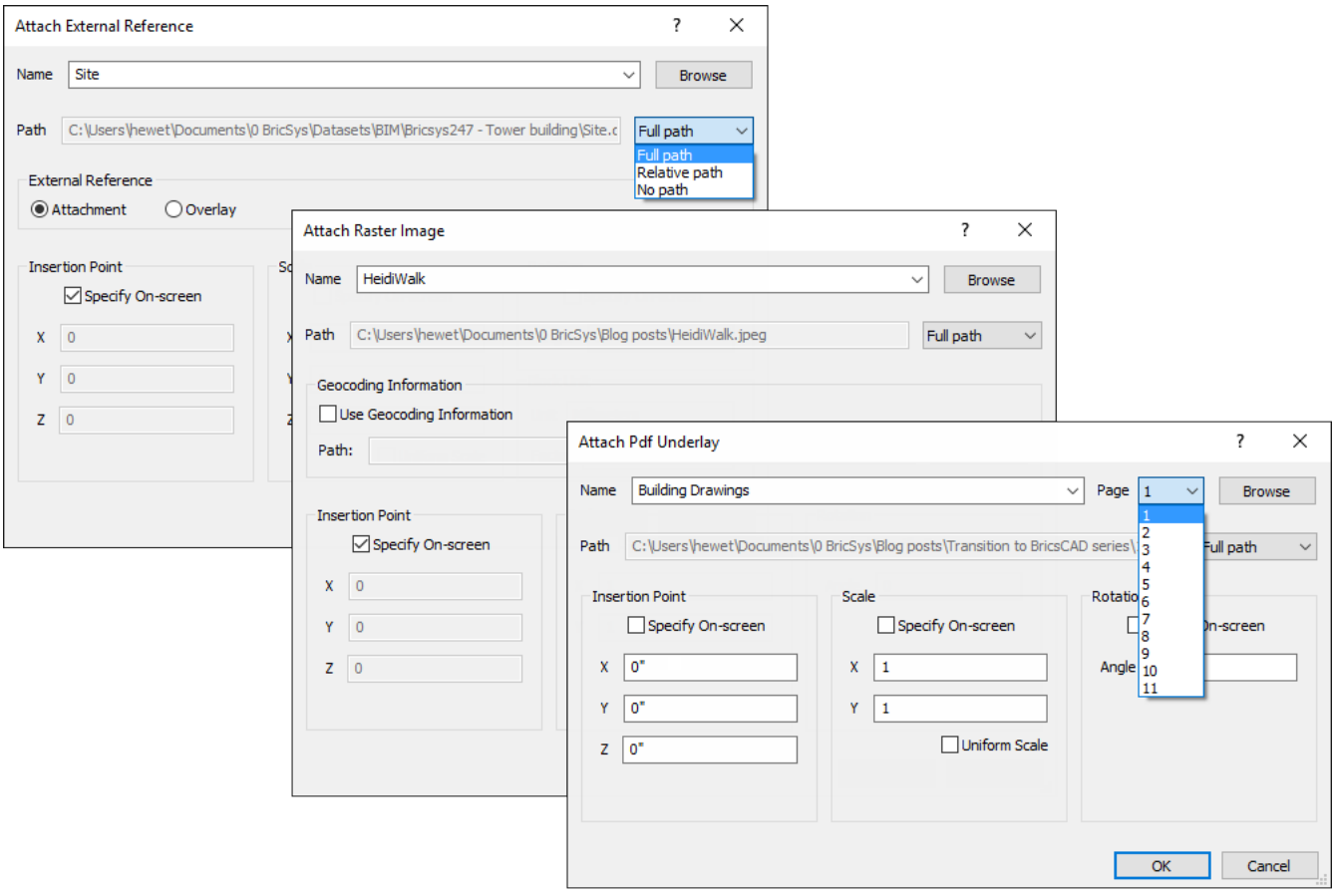


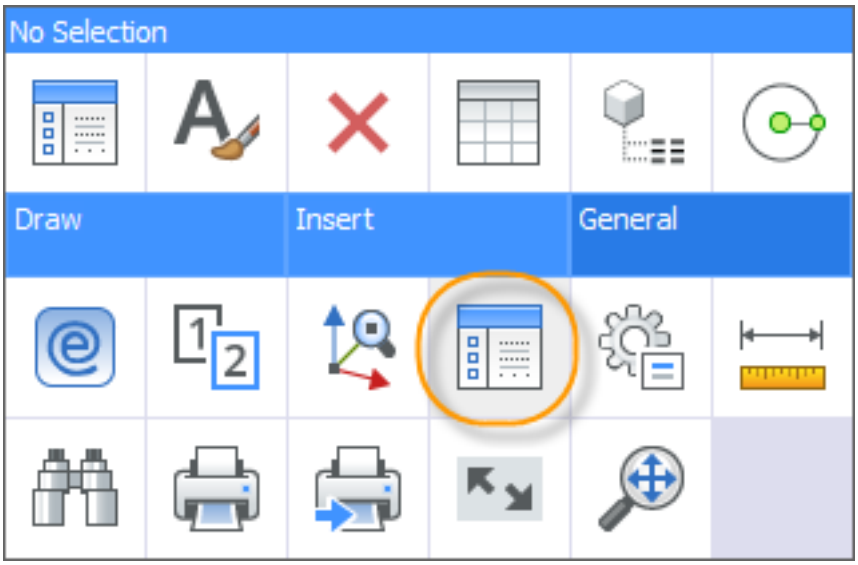
Figure 1.994



Manage Reference Files in Drawing Explorer

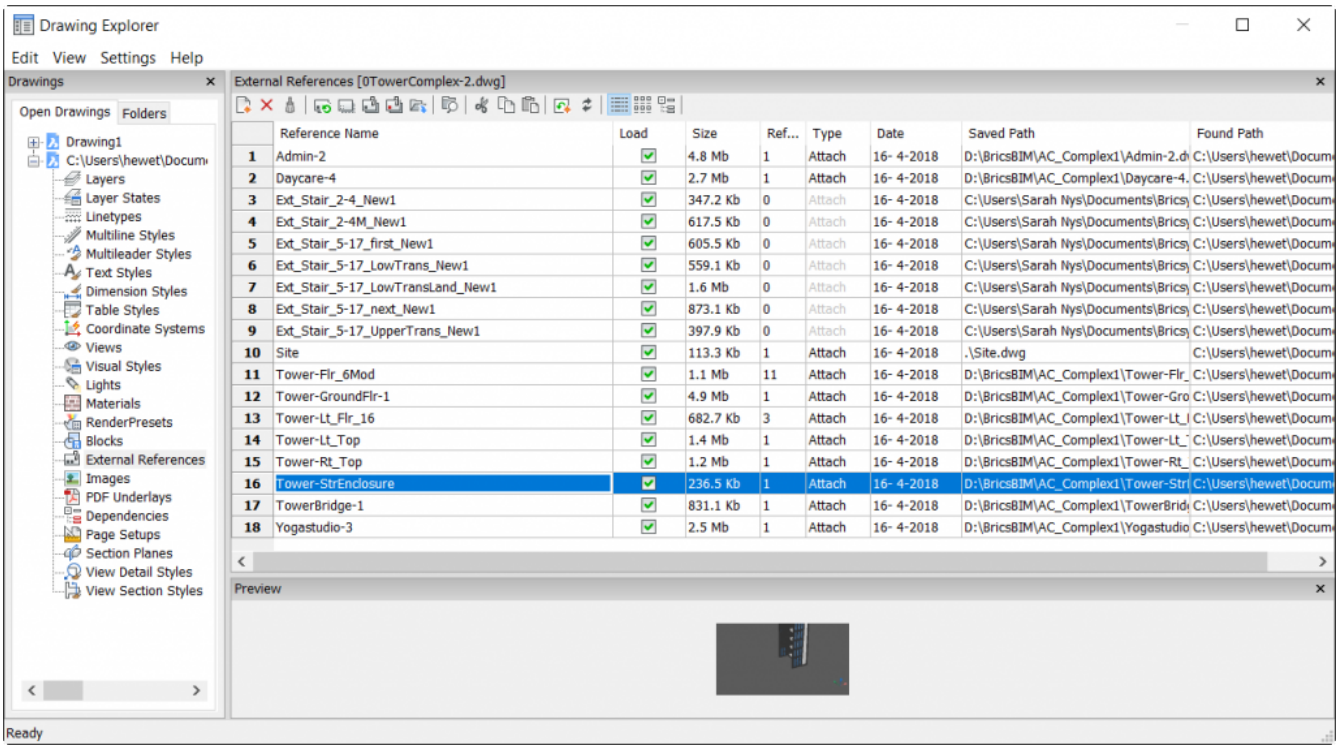
After you attach an external file to the current drawing, BricsCAD creates a reference link to the file. You can view and manage those links like in AutoCAD. However, instead of using an External References palette, you use Drawing Explorer. Access Drawing Explorer from the General menu on the Quad. It's also accessible via other methods including the EXPLORER command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Blocks-Attributes-ExternalReferences/HHBricsCAD-Drawing-Style09.png}) Or, you can use the familiar XREF, IMAGE, or PDF commands. Doing so automatically opens Drawing Explorer with relevant panels displayed for that reference type.

Figure 1.995



External References The External References panels include detailed information such as the referenced DWG file name and size. Additionally, it displays the number of insertions for each DWG reference. Nested references are clearly indicated by the greyed-out attachment type. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BAttributes-ExternalReferences/HHBricsCAD-Working-with-ExternalFiles-03-1024x568.png}) To help you visualize the individual references and relationships between them, you can change to icon or tree view. The right-click menu and toolbar offer additional controls. You can easily attach new xrefs, detach existing ones, or purge those that are unused. And you can reload, unload, bind, insert, and open attached references. Additional items on the right-click menu make it easy to switch the saved path between relative, absolute, or just the file name. And, on the toolbar, the “Run extended search for missing attachments” tool enables you to easily fix broken reference paths either individually or for multiple references simultaneously.

Figure 1.996



Images The Images panels display detailed information about the images that are linked to the drawing. The menu and toolbar offer tools similar to Xrefs but specifically relevant for images. (\vref {fig:C:/Dev/Task-2448/images/staticbri Attributes-ExternalReferences/HHBricsCAD-Working-with-ExternalFiles-06-1024x568.png}) When you add images in BricsCAD, you can select and attach multiple images at the same time, unlike in AutoCAD. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Blocks-Attributes-ExternalReferences/HHBricsCAD-Working-with-ExternalFiles-07-1024x534.png})

Figure 1.997

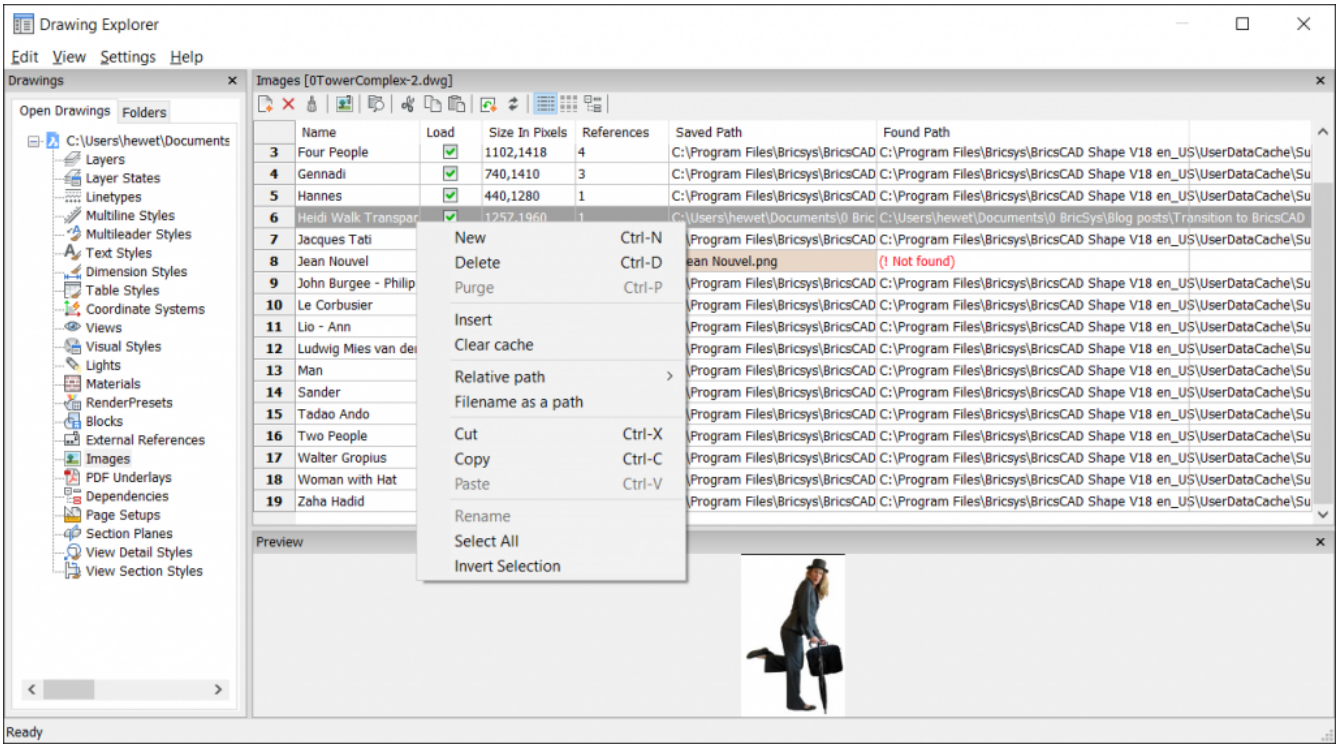
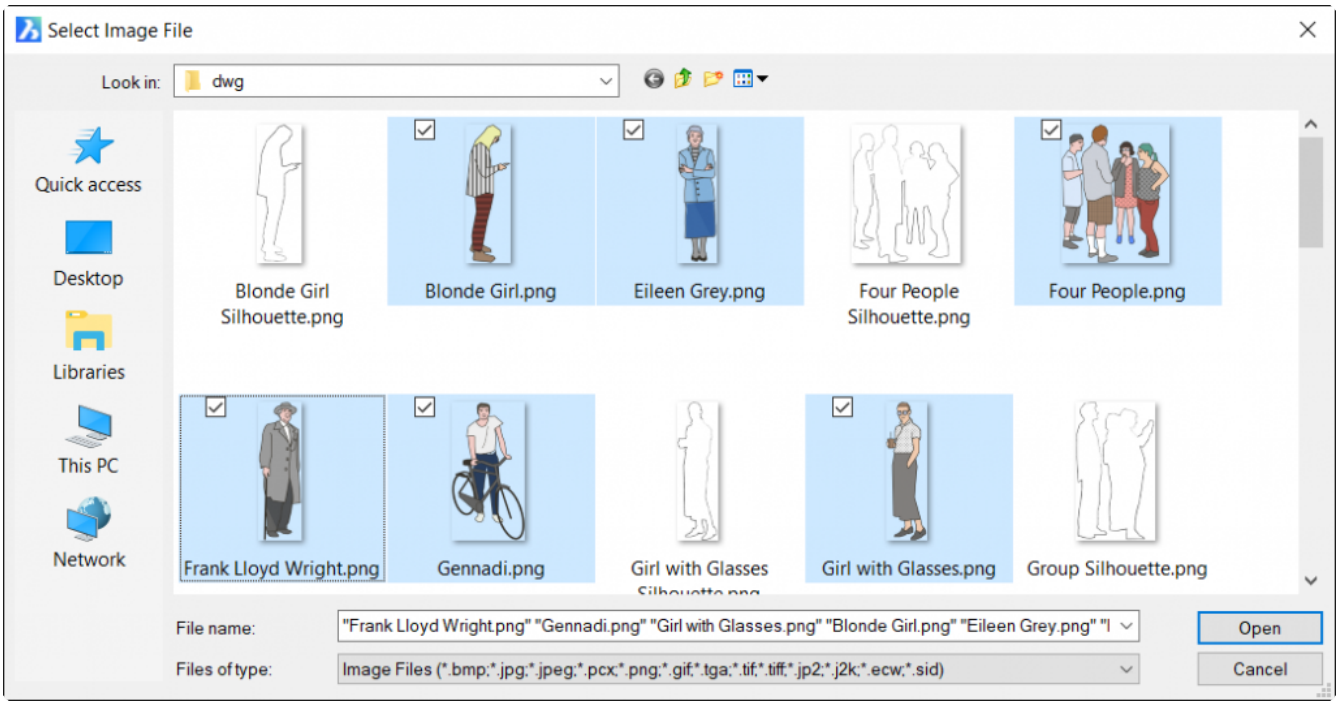
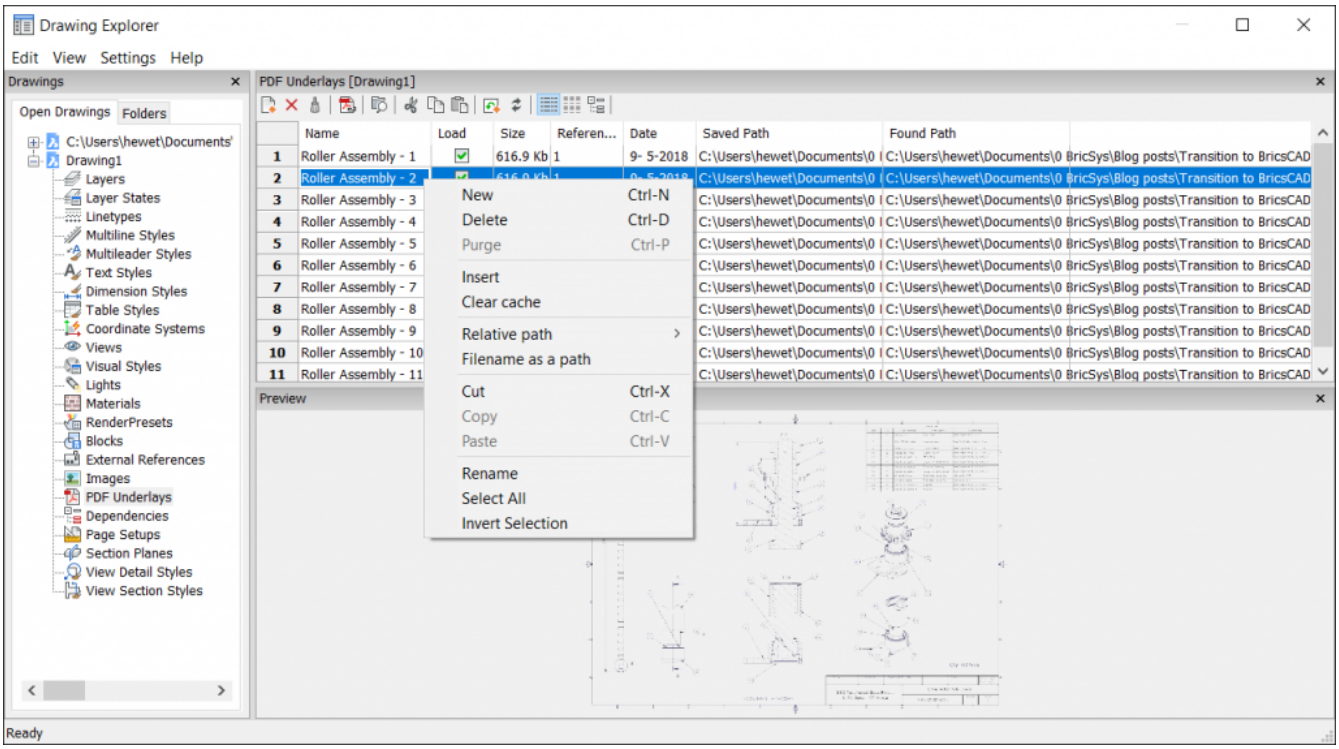


Figure 1.998



PDF Files The PDF Underlays panels display detailed information about PDF files that are linked to the drawing. The menu and toolbar offer the same tools as for images. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/v Attributes-ExternalReferences/HHBricsCAD-Working-with-ExternalFiles-08-1024x568.png}) Note: The PDFIMPORT (6.24.17) command imports geometry from PDF filesandconverts the content to simple CAD entities; also converts a PDF underlay already attached to the drawing to CAD entities.

Figure 1.999



Tutorial: Clipping Attached Images and PDF Files

Click here (1.3.16) to watch

1.22.19 Editing blocks and external references

Commands: REFEDIT (6.26.12), -REFEDIT (6.26.13), REFSET (6.26.14),REFCLOSE (6.26.11), BEDIT (6.6.7), -BEDIT (6.6.8) The **Refedit** command edits block references and externally-referenced drawings (Xrefs) through a dialog box (short for "reference editor"). **Refedit** is referred to as **in-place reference editing** . **RefEdit** either works on a portion of the reference or on the entire drawing or all entities in the block. The **-Refedit** command edits block references and externally-referenced drawings through the command bar. The **Refset** command adds and removes entities from the reference being edited. The **Refclose** command closes the reference editor. The **Bedit** command edits the entities that make up blocks in a temporary drawing session called the Block Editor, which uses the local coordinate system of the block. The **-Bedit** command prompts for the name of a name at the command line, and then opens the Block Editor environment The **XFADECTL** system variable sets the fading level for the entities which are not in the working in the current refedit session. Values between 0 (no fading) and 90 are accepted.

To edit Xrefs or blocks

1. Do one of the following:

   - Click the **Refedit** tool button () on the Refedit toolbar.
   - Type **refedit** in the command bar, then press Enter.
   - You are prompted: Select reference: Click the Xref or block.
   - Double click the Xref or block.

   The Reference Edit dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source

2. The **Reference Edit** dialog helps you to visually identify the reference to edit and controls how the reference is selected.
   - **Path** : Location of the selected reference; if the reference is a block, no path is displayed.
   - **Reference Name** : The name of the currently selected reference and any references, that are nested within the selected reference.
   - **Preview** : Displays the reference as it was last saved in the drawing. Please note that the preview image is not updated when changes are saved back to the reference.

3. (option) If multiple nested references display in the **Reference name** tree, select areference to modify. Click the expand/collapse button to show/hide nested references. You can edit only one reference a time.

4. Select one of the following to **Select nested entities ...** :

   - **Automatically** : All the entities and nested blocks in the selected reference are included in the reference editing session.
   - **Prompt** : After you close the **Reference Edit** dialog, you are prompted to select the enties and nested blocks in the reference that you want to edit. Only these entities will be temporarily extracted and be made available for modification within the context of the current drawing.

   This temporary working set of entities can be edited and then saved back to update the reference being edited. To distinguish 'current drawing' from 'editing set' BricsCAD displays all the entities of the current drawing as faded. The fading level is controlled through the **XFADECTL** settings variable, of which the default value is set to **50**.

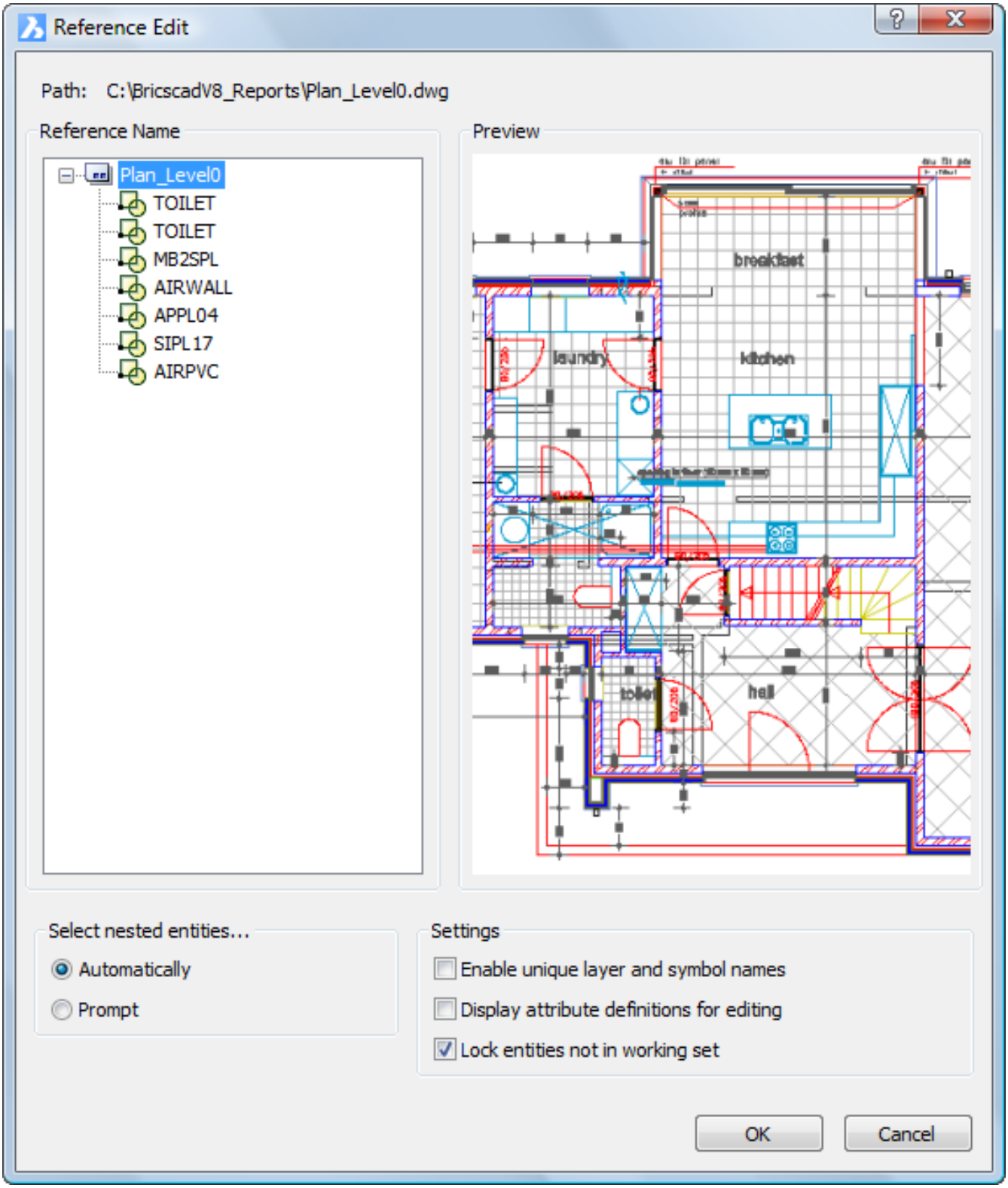
5. Check the **Settings**:

   - **Enable unique layer and symbol names** :
     - If selected, named objects in xrefs are altered, adding a **$#$** prefix.
     - If cleared, the names of layers and other named objects remain the same as in the reference drawing.
   - **Display attribute definitions for editing**: Controls whether attribute definitions in blocks are extracted and displayed during reference editing.
   - **Lock entities not in working set** : Entities in the reference being edited that are not included in the temporary working set (see step 5) cannot be manipulated, nor entities in the host drawing or other Xrefs. This behavior is similar to objects on locked layers.

6. Click the **OK** button to start editing the working set (see step 5). The Reference Edit dialog closes.

7. (option) Click the **Add to Refedit** tool button () to add entities to the working set. The selected entities will be removed from the host drawing and added to the reference being edited when the working set is saved back.

Figure 1.1000



8. (option) Click the **Remove from Refedit** tool button () to remove entities from the working set. The selected entities will be removed from the reference being edited when the working set is saved back. These entities are added to the host drawing.

9. Do one of the following:

   - Click the **Refclose and Save** button () to save the changes back to the Xref source drawing or the block definition in the current drawing.
   - Click the **Refclose and discard** button () to stop the refedit session without saving. The source drawing or block definition is left unchanged.

1.22.20 Working with groups

Commands: GROUP (6.15.22) and -GROUP a (6.15.23) The **Group** command creates and modifies named groups of entities, in a dialog box. The **-Group** command creates and modifies named groups of entities, at the command line. Working with groups is a technique to manipulate a set of several objects as though it were a single object. A group can also be considered as a 'Named Selection'. You can pick a single object and the entire set is selected as though it were a block, or you can edit individual objects without affecting the rest of the group and without breaking up the group. You can use nested groups if necessary and a single entity can be a member of different groups. **Note:** Make sure the value of the **PICKSTYLE** variable is set to either 1 or 3 to enable the selection of groups.

To create a group

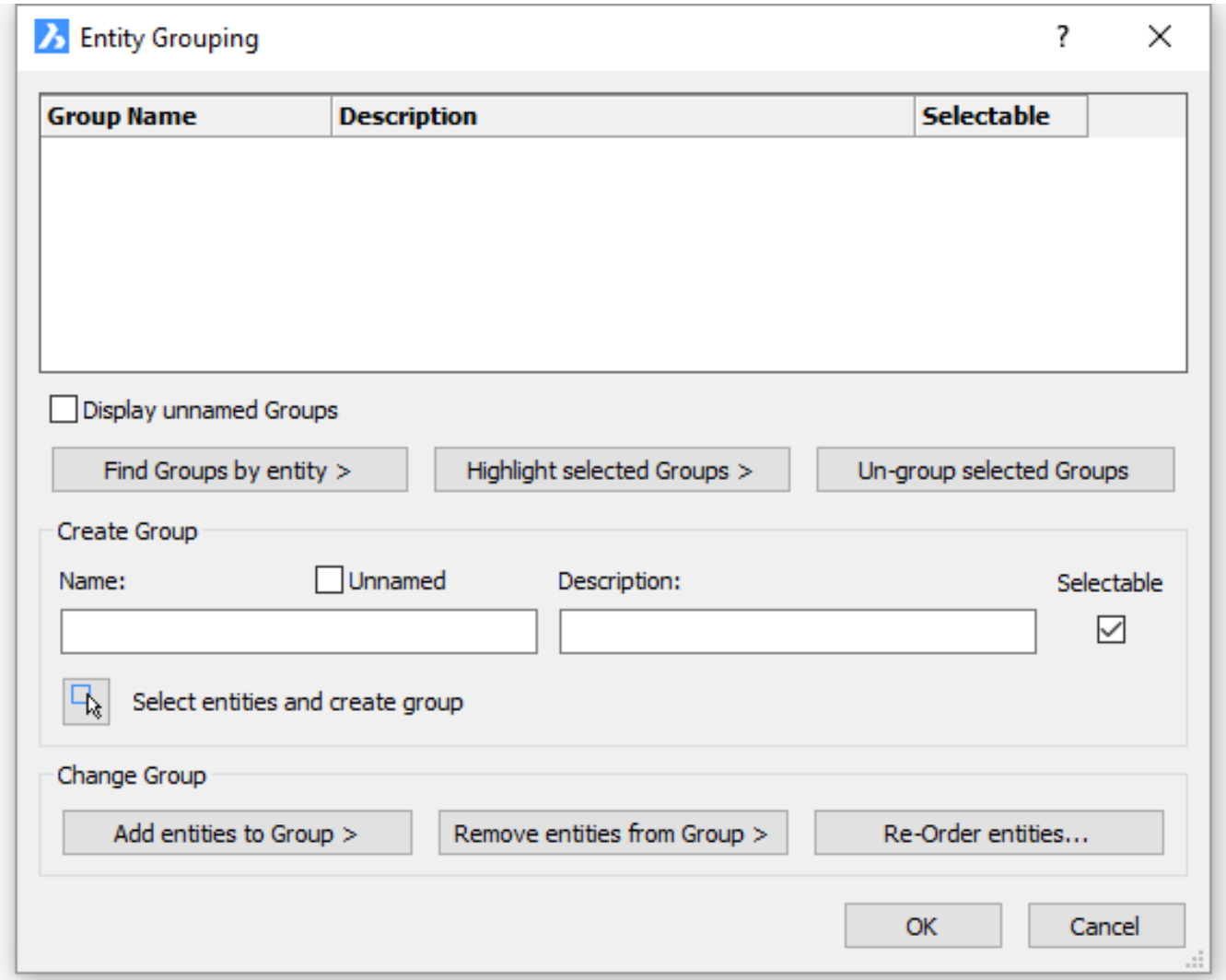
1. Do one of the following:

   - Click the **Group** tool button ( ) on the Tools toolbar.
   - Choose **Group** in the **Tools** menu.
   - Type **group** in the command bar, then press Enter.

   The **Entity Grouping** dialog window opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG

2. Do one of the following:
   - Type a name in the **Name** field.
   - Check the **Unnamed** option to create an unnamed group.
3. (option) Type a description in the **Description** field.
4. (option) Check / uncheck the **Selectable** option.
5. Click the **Select entities and create group** button. The **Entity Grouping** dialog temporarily closes to let you select entities.
6. Select the entities (1.23.6) you want to include in the new group.
7. Right click to conclude the selection of entities. The **Entity Grouping** dialog reopens. The newly created group is added.
8. Click the **OK** button to close the Entity Grouping dialog

Figure 1.1001



To modify a group

1. Do one of the following:
   - Click the **Group** tool button ( ) on the Tools toolbar.
   - Choose **Group** in the Tools menu.
   - Type **group** in the command bar, then press Enter.

   The **Entity Grouping** dialog window opens.
2. Select the group you want to modify.
3. (option) Double click in the **Name** field to edit the group name.
4. (option) Double click in the **Description** field to edit the group description.
5. (option) Click in the **Selectable** column to toggle the **Selectable** option on/off.
6. (option) Click the **Add entities to group** button to add new entities to the group. The **Entity Grouping** dialog temporarily closes to let you select the new group entities.
7. (option) Click the **Remove entities from group** button to remove entities from the group. The Entity Grouping dialog temporarily closes to let you select the entities.
8. Click the **OK** button to close the Entity Grouping dialog

To ungroup entities

1. Do one of the following:
   - Click the **Group** tool button ( ) on the Tools toolbar.
   - Choose **Group** in the Tools menu.
   - Type **group** in the command bar, then press Enter.

   The Entity Grouping dialog window opens.
2. Select the group or groups you want to delete.
3. Click the Un-group selected groups button. The selected groups are removed.

Note: When you ungroup entities, the entities remain in the drawing but the group is deleted from the drawing.

To change the order of entities

1. Do one of the following:
   - Click the **Group** tool button ( ) on the Tools toolbar.
   - Choose **Group** in the Tools menu.
   - Type **group** in the command bar, then press Enter.

   The **Entity Grouping** dialog window opens.
2. Click the **Re-Order entities** button. The Order Group dialog opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom
3. (option) Click the **Highlight** button to see the current position of each entity. The **Object Grouping box** opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExternalRefe
4. (option) Click the **Reverse Order** button to reverse the order of all entities in the group.
5. (option) To change the order of a single entity do the following:
   - In the **Remove from position** field type the current position of the entity you want to move.
   - In the **Enter new position number for the object** field type the new position.
   - (option) Type the number of objects you want to reorder in the **Number of objects** field.
6. Click the **Re-Order** button. A message box displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

Figure 1.1002

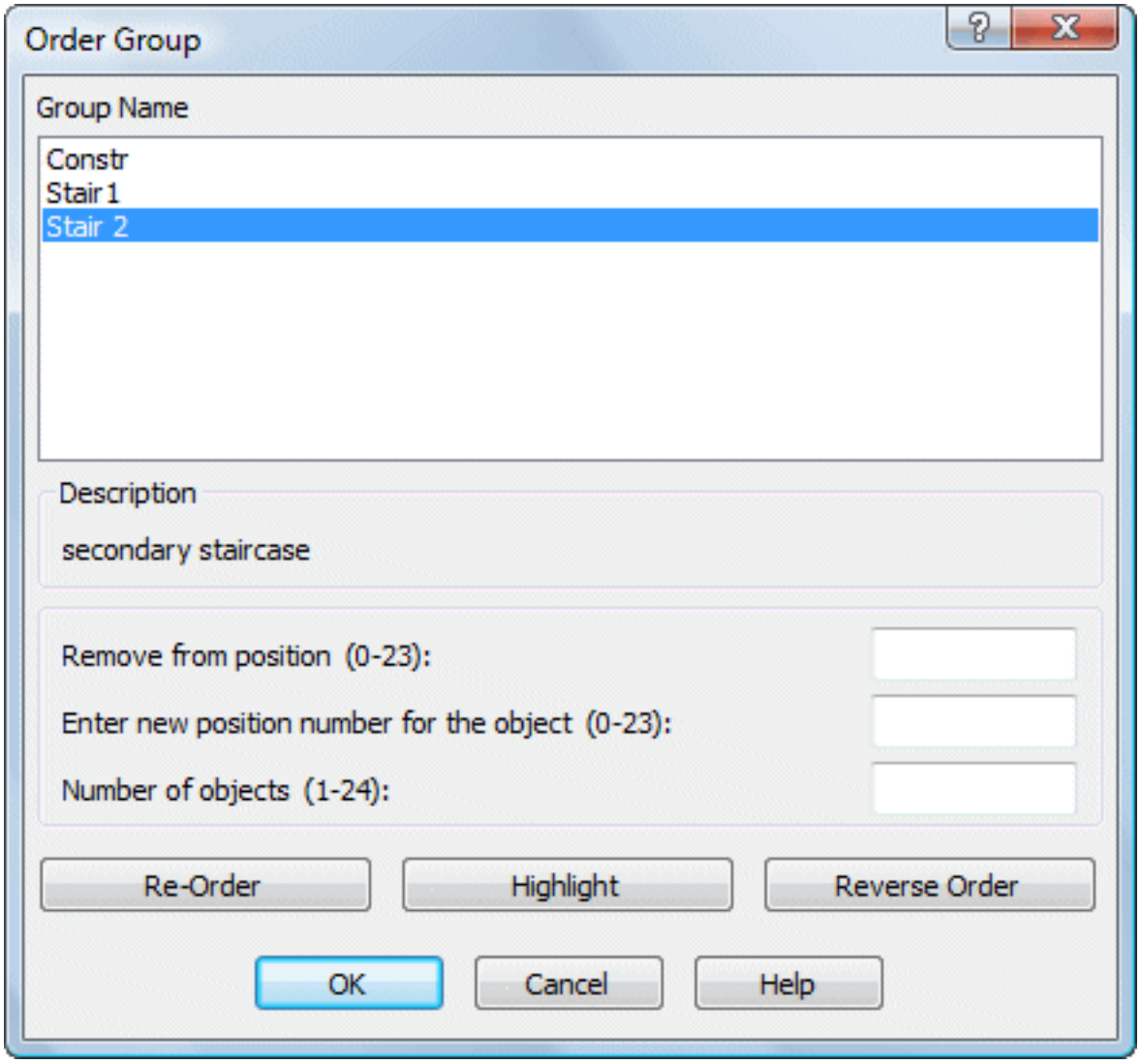


Figure 1.1003

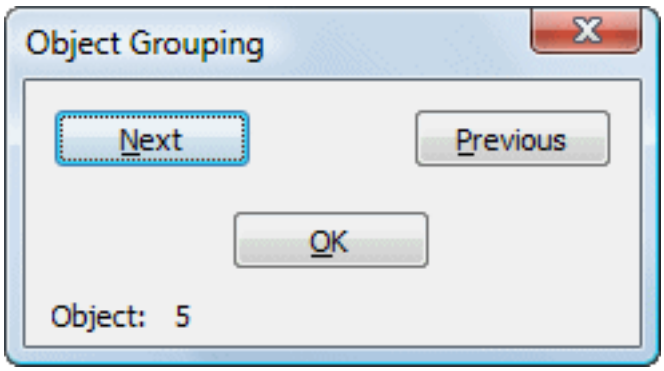
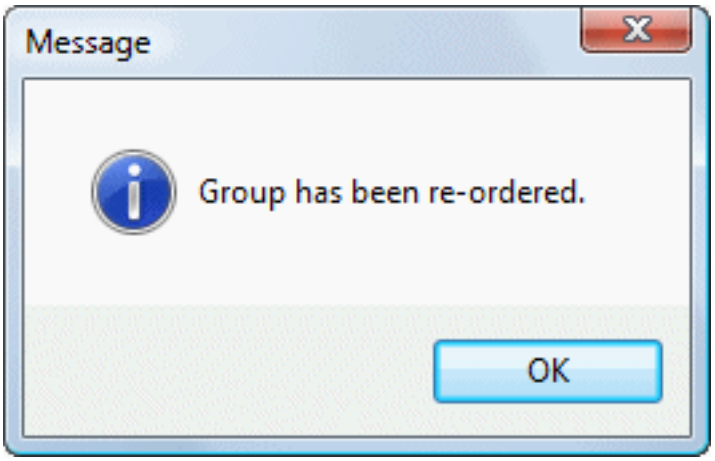


Figure 1.1004



1.22.21 Working with underlays

Commands: PDFADJUST (6.24.13), PDFATTACH (6.24.14), -PDFATTACH (6.24.15), PDFLAYERS (6.24.18), PDFCLIP (6.24.16) PDF files can be attached as an underlay to a drawing file. Much like raster image files and external references (Xrefs) a PDF underlay is not part of the drawing, but is linked to it. In the parent drawing the path to the underlay is saved. You can edit the path to make sure the correct PDF is found. By default the folder of the parent drawing is search first. If the PDF underlay is not found there, the saved path folder is searched. If the PDF underlay is not found there either, BricsCAD reports a 'Missing or invalid reference' in the drawing. The **PdfAdjust** command allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar. The **PdfAttach** and **-PdfAttach** commands insert a PDF file as an underlay into the current drawing. The **PdfLayers** commandaAllows to control the display of layers in a PDF underlay. The **PdfClip** command crops the display of a PDF. You can choose between a polygonal or a rectangular boundary. Each instance of the same PDF underlay file can have a different boundary.. The **PDFOSNAP** system variable allows to snap to the geometry in the PDF underlay if the system variable is On. The **PDFNOTIFY** system variable controls whether warnings for missing or modified pdf underlays are issued. The **XNOTIFYTIME** system variable defines the frequency to check for modified externally referenced resources: xrefs, images and pdf attachments if **XREFNOTIFY, IMAGENOTIFY** and/or **PDFNOTIFY** is ON. If the value of the **Frame** system variable = 3, the **PDFFRAME** system variable controls the visibility of PDF frames. The options are:

- 0: Hide PDF frames.
- 1: Display and plot PDF frames.
- 2: Display but do not plot PDF frames.

To control the PDF underlay caching

A multi-resolution persistent image cache is used to display attached Pdf underlays, enabling (very) fast zoom and pan operations. The highest cached resolution is 5000 x 5000 pixels. Still, when zooming in very close, the display of the Pdf underlay will become pixelated. So a hybrid modus can be used which switches to real-time generation of crisp Pdf underlay display when zooming in very close. The initial generation of the image cache may take a few seconds, from then on processing gets (very) fast, and remains like that in subsequent sessions. PDFCACHE (1.13.11): System variable that controls how pdf caching is applied

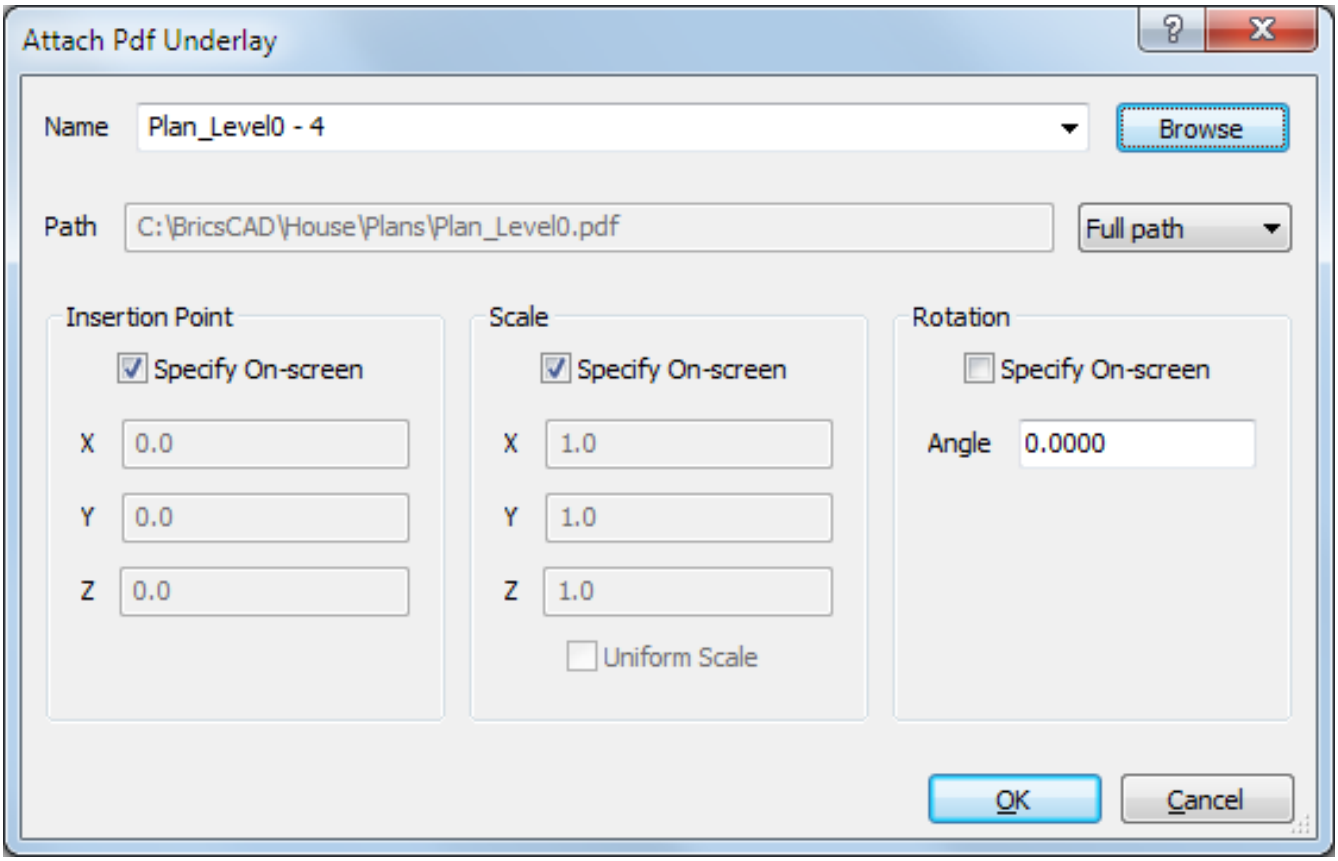
- 0 = don't use Pdf cache, always use real-time generation of the Pdf underlay image. (this mode was the default in previous BricsCAD versions)
- 1 = use Pdf cache, switch to real-time generation when zooming in very close.
- 2 = always use Pdf cache (default).

To attach a PDF underlay

1. Do one of the following:
   - Click the **Attach pdf** tool button () on the Insert toolbar.
   - Click the **New** tool button () on the Details toolbar of the Drawing Explorer - PDF Underlays (1.14.2) dialog.
   - Type **PDFattach** in the command bar, then press Enter.

   The Select PDF Underlay File dialog opens.
2. Select the file, then double click the file or click the **Open** button on the Select PDFUnderlay File dialog. The Attach PDF Underlay dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source
3. In case of a multi-page PDF, click the down arrow in the **Name** field, then select the pageof the PDF you want to load.
4. Click the **Path** option button, then select either:
   - **Full path**: The full path will be searched when the drawing is loaded. If the overlay is not found there, the folder of the parent drawing is searched. If the overlay file is not found there either, "Missing or invalid reference" displays at the insertion point of the PDF overlay.
   - **Relative path**: The relative path, with respect to the folder of the parent drawing, will be searched when the drawing is loaded. If the overlay is not found there, the folder of the parent drawing is searched. If the overlay file is not found there either, "Missing or invalid reference" displays at the insertion point of the PDF overlay.

Figure 1.1005



- **No path**: Only the folder of the parent drawing is searched when the drawing is loaded. If the overlay file is not found there, "Missing or invalid reference" displays at the insertion point of the PDF overlay.

5. Choose whether you want to specify the **Insertion Point** , **Scale** and the **Rotation** angleon-screen or not.
6. Click the **OK** button. The Attach PDF Underlay dialog closes.
7. Depending on the insertion options chosen in the previous step you are prompted to specify the **Insertion Point**, **Scale** and/or **Rotation** angle.

To insert a PDF underlay

If you want to insert a second instance of a PDF that is already attached or another page of such underlay do the following:

1. Choose **Drawing Explorer > PDF Underlays...** in the Tools menu. The Drawing Explorer - PDF Underlays dialog opens.
2. Select the PDF underlay.
3. Do one of the following:

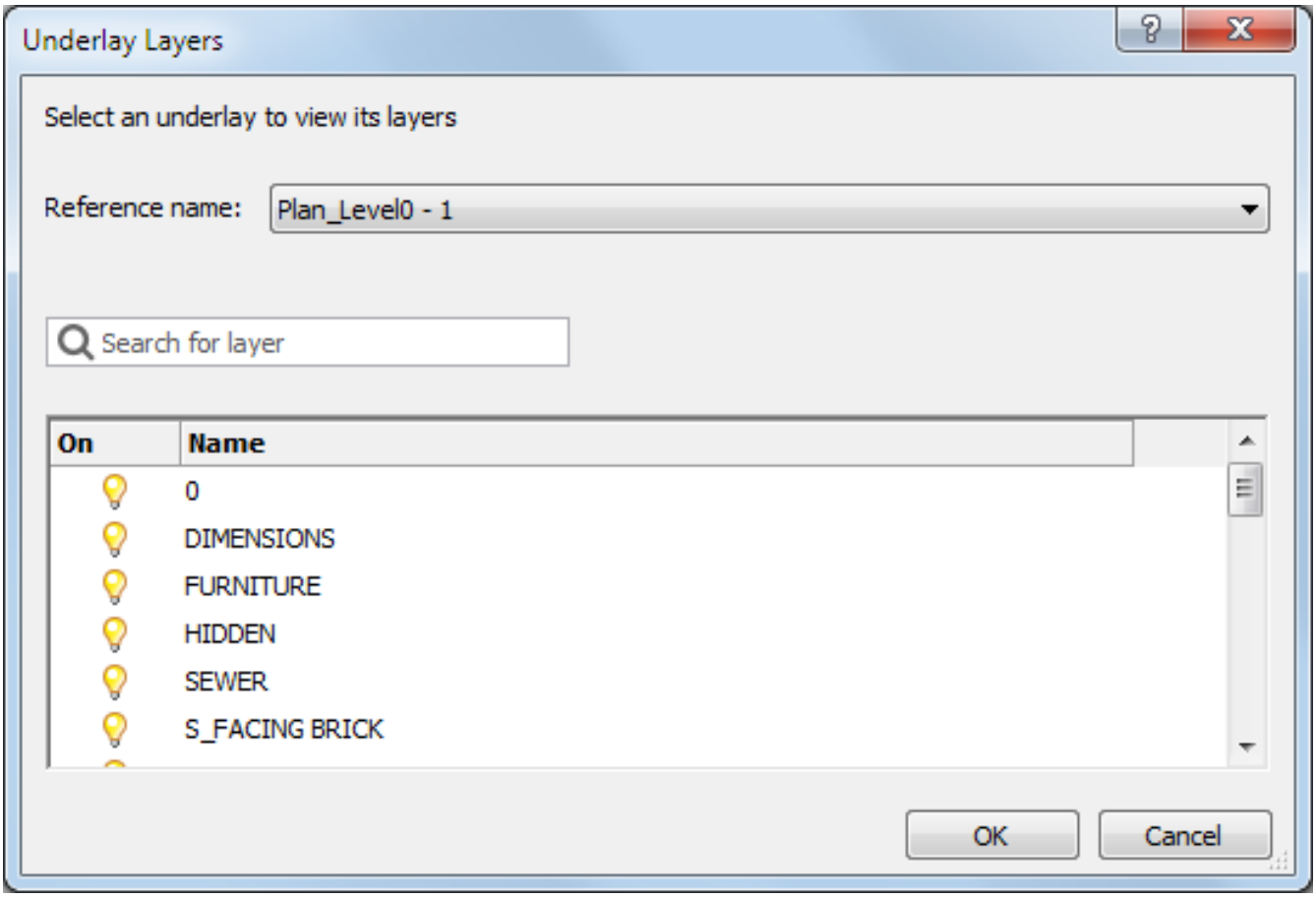
   - Right click and choose Insert in the context menu.
   - Choose insert in the Edit menu on the Drawing Explorer - PDF Underlays dialog.

   The Drawing Explorer - PDF Underlays dialog closes.
4. Continue with steps 3 through 7 of the previous procedure.

To control the display of a PDF underlay

1. Click the frame of the PDF underlay. The frame of the selected PDF highlights. The properties of the underlay display in the Properties Bar (1.12.4).
2. Under **Misc** , click **Show underlay** in the Properties Bar, then select **Yes** or **No**.

Figure 1.1006



To set the layer display in a PDF underlay

1. Type **PDFlayers** in the command bar, then press Enter. You are prompted: Select PDF underlay.
2. Click the frame of the PDF underlay. The frame of the selected PDF highlights. The Underlay Layers dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttributesExter
   The icon in the **On** column indicates the current state of the layers On ( ) or Off ( ).
3. Click the icon in the **On** column to toggle the display of a layer.
4. (option) Click the **Search for layer** field, then type a layer name to search for a layer. Use wildcard characters (? or *) to limit the number of layers in the list: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG
   Layernames starting with 'S_' only. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sourc
   Layernames of which the second and third character is '_F' only.
5. Do one of the following:
   - Click the **Reference name** button to select another PDF underlay.
   - Click the **OK** button to stop.

Figure 1.1007

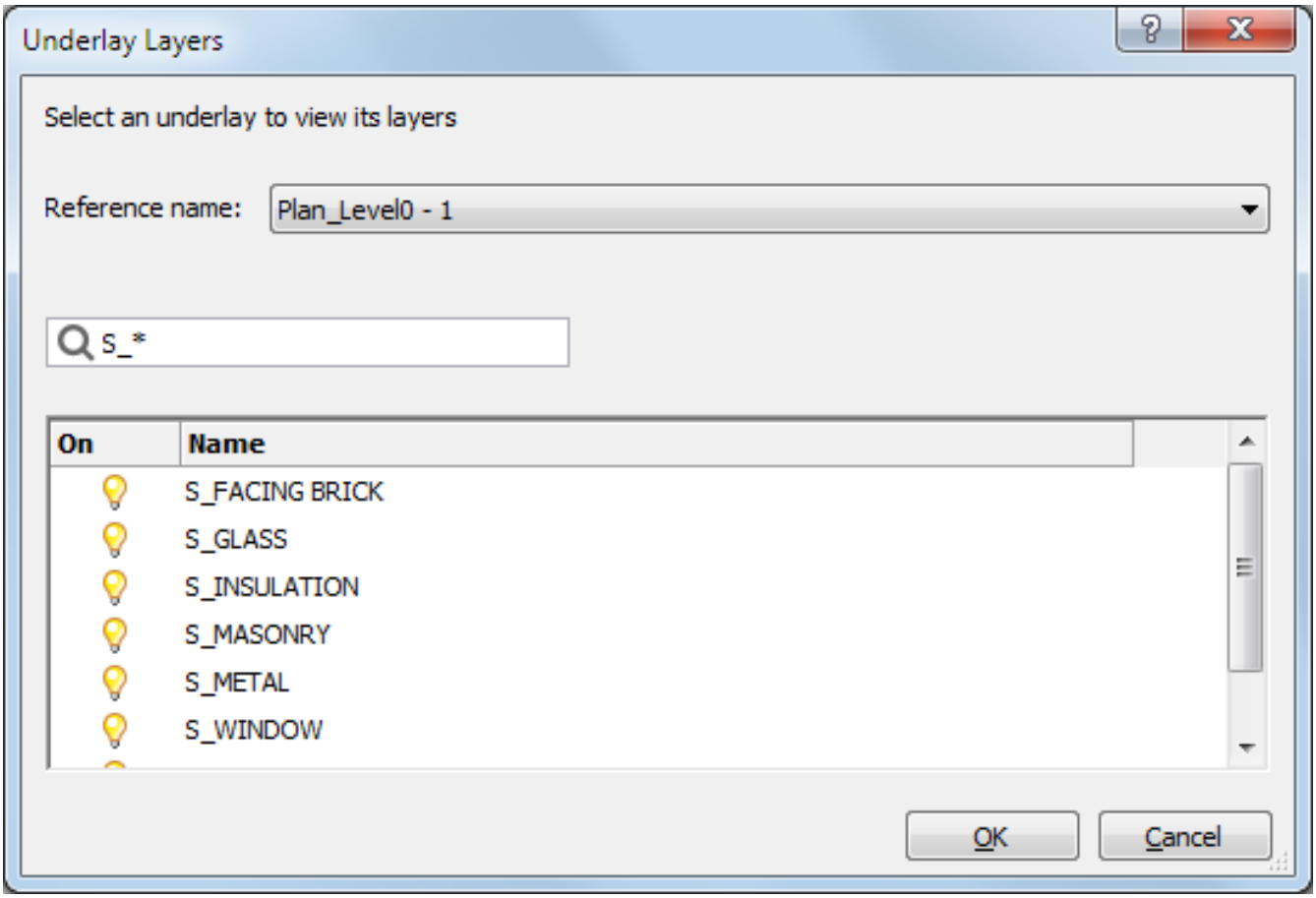
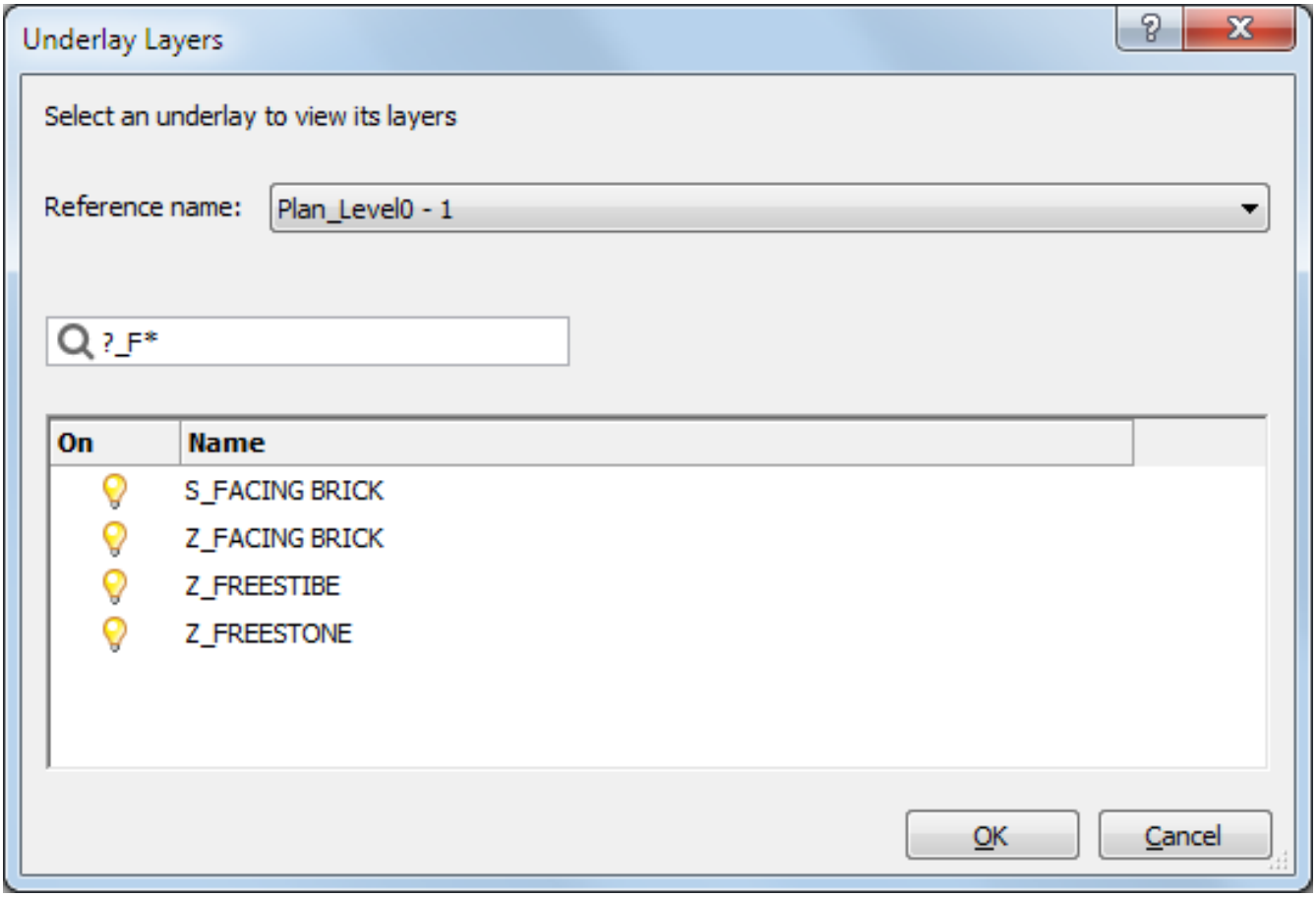


Figure 1.1008

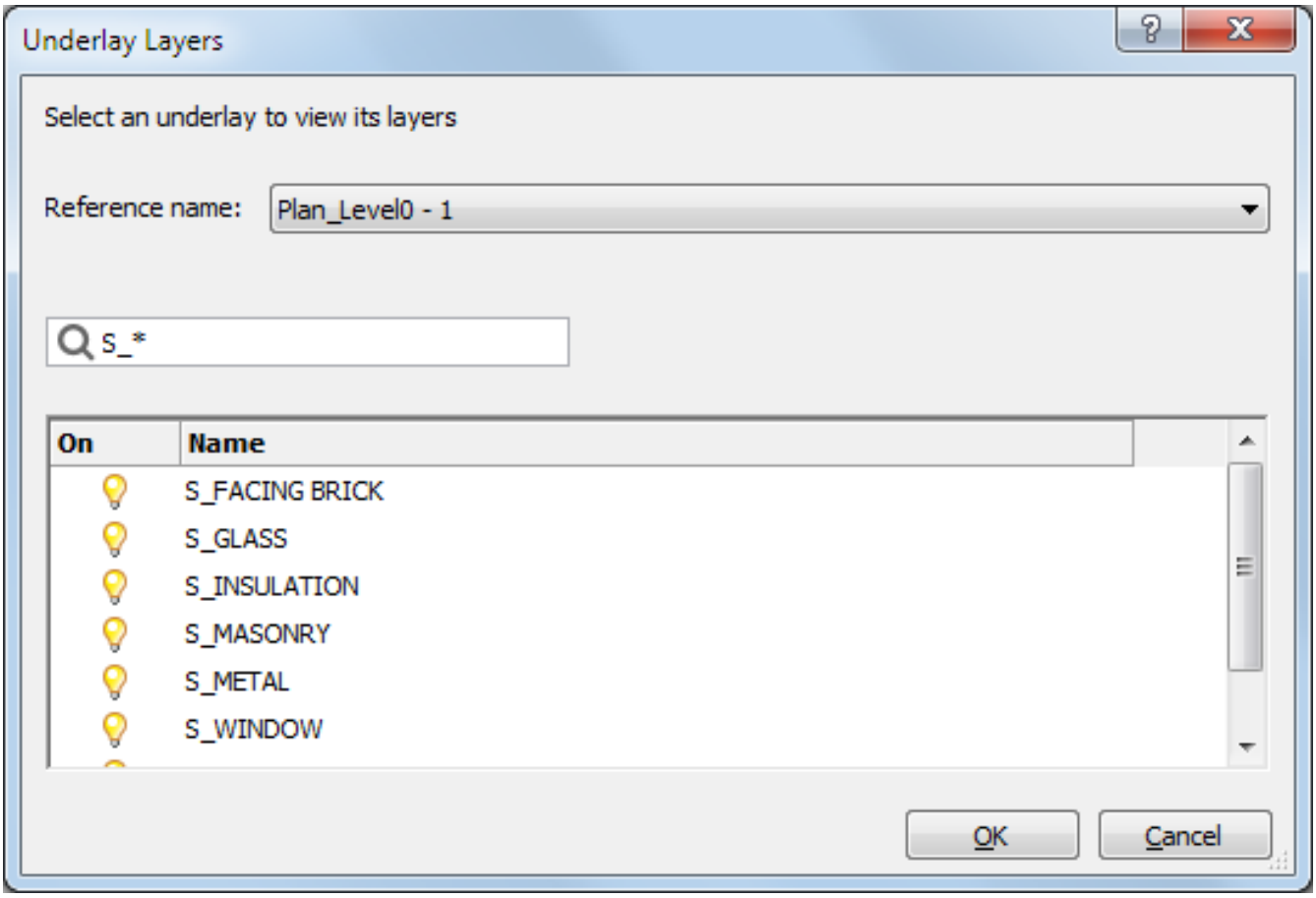


To clip a PDF underlay

1. Type **pdfclip** in the command bar, then press Enter. You are prompted: Select PDF underlay:
2. Click the frame of the PDF underlay. The frame of the selected PDF highlights. You are prompted: Enter PDF clipping option [ON/OFF/Delete/New] <New>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
3. Do one of the following:
   - Choose **New boundary** in the prompt menu.
   - Press Enter to accept the **New** default command option.

   You are prompted: Enter PDF clipping type [Polygonal/Rectangular] <Rectangular>:
4. Do one of the following:
   - Press Enter to accept the Rectangular default command option to define a rectangular clipping boundary.
   - Type P, then press Enter to define a polygonal clipping boundary.
5. Define the clipping boundary. The defining points must lie inside the frame of the PDF underlay. If you click outside the PDF underlay the point is placed on the PDF underlay frame.

Figure 1.1009



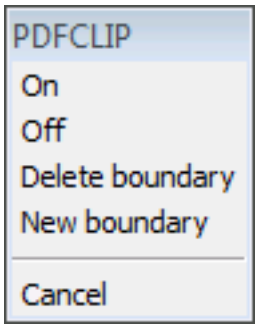
To toggle the display of the clipped part of a PDF underlay

1. Click the frame of the PDF underlay. The frame of the selected PDF highlights. The properties of the underlay display in the Properties Bar (1.12.4).
2. Under **Misc** , click **Show clipped** in the Properties Bar, then select **Yes** or **No**.

To delete a clipping boundary

1. Type **pdfclip** in the command bar, then press Enter. You are prompted: Select PDF underlay:
2. Click the frame of the PDF underlay. The frame of the selected PDF highlights. You are prompted: Enter PDF clipping option [ON/OFF/Delete/New] <New>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
3. Do one of the following:
   - Choose **Delete boundary** in the prompt menu.

Figure 1.1010



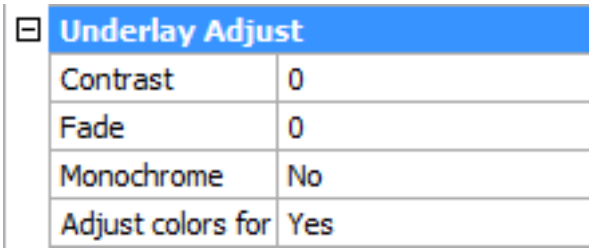
- Type D in the command bar, then press Enter.

The clipping boundary is deleted and the complete PDF underlay displays.

To adjust the display settings of a PDF underlay

The PDFADJUST (6.24.13) command allows to adjust the **Fade** , **Contrast** and **Monochrome** settings of a PDF-underlay in the command bar. Select the PDF underlay, then adjust the display settings under **Underlay Adjust** the Properties Bar (1.12.4): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/12BlocksAttri

Figure 1.1011



1.22.22 Components and Component Inserts

A component is a named group of entities, which can be inserted in a drawing with theBmInsert (6.8.12) command. Components can be parametric: e.g. windows and doors (5.3.9)in a BricsCAD BIM and standard mechanical components (3.6.9)in BricsCAD Mechanical. Any .dwg file can be used as a component. The same component can be inserted once or several times into one or more other components. When the component is modified, all inserts of the component will reflect these changes automatically. Components cannot be self-referencing. A component insert is a named entity, of which the default name is composed of the name of the component and the serial number of the insert. Internally a component insert is represented either as a reference to an anonymous block or as an external reference (1.22.22). Properties of inserted components, including their parameters can be edited in theMechanical Browser (3.2.2) andProperties (6.24.49) panel.

External and local components

Components can be either local or external. A local component is stored in the .dwg file with the assembly as anonymous block. An external component is defined in a separate .dwg file and represented by an xref or by a local copy stored as anonymous block. When the corresponding .dwg file is modified all inserts of external components can be updated using BMUPDATE (6.8.25) command or Mechanical Browser context menu. If you use external components in your assembly, always remember to transfer all related files when you want to share your model. Use the ETRANSMIT (1.14.10) command to create a package of a drawing file and all its dependencies, such as external references, images, font files, plot configuration files, plot style tables and font map files. You can always convert a local component to an external one and vice versa. To edit a local component, it first needs to be converted to an external component using the BMEXTERNALIZE (6.8.8) command. Alternatively, you can use the BMOPENCOPY (6.8.18) command to open a copy of a component, edit and save it, and then use the BmReplace (6.8.21) command to replace the original with the copy. You can choose the default insertion type for your component: **Local** or **External.** To change the type, select the root node in the Mechanical Browser (3.2.2) and select **Local** or **External** from **Insert as** properties: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03D
Depending on the value of BMAUTOUPDATE (1.13.11), inserts of external components are updated automatically on opening the assembly document or manually with the BmUpdate (6.8.25) command. You can change this value in the Settings dialog (1.13.11): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Dra

Figure 1.1012

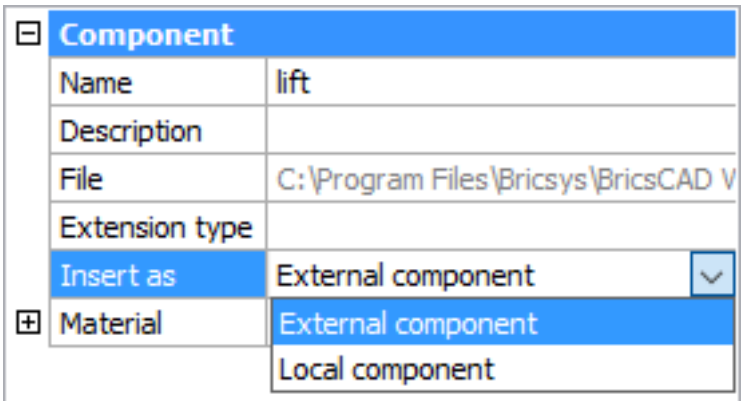
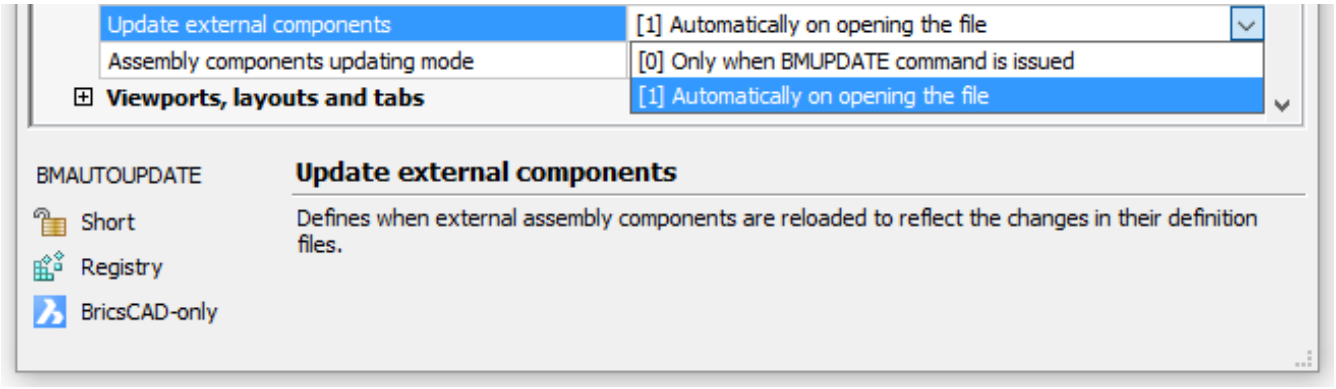


Figure 1.1013



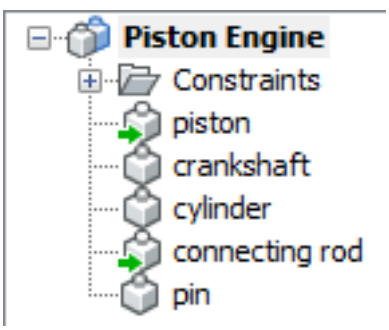
Converting components

Do one of the following:

- Click the **Switch Component to External** ( ) or **Switch Component to Local** tool on the **Assembly**ribbon tab. The tools are also available on the Assemblytoolbar and in the Assemblymenu;
- Right click on the component insert in the Mechanical Browser (3.2.2) and select **Switch to External** or **Switch to Local** in the context menu;
- Hover your mouse cursor over a component insert in a drawing window and select **Switch to External** ( ) or **Switch to Local** (===ERROR WITH DOWNLOADED IMAGE===) in the **Assembly Modeling** command group in the Quad (2.2.3);
- Type **BmExternalize** or **BmLocalize** in the command bar.

When you convert an external component to local, the .dwg file with the component definition is not deleted, but its future changes will not affect your assembly. You can always distinguish between inserts of local components ( ) and inserts of external components ( ) by their icons in the Mechanical Browser (3.2.2): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/M

Figure 1.1014



Parametric Components

BricsCAD allows users to make parametric changes of inserted components at the assembly level. A component is considered parametric if it contains at least one 2D (1.15.6) or 3D (1.19.4) dimensional constraint or an associative array with an expression. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingE

NOTES

- BricsCAD automatically maintains associativity between the definition of a parametric component and its inserts. So you can open the parametric component for editing, modify its geometry, save it, and then execute the BmUpdate (6.8.25) command to update the mechanical structure of your assembly see the result.
- If you need to go back to the default value of a component' parameter, just edit the **Expression** field and delete its content.

You can control whether a particular component parameter will be available for editing in a drawing containing the inserts of this components (at the assembly level). To do that, select the corresponding parameter in the Mechanical Browser (3.2.2)and change the value of the **Exposed** field in the properties grid of the parameter: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/P
The options are:

- **Off**: Hides the parameters at the assembly O.
- **On**: Makes the parameter visible at the assembly level.
- **Auto**: Hides all parameters which values are controlled by expressions, and exposes all other parameters.

You can associate units with a parameter. If you assign a value, directly or via an expression, to a component parameter at the assembly level, it is automatically converted from drawing units of the assembly drawing to the units of the parameter. Values assigned to unitless parameters are not converted. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/ParameterPropert
By default all parameters are unitless except parameters of dimensional constraints and parameters, which are used as expressions for dimensional constraints. BricsCAD automatically assigns drawing units to such parameters.

Figure 1.1015

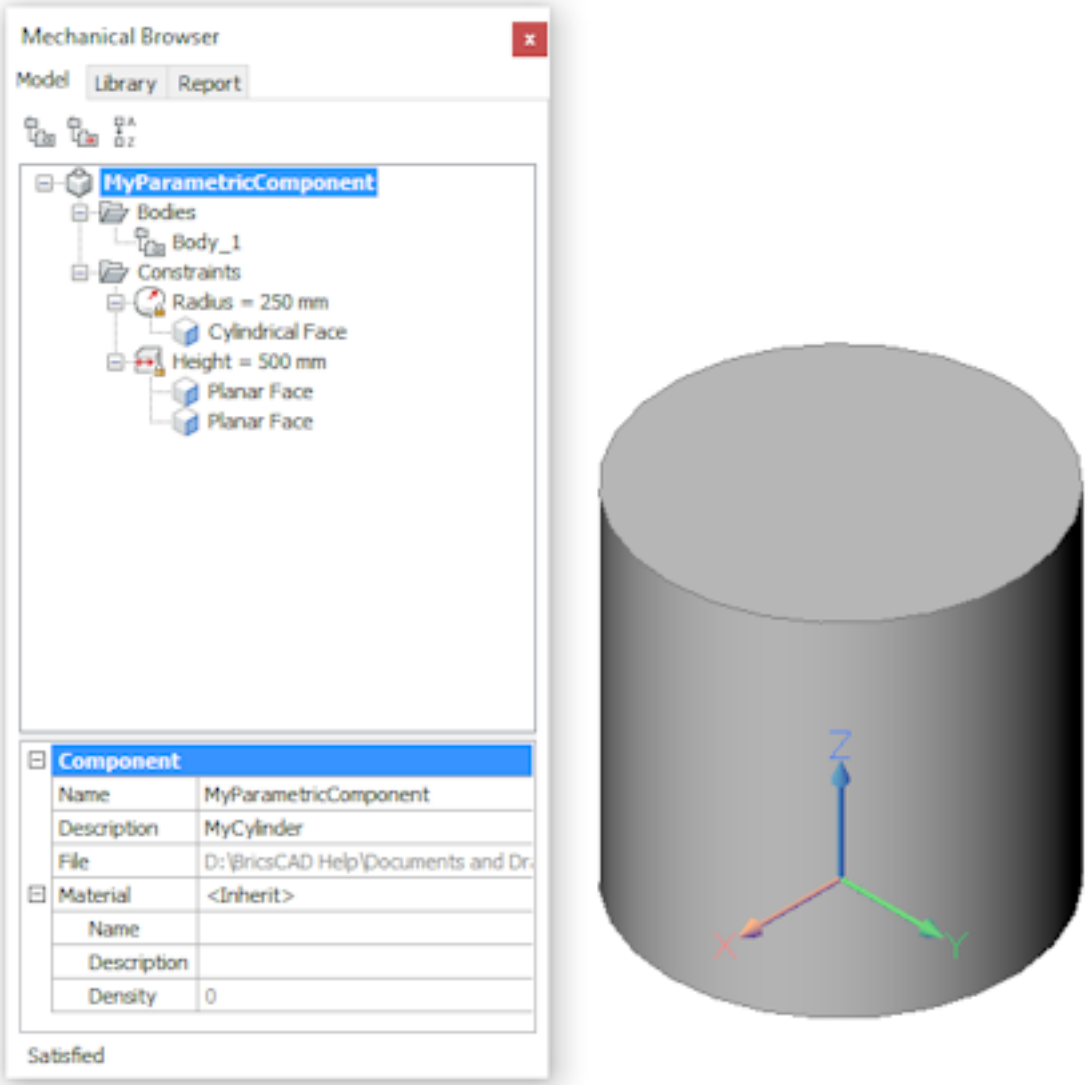


Figure 1.1016

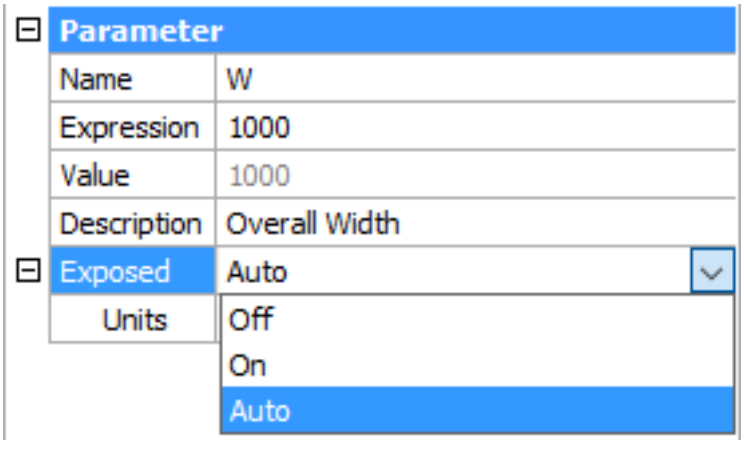
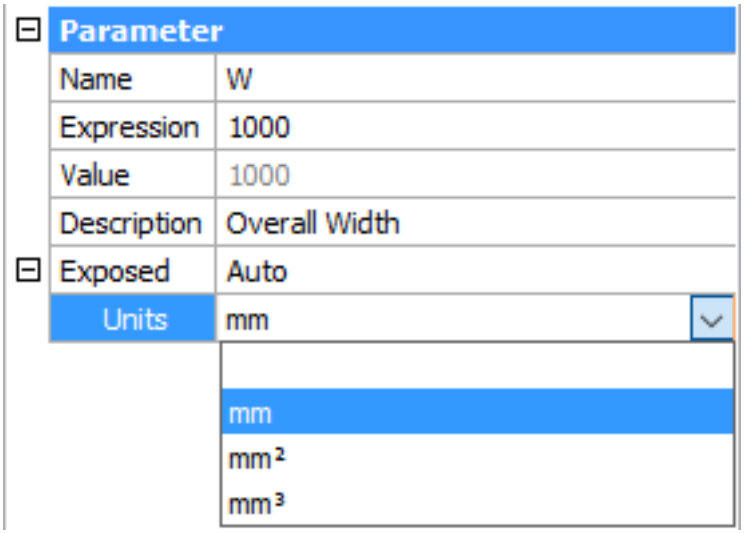


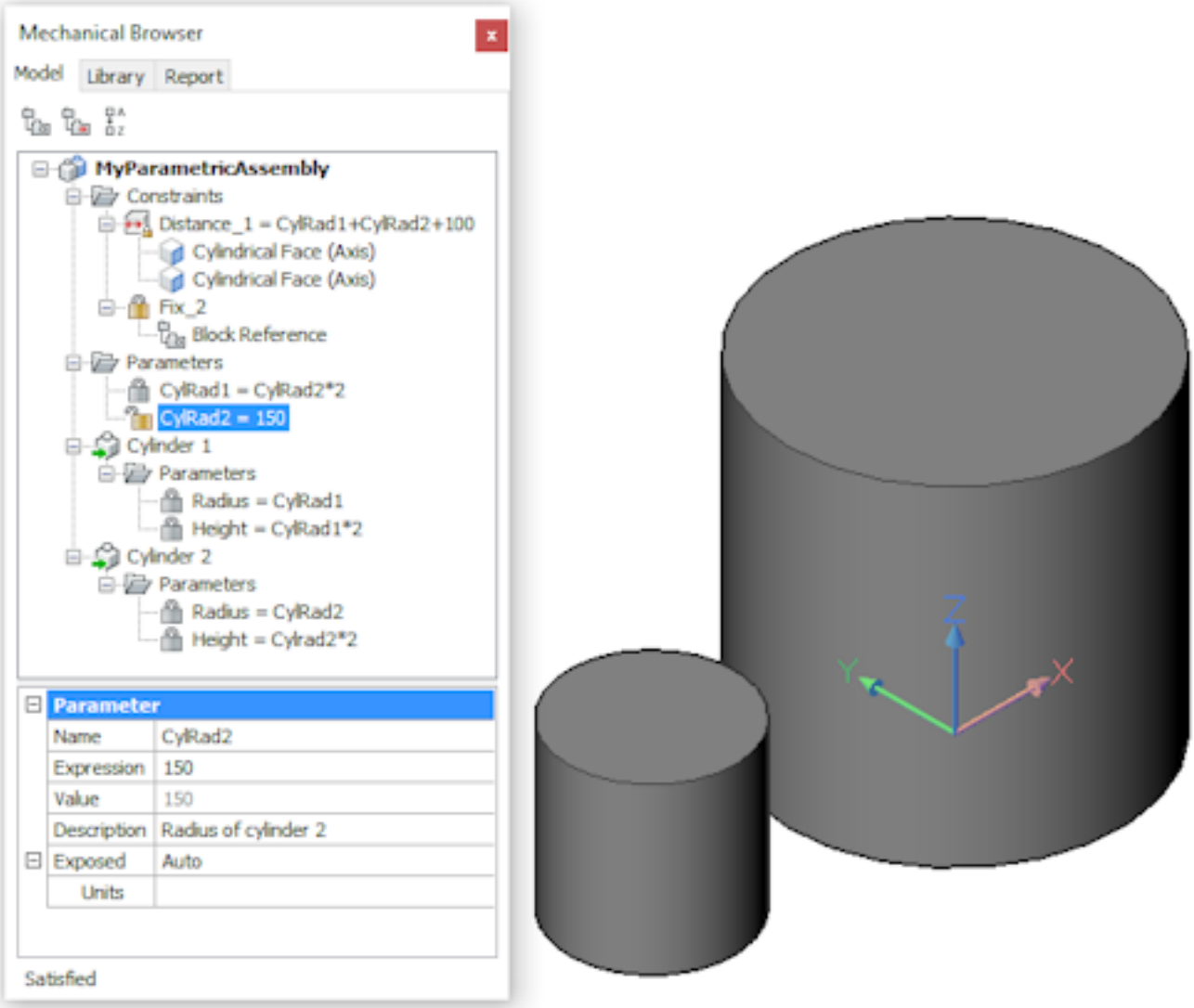
Figure 1.1017



To edit the parameters of a parametric component insert

When a parametric component is inserted in a model, you can edit the parameters either in the Mechanical Browser (3.2.2) or in the Properties Bar (1.12.4). The component insert immediately updates according to the new value of a parameter. Please note that this does not modify the definition of the parametric component, which is a separate drawing file. As a result, multiple inserts of the same parametric component can exist in the same model, with different values for each of their parameters: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source

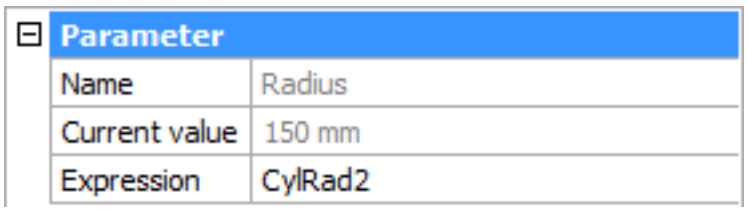
Figure 1.1018



To edit a parameter in the Mechanical Browser

1. Expand the **Parameters** node of the component.
2. Select the parameter.
3. In the **Parameter** section, type a new value in the **Expression** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

Figure 1.1019



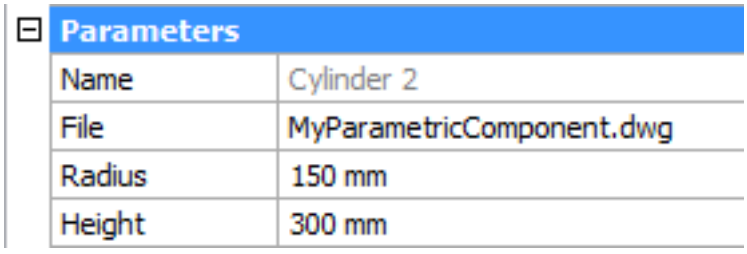
To edit a parameter in the Properties Bar

1. Select the component in the drawing.
2. Expand the **Parameters** settings group.
3. Type a new value for one of the available parameters. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

NOTES

- In the Properties Bar (1.12.4) always the current value displays. In case this value is controlled through an expression, the expression will be overwritten by the newly entered value.
- If multiple parametric components are selected, the shared parameters can be edited simultaneously. The settings fields of parameters that currently have different values read *Varies*. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sou

Figure 1.1020



Replacing Components

Component inserts can be replaced with another component using the BmReplace (6.8.21) command. Only top-level inserts can be replaced.

1. Do one of the following:

   - Click **Replace Component** tool () on the **Assembly** ribbon tab. The tool is available on the Assembly toolbar and in the Assembly menu.
   - Right click on the component insert in the Mechanical Browser (3.2.2) and select **Replace...** or **Replace all inserts...** in the context menu.
   - Hover over a component insert in a drawing window and select the **Replace** tool () in the Quad (2.2.3)
   - Type **BmReplace** in the command bar, the File open dialog displays.

2. Select a .dwg file for the replacement component and press the **Open** button. You are prompted: Replace file for all similar inserts [Yes/No] <Yes>:

3. Do one of the following:

   - **Yes** : Replace all inserts of the same name.
   - **No** : Replace the selected insert only.

Note: Parameter values which are shared between the replacement component and the replaced component are copied automatically, which allows intelligent replacement of parametric components.

Difference between components, blocks and external references

Components are based on blocks and external references (xrefs), but the behavior of components is different. An external component keeps a link to a file, much like an xref does. But an external component can either be represented by an xref (BMINSERT (6.8.12) acts like XATTACH (6.34.1)) or by a local anonymous block (BMINSERT (6.8.12)acts like INSERT (6.17.13)). Components can be either parts or assemblies:

- **Part**: composed of entities only; no inserts of subcomponents.
- **Assembly**: composed of entites and subcomponent inserts. Even when the subcomponents are removed, the assembly status is kept.

When an external component is a part and its parameters are not modified, then the component is inserted as an xref, which is listed in the External References section of the Drawing Explorer (1.14.2) dialog. Xref layers have a file_name prefix. When an external component is an assembly, then it is inserted as a local anonymous block. All its layers are merged with the layers of main document and no prefix is added to layer name.

Converting blocks and external references

Drawings with blocks and external references can be converted to mechanical assemblies: block references are converted to inserts of local components, and external references are converted to inserts of external components.

1. Do one of the following:

   - Click the **Initialize Mechanical Structure** tool () in the ribbon tab. The tool is also available in the **Assembly** toolbar and in the Assembly menu.
   - Type **BmMech** in the command bar

2. You are prompted: Name for new mechanical component <test>: Type a name for the mechanical component or press Enter to accept the current name or the default **Component_1** name.

3. You are prompted: Convert blocks and external references to mechanical components? [No/Yes] <Yes>:Press **Enter** to convert blocks and external references to mechanical components.

Mechanical components can be converted back to blocks and external references.

1. Do one of the following:

- Click The **Remove Structure** tool () on Assembly ribbon tab. The tool is also available in the Assembly toolbar and in the Assembly menu.
- Type **BmUnmech** in the command bar.

2. You are prompted: Warning: this command removes all mechanical structure from the current document. Proceed? No/<Yes>: Press **Enter** to:
   - Convert external components to External References (Xrefs).
   - Convert local components to blocks.

Component-based features

A feature is a physical constituent of a part, which has engineering significance. Examples of features are holes, ribs, slots, pockets, forms. Being applied to a 3D solid, a feature removes and/or adds geometry to. You can create your own features using BricsCAD component extension technology. An extended component has **BC_SUBTRACT** and/or **BC_UNITE** layers. When you insert such a component on a face of a 3D solid using the BmInsert (6.8.12) command, this target solid is modified as follows:

- All 3D solids on the**BC_SUBTRACT** layer of the component are subtracted from the target 3D solid.
- All 3D solids on the**BC_UNITE** layer of the component are united with the target 3D solid.

All faces created after these Boolean operations form a component-based feature, are updated when you move the component insert or when modifying its parameters, when present. When the insert is deleted, the target solid is healed. Some examples of parametric component-based features are included with BricsCAD. You can use them as samples for creating your own features.

Assigning physical materials to components

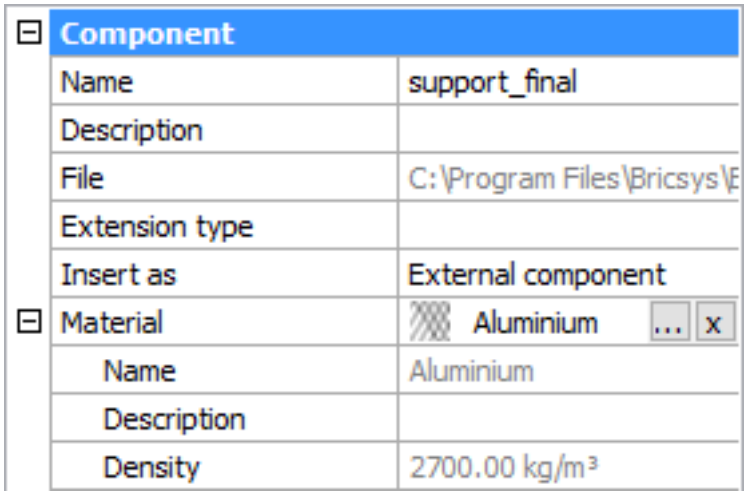
A mechanical component can have aphysical material (2.6.10)assigned.If a mechanical component does not have an assigned material, it will inherit the material from its parental component. If the parental component does not have an assigned material either, the material of the closest component with an assigned material in the hierarchy will be used. As a result the same material can be assigned to all components that have a common parent.

1. Select the root component in the Mechanical Browser (3.2.2).
2. Select the**Material** property and press the **Browse** button (...). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h
3. In the **Physical Materials (6.7.39)** dialog select a material in either the **In Project** or the**In Library** list. Materials which are chosen from the **In Library**are copied to the **In Project** database automatically.
4. Click the **OK** button to assign the selected material to the component.

To remove the material from the component:

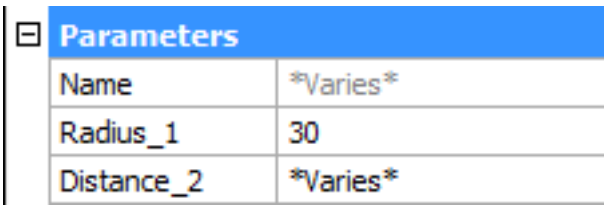
1. Select the root component in the Mechanical Browser (3.2.2).
2. Select the**Material** property and press the **Clear** button (X)

Figure 1.1021



Inserting a component with a physical material When you insert a mechanical component in a model, all its materials are copied to the material library of the model. However, if a material with the same name already exists in the target model, this material will be used instead. As a result the properties of the material in the target model are used, replacing the properties of the material from the inserted component. **Note:** Materials which are copied to the document material library will not be erased in Undo operations.

Figure 1.1022



1.23 Modifying Entities

1.23.1 Modifying Commands Overview

All entity modification tools can be found either on the **Modify** toolbar or in the **Modify** menu.

Icon	Tool name	Command	Description
	Manipulator (1.23.10)	MANIPULATE (6.21.2)	Launches the Manipulator widget; rotates, moves, mirrors and scales entities. Keep the Ctrl-key pressed while starting an action to manipulate a copy of the selected entities.
	Move (1.23.11)	MOVE (6.21.30)	Moves entities to another location in the same drawing or into another drawing.
	Copy (1.23.16)	COPY (6.9.50)	Draws a duplicate or multiple duplicates of the selected entities.
	Offset (1.23.18)	OFFSET (6.23.3)	Creates a parallel or offset copy of curves and lines.
	Scale (1.23.24)	SCALE (6.27.6)	Changes the scale of existing entities, either enlarging them or reducing them proportionately in x, y, and z directions.
	Rotate (1.23.12)	ROTATE (6.26.34)	Rotates entities around a specified point.
	3D Rotate (1.23.12)	ROTATE3D (6.26.35)	Rotates entities around a three-dimensional axis.
	Mirror (1.23.13)	MIRROR (6.21.18)	Moves or copies the reflected image of entities about a line.
	3D Mirror (1.23.13)	MIRROR3D (6.21.19)	Moves or copies the reflected image of entities about a plane.
	Array (1.23.19)	ARRAY (6.4.16)	Creates multiple copies of entities in one of two symmetrical patterns: rectangular (rows and columns) or polar (circular).

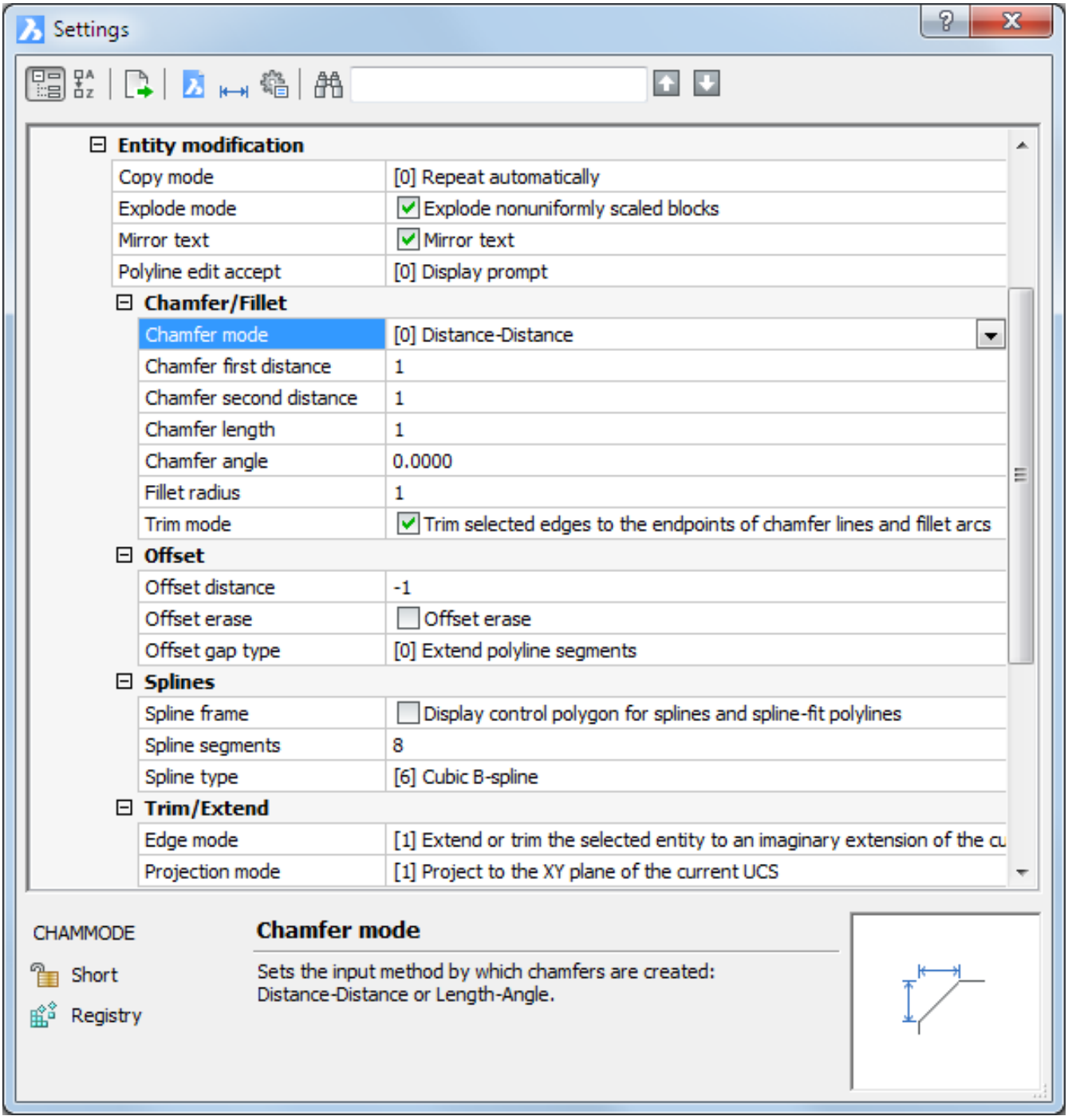
	3D Array (1.23.19)	3DARRAY (6.3.2)	Creates multiple copies of entities in three dimensions. Entities are arrayed in a three-dimensional rectangular (rows, columns and levels) pattern or a two-dimensional polar (circular) pattern in three-dimensional space.
	Break (1.23.26)	BREAK (6.6.18)	Splits an entity into two entities.
	Join (1.23.25)	JOIN (6.18.1)	Joins two entities (two or more lines or two or more arcs) into one entity.
	Trim (1.23.21)	TRIM (6.29.26)	Erases the portions of selected entities that cross a specified boundary.
	Align (1.23.14)	ALIGN (6.4.6)	Aligns selected entities with other entities in three-dimensional space.
	Extend (1.23.20)	EXTEND (6.13.19)	Lengthens lines, arcs, polylines or rays to meet another entity.
	Edit Length (1.23.22)	LENGTHEN (6.20.21)	Changes the length of a line, polyline, freehand entity or arc.
	Stretch (1.23.23)	STRETCH (6.27.49)	Moves a portion of a drawing while retaining connections to other parts of the drawing.
	Chamfer (1.23.28)	CHAMFER (6.9.9)	Creates a chamfer, or a beveled edge, at the intersection of two 3D solids, lines, rays, or infinite lines.
	Fillet (1.23.27)	FILLET (6.14.7)	Creates a fillet, or rounded corner, at the intersection of 3D solids, two lines, rays, or infinite lines.

	Measure (1.23.39)	MEASURE (6.21.13)	Divides a selected entity into segments by placing markers (points or blocks) at specified intervals along its length or circumference.
	Divide (1.23.40)	DIVIDE (6.10.33)	Places markers (points or blocks) along a selected entity. The markers evenly divide the entity into the specified number of equal parts.
	Hatch Edit (1.17.29)	HATCHEDIT (6.16.3)	Edits hatches and gradient fills in a dialog box
	Edit Polyline (1.23.38)	PEDIT (6.24.20)	Edits a two-dimensional or three-dimensional polyline, or a polygon mesh.
	Spline Edit	SPLINEDIT (6.27.40)	Edits a two-dimensional or three-dimensional spline.
	Flatten	FLATTEN (6.14.10)	Projects 2D and 3D entities onto the XY-plane of the current view.
	Explode (1.23.36)	EXPLODE (6.13.13)	Ungroups a block, polyline, polyface mesh, solid, or hatch, creating separate entities for each element.
	Change (1.23.15)	CHANGE (6.9.10)	Changes the location, size, color, elevation, layer, linetype, linetype scale, lineweight, and three-dimensional thickness of entities.
	Properties (1.12.4)	PROPERTIES (6.24.49)	Opens the BricsCAD Properties Bar (if not already open).

1.23.2 Entity Modification Settings

Entity modification settings are found in the **Entity modification** settings group of the **Drawing** settings category in the Settings (1.13.11) dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities

Figure 1.1023



1.23.3 Match Properties

Command: MatchProp (6.21.8) The **Matchprop** command assigns the properties of one entity to one or more other entities (short for "match properties").

To copy properties between entities

1. Do one of the following:

   - Click the **Match** ( ) tool button on the **Standard** toolbar.
   - Type **Matchprop** in the command bar then press Enter.

   You are prompted: Select entity to copy properties from:

2. Select the source entity. **You are promted:**Active Settings: 'Color' 'Layer' 'Linetype' 'Linetype Scale' 'Lineweight' 'Thickness' 'Plot Style' 'Dimension' 'Polyline' 'Material' 'Text' 'Shadow Display' 'Hatch' 'Multileader' Select entities to copy properties to or [Settings]:

3. (option) Type **S** , then press Enter. The **Match Properties** dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/statich The entity properties you are about to copy display in the Match Properties dialog (1). Adjust the settings, then click the **OK** button.

4. Select the target entities. **You are prompted:**Select entities to copy properties to or [Settings]:

5. Do one of the following:

   - (option) Repeat step 3 to adjust the settings, then select more target entities.
   - Right click or press Enter to stop.

Figure 1.1024

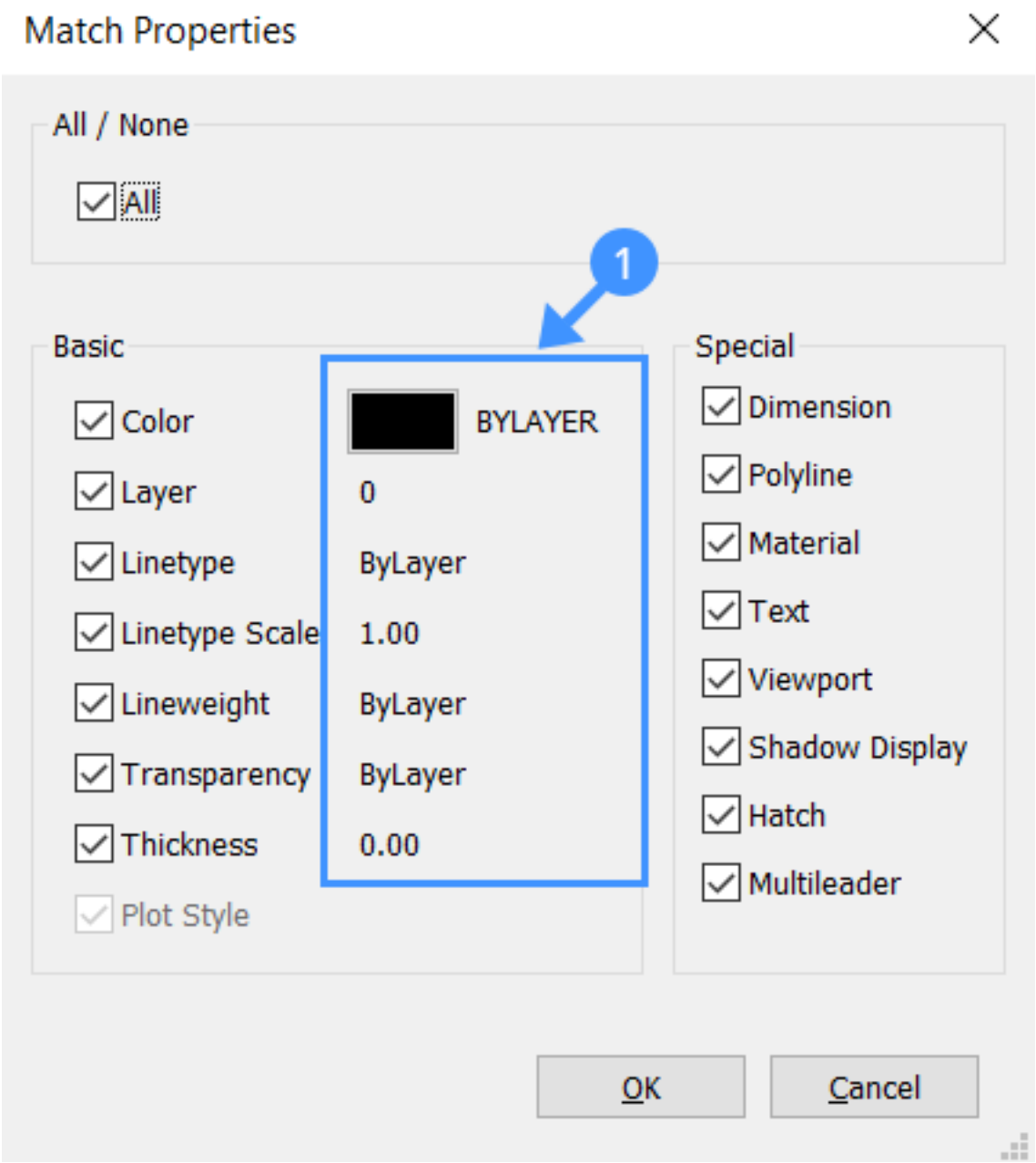


Figure 1.1025

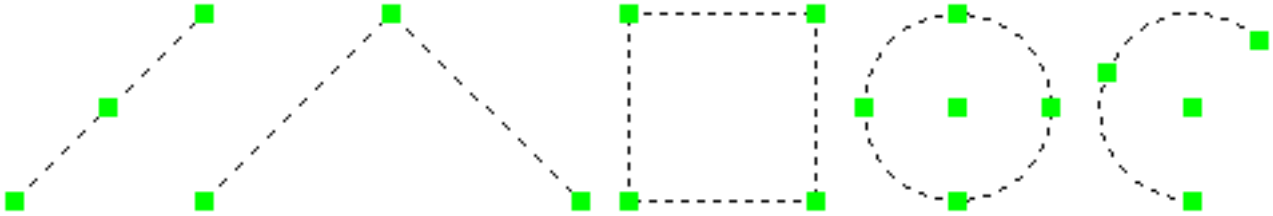
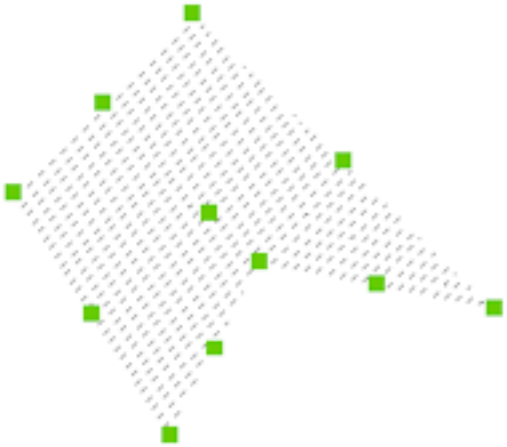


Figure 1.1026



1.23.4 Grips editing

Grips display as small colored squares at key-points on selected entities. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscon
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/10
Whether grips display or not and the appearance of grips is controlled through a series of grips settings (1.23.5). When the display of grips is turned on, grips can be dragged using the mouse in order to stretch (1.23.4), move (1.23.4), mirror (1.23.4), rotate (1.23.4) or scale entities (1.23.4). When the GEOMRELATIONS (1.13.11) system variable is set,the geometric relation (perpendicular or tangent) of the entity being edited with crossing or nearby other lines, polylines, arcs or circles is indicated (see Geometric Relations (1.15.15)). The **Copy** option of the grips editing commands allows to create multiple copies of the entities being edited. To select a grip, move the mouse over it, then click. Selected grips display in a different color, which is defined by the **HOTGRIP** system variable. A selected grip is called a hot grip. By default hot grips display in red. The **midpoint** grip on lines (1.17.9) and infinite (1.17.11) lines, the **startpoint** grip on rays (1.17.10), the **center** grip on circles (1.17.15), arcs (1.17.16), ellipses (1.17.17) and elliptical arcs (1.17.18), the grip on points (1.17.25) and the **insertion point** grip of texts (1.21.20), mtexts (1.21.20), blocks (1.22.5) and Xrefs (6.34.8) can be used to quickly move such entities.

NOTE	When DRAGSNAP (1.15.15) is ON, dragged entities display at the current entity snap location.

Selecting multiple grips

1. Select all entities you want to edit.
2. Press and hold the Shift key, then click the first grip. The selected grip displays in red.
3. Keep the Shift key pressed, then click more grips. All selected grips display in red. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/10HotGrips.png})
4. Release the Shift key, then click the hot grip you want to use as the base point. The hot grips move simultaneously.

Figure 1.1027



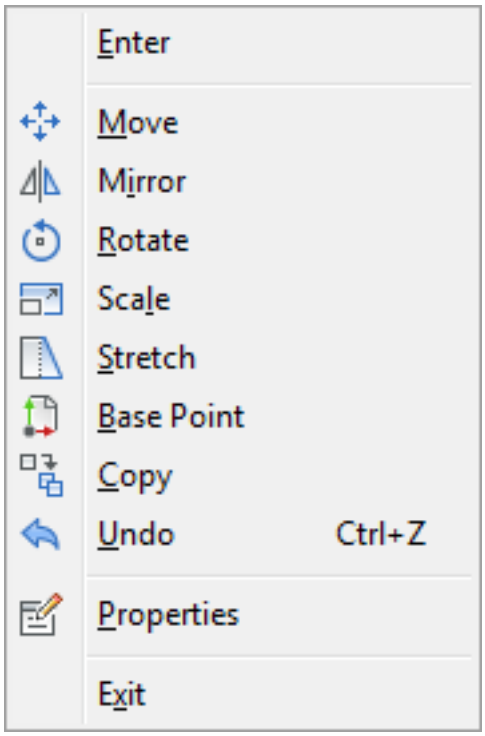
Keyboard	Prompt Box	Description
B	Base Point	Prompts you to specify a new base point, other than the selected grip.
C	Copy	Copies the selected entities and keeps the originals. By default, multiple copies can be created. Press Enter to stop making copies.
U	Undo	If the **Copy** option is chosen, deletes the latest copy.
R	Reference	(Applies to **Grip Rotate** only) Allows to specify a reference angle. Type the angle in the command bar or specify two points.
X	Exit	Aborts the grip edit command.

The grips editing context menu

The grips editing commands can either be chosen by repeatedly hitting the space bar or the Enter key or in a context menu. To display the context menu, do the following:

1. Select the entities you want to manipulate, then drag one of the grips.
2. Right click. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/11Modifyin

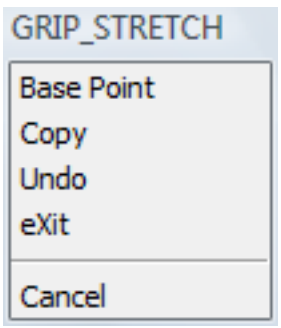
Figure 1.1028



To stretch entities using grips

1. Select the entity or entities.
2. Select the grip or grips (1.23.4) you want to manipulate.
3. Click the grip you want to use as the base point. All selected grips move simultaneously. The **GRIP_STRETCH** prompt box displays: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/1
You are prompted: Enter new point: Base point/Copy/Undo/eXit:
4. Do one of the following to define the displacement point:
   - Click in the drawing.
   - Enter the coordinates (1.15.20), then press Enter.
   - Key-in (1.15.3) the distance, then press Enter. Use ORTHO (1.15.2) or Polar Tracking (1.15.16) to control the angle.

Figure 1.1029

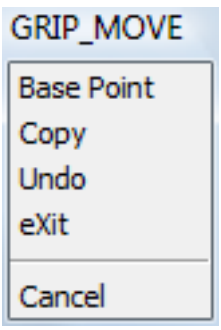


To move entities using grips

1. Select the entity or entities.
2. Select the grip or grips (1.23.4) you want to manipulate.
3. Click the grip you want to use as the base point for the move. The **Grip Stretch** command is launched.
4. Do one of the following:
   - Right click, then select **Move** in the context menu.
   - Hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Move** is active.

   The **GRIP_MOVE** prompt box displays: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/
   You are prompted: Enter move point: Base point/Copy/Undo/eXit:
5. Do one of the following to define the displacement point:
   - Click in the drawing.
   - Enter thecoordinates (1.15.20), then press Enter.
   - Key-in (1.15.3) the distance, then press Enter. UseORTHO (1.15.2)orPolar Tracking (1.15.16)to control the angle.

Figure 1.1030

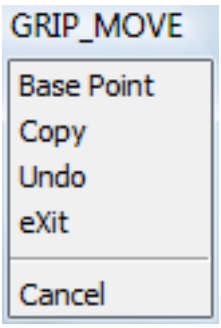


To copy entities using grips

1. Select the entity or entities.
2. Select the grip or grips (1.23.4) you want to manipulate.
3. Click the grip you want to use as the base point for the move. The **Grip Stretch** command is launched.
4. Do one of the following:
   - Right click, then select **Move** in the context menu.
   - Hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Move** is active.

   The **GRIP_MOVE** prompt box displays: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/
   You are prompted: Enter move point: Base point/Copy/Undo/eXit:
5. Do one of the following
   - Choose **Copy** in the context menu.
   - Type **C** , then hit the **Space bar** or **Enter** key.
6. Do one of the following to define the displacement point:
   - Click in the drawing.
   - Enter thecoordinates (1.15.20), then press Enter.
   - Key-in (1.15.3) the distance, then press Enter. UseORTHO (1.15.2)orPolar Tracking (1.15.16)to control the angle.
7. Repeat the previous step to create more copies or press the Space bar or Enter key to stop.

Figure 1.1031

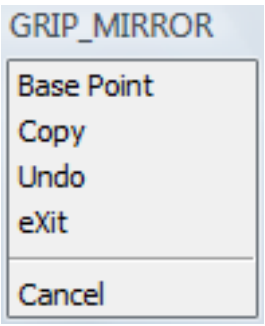


To mirror entities using grips

1. Select the entity or entities.
2. Select the grip or grips (1.23.4) you want to manipulate.
3. Click the grip you want to use as the base point for the mirroring. The **Grip Stretch** command is launched.
4. Do one of the following:
   - Right click, then select **Mirror** in the context menu.
   - Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Mirror** is active.

   The **GRIP_MIRROR** prompt box displays. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V1
   You are prompted: Enter second point: Base point/Copy/Undo/eXit.
5. Specify a second point to define the mirror axis. The selection is mirrored.

Figure 1.1032

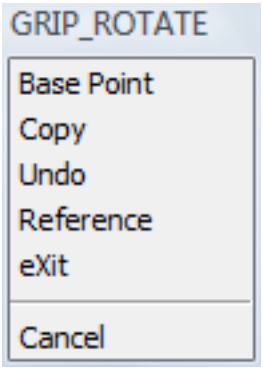


To rotate entities using grips

1. Select the entity or entities.
2. Select the grip or grips (1.23.4) you want to manipulate.
3. Click the grip you want to use as the base point for the rotation. The **Grip Stretch** command is launched.
4. Do one of the following:
   - Right click, then select **Rotate** in the context menu.
   - Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Rotate** is active.

   The **GRIP_ROTATE** prompt box displays. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V1
   You are prompted: Enter rotation angle: Base point/Copy/Undo/eXit.
5. Do one of the following to rotate the selection:
   - Specify a point to define the rotation angle.
   - Type the rotation angle in the command bar, then press Enter

Figure 1.1033



To scale entities using grips

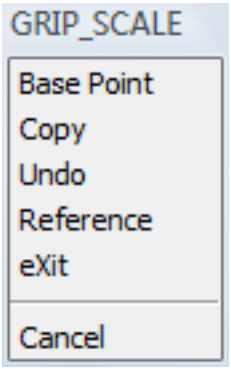
1. Select the entity or entities.
2. Select the grip or grips (1.23.4) you want to manipulate.
3. Click the grip you want to use as the base point for the scaling. The **Grip Stretch** command is launched.
4. Do one of the following:
   - Right click, then select **Scale** in the context menu.
   - Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Scale** is active.

   The **GRIP_SCALE** prompt box displays. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18

   You are prompted: Enter scale factor: Base point/Copy/Undo/eXit.
5. Type the scale factor in the command bar, then press Enter to scale the selection.

Tutorial: Grips Editing in BricsCAD Click here (1.4.6) to watch

Figure 1.1034



1.23.5 Selection and grip settings

When you select entities in a drawing, the selection method and the display of selected entities, e.g. whether grips display or not, is controlled by a number of settings.

Selection settings

Selection settings sit in the **Selection** settings group of the **Program Options** settings category in the Settings (1.13.11)dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/

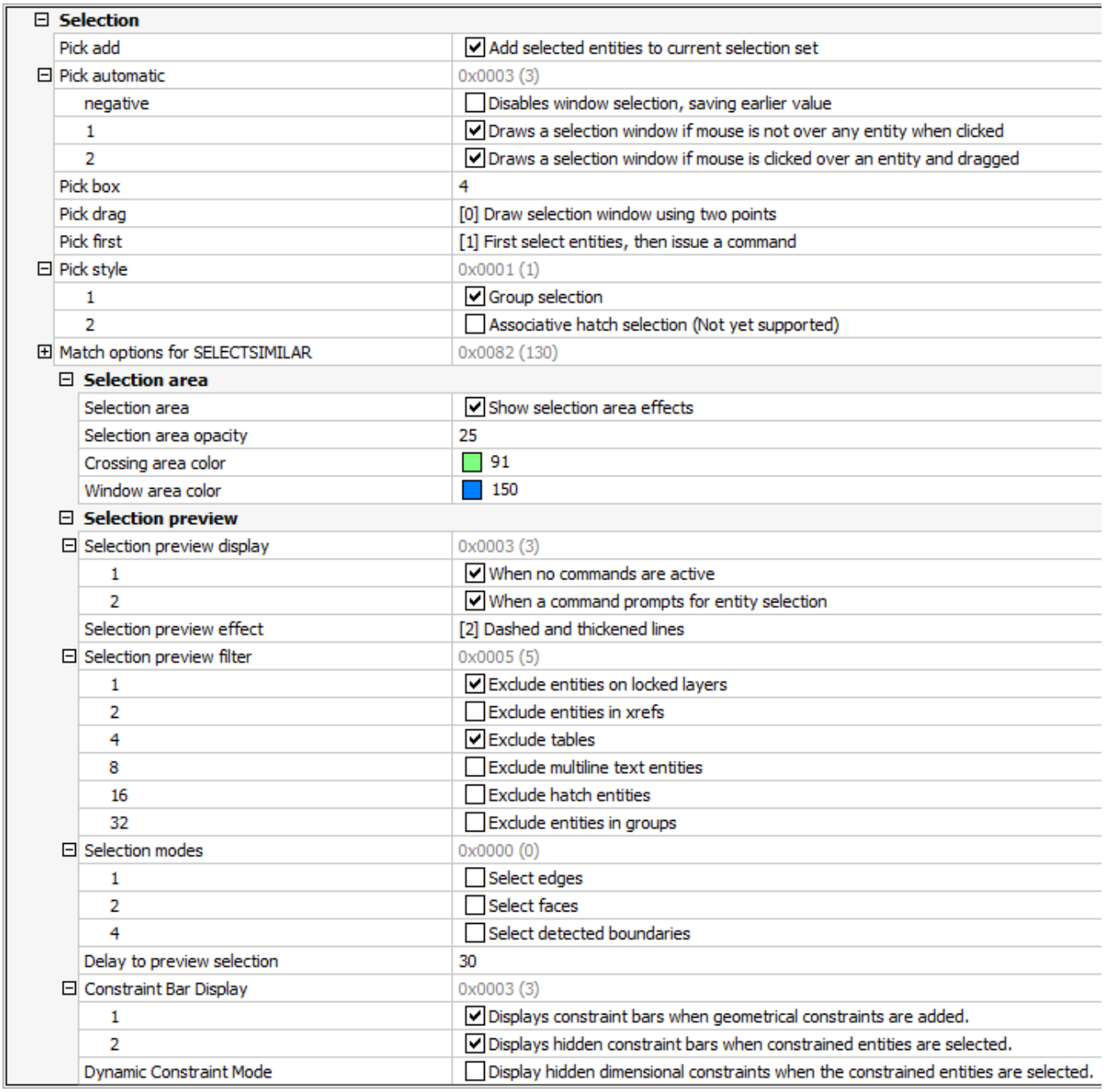
Name	System Variable	Description
Highlight	HIGHLIGHT	Determines wether or not entities highlight on screen when selected.
Image highlight	IMAGEHLT	Determines whether the entire raster image highlights or only the raster image frame when selected.
Pick add	PICKADD	Controls whether subsequent selections replace the current selection set or add to it. **ON** : The selection set is extended if you select additional entities. Press and hold the Shift key to remove entities from the selection set. **OFF** : You cannot add entities to a selection set. The newly selected entity or entities replace the existing selection set. However, if you press and hold the **Shift** key, you can add entities. If you select an entity that was already selected while pressing the **Shift** key, this entity is removed from the selection set.
Pick automatic	PICKAUTO	Controls automatic window selection (inside or crossing) when selecting entities. The value is stored as a bitcode using the sum of the selected options. • negative: Disables window selection, saving current value. • 1 = draws a selection window if the mouse is over a blank area of the drawing when clicked. • 2 = draws a selection window if the mouse is over an entity when dragged.
Pick box	PICKBOX	Defines the size of the small square at the end of the selection cursor ( ) . If you select an entity by clicking the **Pick Box** must touch or overlap the entity. The default size of the **Pick Box** is 3.

Pick drag	PICKDRAG	Controls the method of drawing a selection window. **ON** : Allows to define a selection window by dragging:press and hold the left mouse button to define the first corner of the rectangle, then move the mouse to define the size of the selection window and release the mouse button to define the opposite corner. **OFF** : Define the selection window by clicking two opposite corners.
Pick first	PICKFIRST	Controls whether you select objects before or after you issue a modification command. **ON** : Allows to first compose a selection set, then launch a modification command. **OFF** : You must first start the command, then compose the select entities.
Pick style	PICKSTYLE	Controls the selection of groups and associative hatches. • 0 = No group selection, nor associative hatch selection. • 1 = Group selection: if a member of a selectable group is selected, all members of the group are selected. • 2 = Associative hatch selection: the hatch and its boundary are selected, no matter what is picked: the hatch or the boundary (not supported yet). • 3 = Group selection and Associative hatch selection.
Match options for SELECTSIMILAR	SELECTSIMILARMODE	Controls which properties must match for an entity of the same type to be selected with the SELECTSIMILAR (6.27.21) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Selection area	SELECTIONAREA	Controls the display of selection area effects.
Selection area opacity	SELECTIONAREAOPACITY	Controls the transparency of the selection area: the higher the value the more opaque the selection area.
Crossing area color	CROSSINGAREACOLOR	Specifies the color for crossing selection areas.
Window area color	WINDOWAREACOLOR	Specifies the color for window selection areas.

Selection preview display	SELECTIONPREVIEW	Controls when entities highlight when the cursor hovers over them: • 0 = never; • 1 = when no commands are active; • 2 = when a command prompts for entity selection; • 3 = both, when no commands are active and when a command prompts for entity selection. When the QUAD (1.12.5) cursor menu is active, SELECTIONPREVIEW is neglected, unless a command prompts for entity selection.
Selection preview effect	PREVIEWEFFECT	Controls the appearance of highlighted entities: 0=dashed lines; 1=thickened lines; 2=dashed and thickened lines (not yet supported).
Selection preview filter	PREVIEWFILTER	Controls which entity types are excluded from selection previewing. The options are: • Entities on locked layers • Entites in Xrefs • Tables • Mtext entities • Hatches • Groups
GL Selection Highlight Style	HIGHLIGHTEFFECT	Specifies the highlight method: • 0=dashed lines • 1=use different color • 2=use thickened line • 3=use different color and thickened line.
Selection Highlight color	HIGHLIGHTCOLOR	Specifies the highlight color to be used when HIGHLIGHTEFFECT is 1 or 3.

Selection Modes	SELECTIONMODES	Defines which subentities should highlight in selection preview. The variable is stored as a bitcode using the sum of the values of all selected options. Click the buttons on the **Selection Modes** toolbar to set the value of SELECTIONMODES. A pressed button indicates the option is selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 • 1 = Enable detection of 3D solid edges ( ) • 2 = Enable detection of 3D solid faces ( )* • 4 = Enable boundary detection ( ). Controls whether closed boundaries in XY-plane of the current coordinate system or on the face of 3D solids highlight when no commands are active. The BOUNDARYCOLOR system variable specifies the color for highlighting detected boundaries. * Hold down the Ctrl-key to invert the status of the **Select Faces** option: if ON allows to select 3D solids, if OFF allows to select solid faces. Repeatedly pressing the TAB key, while keeping the cursor at the same position, highlights all included subentity types one by one. See also window selection (1.23.6).
Constraint Bar Display	CONSTRAINTBARDISPLAY	Controls the visibility of a geometric constraint at its creation or selection.
Dynamic Constraint Mode	DYNCONSTRAINTMODE	Controls the visibility of hidden dimensional constraints when the constrained entity is selected. When the cursor is over an entity with a dimensional constraint, the blue constraint glyph displays.
	User Preference	
Preview selection delay	PreviewDelay	Specifies how many milliseconds to wait before highlighting the (sub)entities under the cursor.

Figure 1.1035



Selection	
Pick add	☑ Add selected entities to current selection set
Pick automatic	0x0003 (3)
negative	☐ Disables window selection, saving earlier value
1	☑ Draws a selection window if mouse is not over any entity when clicked
2	☑ Draws a selection window if mouse is clicked over an entity and dragged
Pick box	4
Pick drag	[0] Draw selection window using two points
Pick first	[1] First select entities, then issue a command
Pick style	0x0001 (1)
1	☑ Group selection
2	☐ Associative hatch selection (Not yet supported)
Match options for SELECTSIMILAR	0x0082 (130)
Selection area	
Selection area	☑ Show selection area effects
Selection area opacity	25
Crossing area color	91
Window area color	150
Selection preview	
Selection preview display	0x0003 (3)
1	☑ When no commands are active
2	☑ When a command prompts for entity selection
Selection preview effect	[2] Dashed and thickened lines
Selection preview filter	0x0005 (5)
1	☑ Exclude entities on locked layers
2	☐ Exclude entities in xrefs
4	☑ Exclude tables
8	☐ Exclude multiline text entities
16	☐ Exclude hatch entities
32	☐ Exclude entities in groups
Selection modes	0x0000 (0)
1	☐ Select edges
2	☐ Select faces
4	☐ Select detected boundaries
Delay to preview selection	30
Constraint Bar Display	0x0003 (3)
1	☑ Displays constraint bars when geometrical constraints are added.
2	☑ Displays hidden constraint bars when constrained entities are selected.
Dynamic Constraint Mode	☐ Display hidden dimensional constraints when the constrained entities are selected.

Grips settings

If the **Grips** system variable is different from zero, grips display on selected entities. **Grips** settings sit under **Display/Viewing** in the **Drafting** settings group of the **Drawing** settings category in the Settings (1.13.11) dialog. Grips can be used to manipulate entities (see Grips Editing (1.23.4)) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU

Name	System Variable	Description
Grips	GRIPS	The value of GRIPS can be 0, 1 or 2 • 0 : No grips. • 1 : Display grips. Polyline endpoint and midpoint grips. • 2 : Display grips. No midpoint grips on polyline segments.
Grips in blocks	GRIPBLOCK	Toggles the display of grips on entities in blocks on/off.
Grip color	GRIPCOLOR	Sets the color of unselected grips. Default value is green (index color no. 72).
Selected grip color	GRIPHOT	Sets the color of selected grips, so called 'hot grips'. Default value is red (index color no. 240).
Hover grip color	GRIPHOVER	Sets the color of an unselected grip when the cursor pauses over it. Default value is blue (index color no. 150).
Grip size	GRIPSIZE	Sets the size of the grip box, expressed in pixels. Default value = 4.
Grip object limit	GRIPOBJLIMIT	Suppresses the display of grips if the number of selected objects exceeds the value of GRIPOBJLIMIT. If set to 0 (zero) grips always display.
Grip tips	GRIPTIPS	(Not supported yet) Determines whether grip tips display when the cursor hovers hover grips on custom entities or dynamic blocks that support grip tips.
Grips attraction	EnableAttraction (*)	Toggles the magnetism of grips on/off.
Grips attraction distance	AttractionDistance (*)	Sets the grips attraction distance: if the cursor is within this distance the cursor is attracted by the grip and jumps to it.

(*) User Preference

Tutorial: Dynamic Grips and Tips Click here (1.4.5) to watch

Figure 1.1036

Grips	
Grips	[1] Turn on endpoint and midpoint grips
Grips in blocks	☐ Assigns grips to entities within the block
Grip color	72
Dynamic grip color	140
Selected grip color	240
Hover grip color	150
Grip size	4
Grip object limit	100
Grip tips	☑ Display grip tips
Grips attraction	☑ Enable grips attraction
Grips attraction distance	4

Figure 1.1037

Match options for SELECTSIMILAR	0x0082 (130)
1	☐ Color
2	☑ Layer
4	☐ Linetype
8	☐ Linetype scale
16	☐ Lineweight
32	☐ Plot style
64	☐ Object style
128	☑ Name

Figure 1.1038

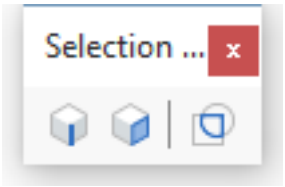


Figure 1.1039

Selection modes	0x0007 (7)
1	☑ Select edges
2	☑ Select faces
4	☑ Select detected boundaries

1.23.6 Selection Methods

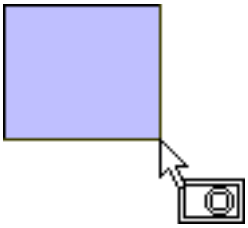
https://www.youtube-nocookie.com/embed/0RgInrXDIVU Before you start to use modification commands, you need to know how to compose a selection set. If the **PICKFIRST** variable is **ON** you can build the selection set before starting the command, else you are prompted to select the entities after you launched the modification command. Using QUICK SELECT (1.23.7) you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set. The SELECTIONPREVIEW (1.23.5) and PREVIEWEFFECT (1.23.5) system variables facilitate the selection procedure.

Composing a selection set before (pre-pick)

To compose a selection set before launching a modification command do one or more of the following:

- Click an entity. The entity highlights and grips show.

NOTE	If the Quad cursor menu (1.12.5) is active you must press and hold the Ctrl key while selecting entities. Click again to unselect.

- Window selection:
  - Click to define the first corner of a selection window. Move the mouse to the left to define a **Crossing Window**:  All entities which overlap the window or are completely inside the window are added to the selection set. The selection window displays in dashed lines with a green background.
  - Click to define the first corner of a selection window. Move the mouse to the right to define a **Window-Inside**:  All entities which are completely inside the window are added to the selection set. The selection window displays in continuous lines with a blue background.
  - Pressing the Ctrl key during window selection allows to select faces or edges of 3D solids and surfaces. The icon next to the cursor indicates the selection mode:  : Solids (Default)  : Faces  : Edges

  Depending on the value of the TIPS (1.13.11) system variable, the Hotkey Assistant widget (1.12.2) displays to indicate the current selection mode: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sou

- Press an hold the **Shift** key, then use one of the above methods to select entities that you want to **remove** from the selection set.

NOTE	If the **PICKADD** system variable is **OFF** , you cannot add entities to a selection set.

Figure 1.1040



Composing a selection set afterwards (post-pick)

Command: SELECT (6.27.17) If you launch a modification command when no selection set is active, **you are prompted** to select entities. BricsCAD provides a range of selection methods to let you compose your selection set easily. The various selection methods are:

- **Add to set**: Selected entities are added to the selection set.
- **Subtract from set**: Selected entities are removed from the selection set.
- **Picking**: Place the pickbox over a part of the object and click.
- **Select all entities**: Select all entities in the entire drawing, including entities which are on hidden layers. Entities on frozen layers are not selected.
- **Previous selection**: Re-use the previous selection set.
- **Last entity in drawing**: Select the most recent entity.
- **Window**: All entities which are inside, overlap or are completely outside a window are selected. You can choose between a **rectangle**, **polygon** or a **circle**.
- **Fence**: All entities that cross a multi-segment line are selected.
- **Point**: Click a point to select all closed entities which enclose this point.
- **Select by Properties**: Displays the **Select by Properties** context menu to select entities by **Color**, **Layer**, **Linetype**, **Name**, **Thickness**, **Type**, **Value**, **Width** or **Handle**.
- **Undo**: Undoes the last selection action.
- **Location**: Displays the **Select Location** context menu.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sc448/i1Modifyat...} (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sc448/i1ModifyatEntities/ART/01SelectUS/V18/UCo1tx.png}) Select Location and Select By Properties context menus

Selecting overlapping entities

When entities are close together or lie on top of each other it is difficult or sometimes impossible to select the entities needed. Cycling through the overlapping entities, highlighting each entity one by one, helps you to select such entities. Make sure SELECTIONPREVIEW (1.23.5) is ON and/or the Quad cursor menu (1.12.5) is active.

1. Place the cursor over the overlapping entities. One of the entities highlights.
2. Repeatedly hit the TAB key. The overlapping entities highlight one by one.
3. Click when the desired entity highlights.
4. Repeat the previous steps to select more entities.

NOTES

- Do not move the mouse while hitting the TAB key, because this will restart the highlight order.
- Hold down the Shift key to cycle the overlapping entities in reverse order.

Tutorial: Those Five Other Selection Modes in BricsCAD Click here (1.2.7) to watch

Figure 1.1041

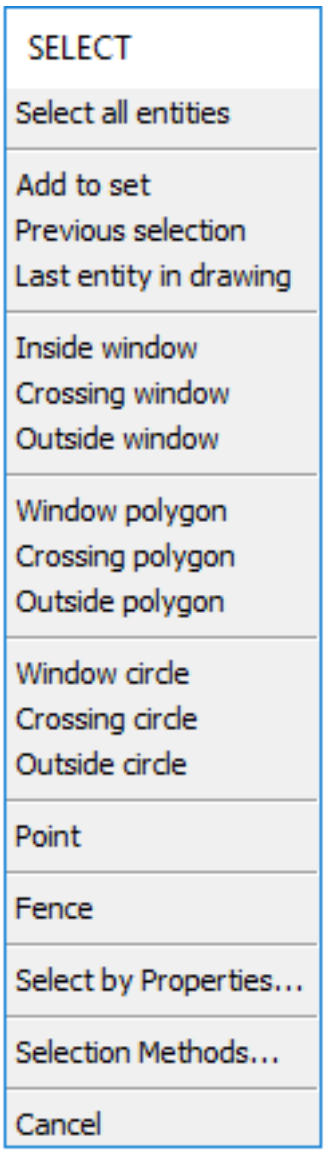
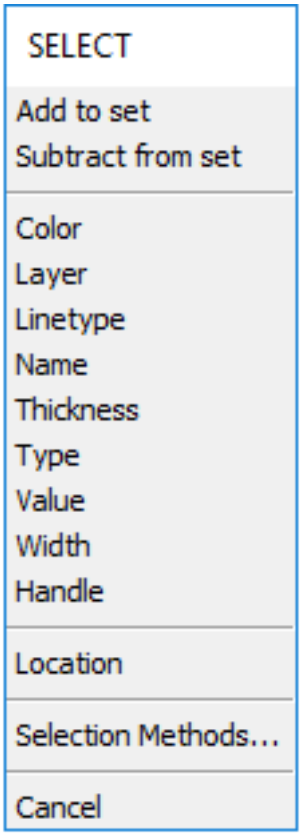


Figure 1.1042



1.23.7 Quick Select

Use smart filters to compose a selection set Using **Quick Select** you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set. After composing the filter criteria you can choose to add the matching entities to a new selection set, add the entities to the current selection set or remove them from the current selection. **Quick Select** applies to the entire drawing or to a previously created selection.

Tutorial: Use Quick Select Click here (1.4.7) to watch

To create a selection set

1. Click the **Quick Select** tool button ( ) on the Properties bar (1.12.4). The **Properties bar** mode is set to **Quick Select** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/AR The **Apply to current selection set** option is checked if a selection set exists. (\vref {fig:C:/Dev/Task-2448/images/staticb

2. Click the **Selection List** button, to see a list of all entity types in the current drawing or current selection set (see previous step). A list of all entities in the drawing and the number of entities of each type displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/01Propertie

3. Select an entity type in the list, e.g. **Line** . Line properties show in the **Properties** Bar. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/01PropertiesBarQSelectLine

4. Specify the property filters, e.g. **Color** , **Layer** and **Linetype** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

5. (option) Specify a different selection operator for each entity property: Click the = sign, then select an operator in the list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/A

6. Do one of the following:

   - Click the **Add to new selection set** button ( ). The entities that match the selected criteria are selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/0
   - Click the **Add to the current selection set** button ( ). The entities that match the selected criteria are added to the current selection.
   - Click the **Remove from current selection** set button ( ). The entities that match the selected criteria are removed from the current selection.

7. (option) Repeat the above procedure to further refine the selection set.

Figure 1.1043

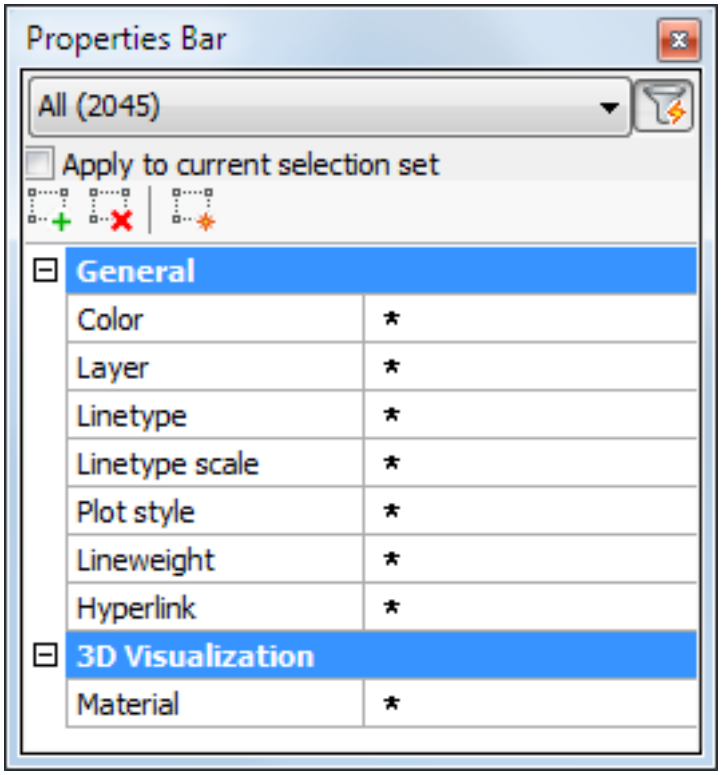


Figure 1.1044

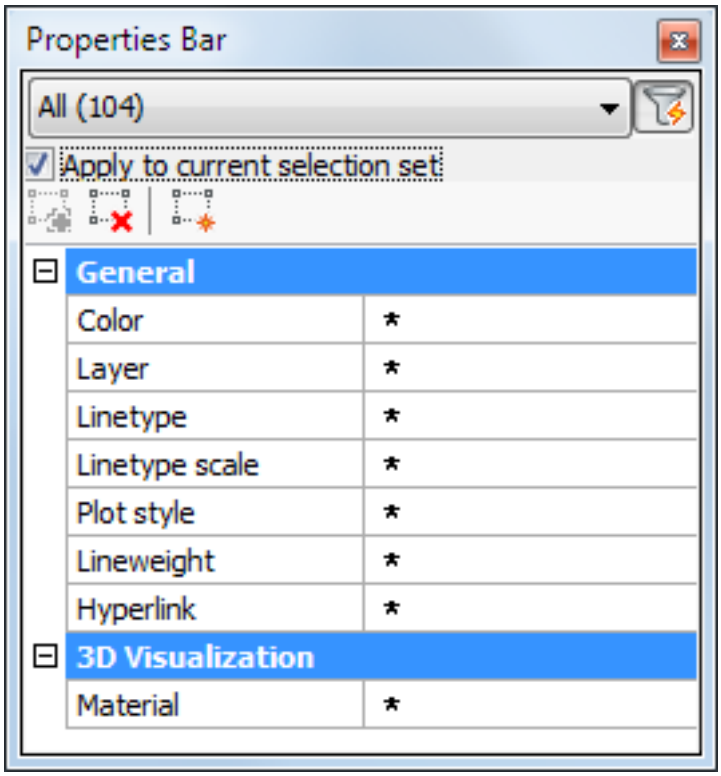


Figure 1.1045

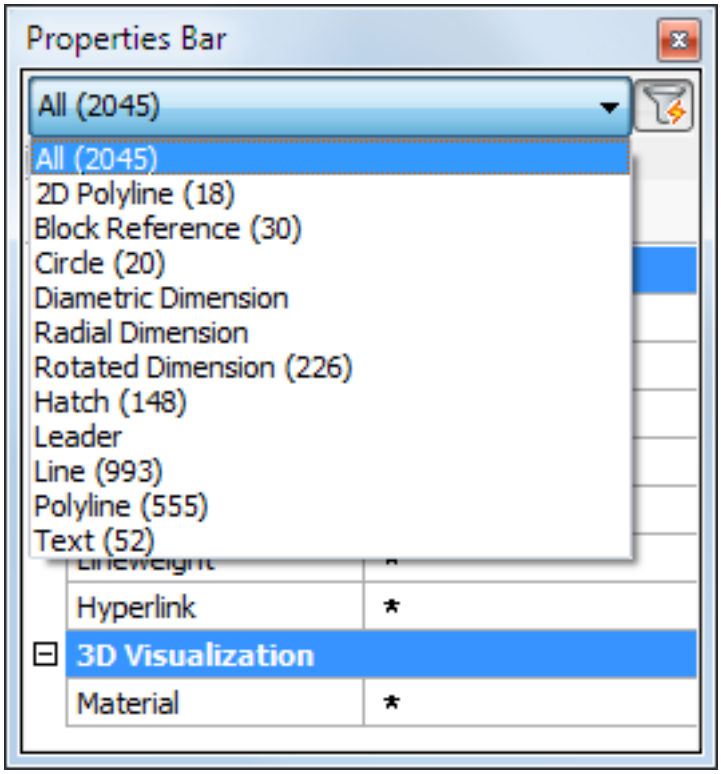


Figure 1.1046

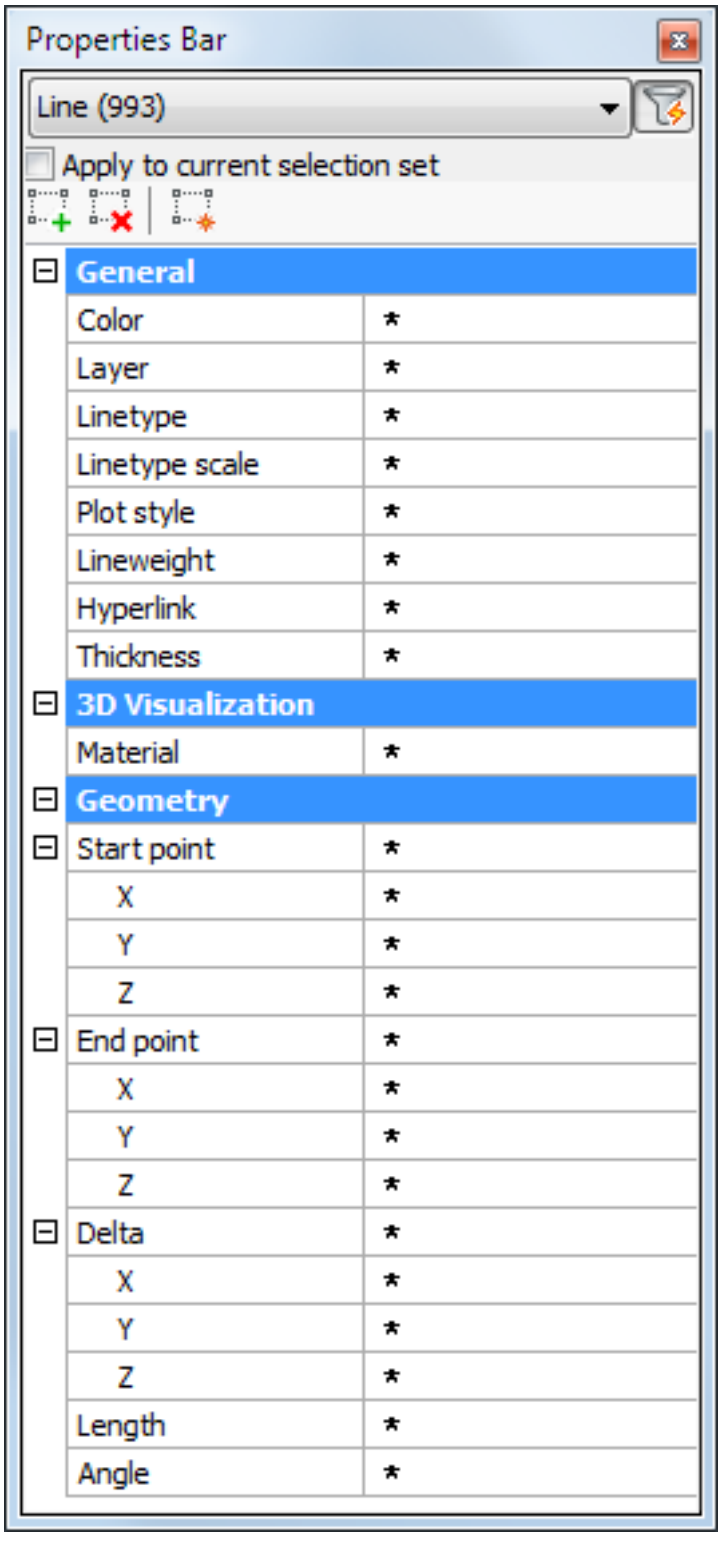


Figure 1.1047

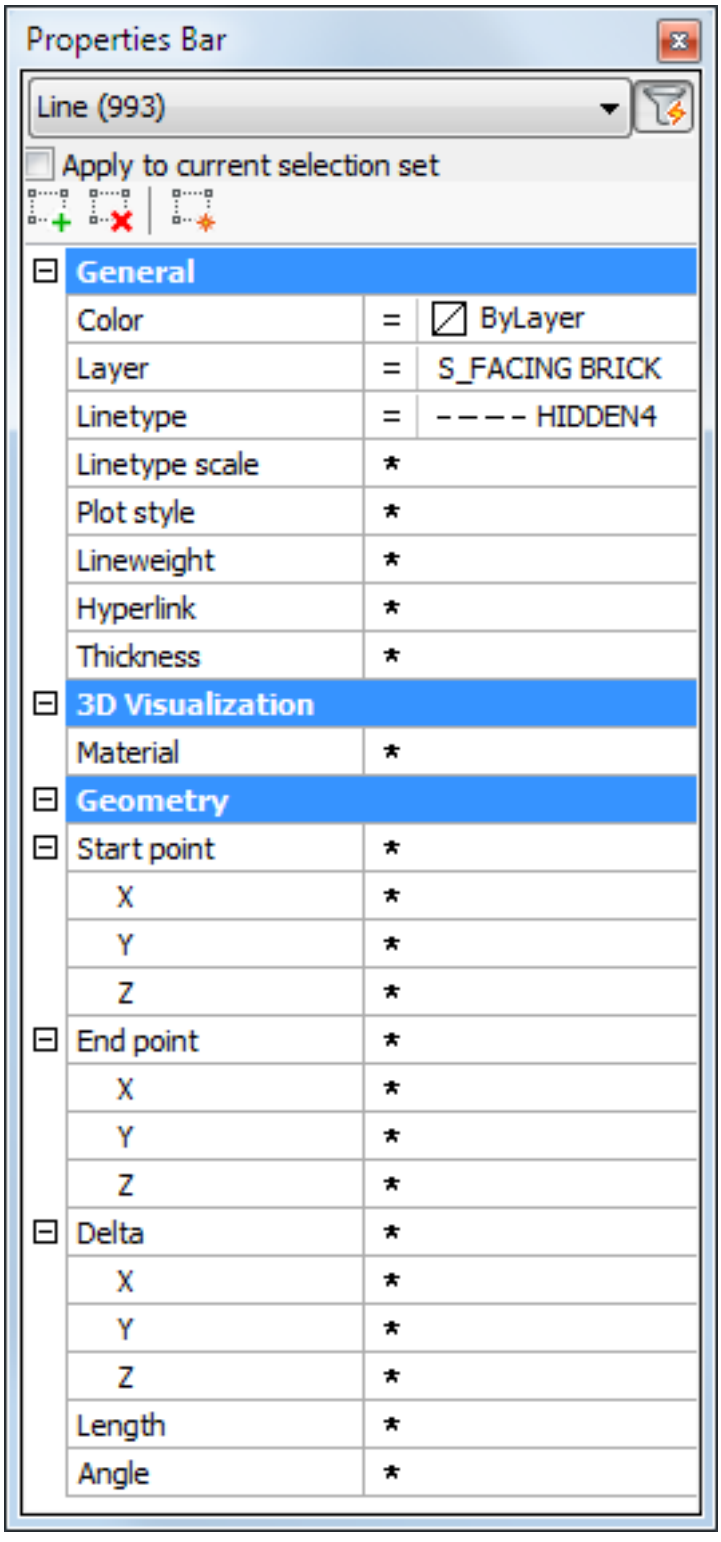


Figure 1.1048

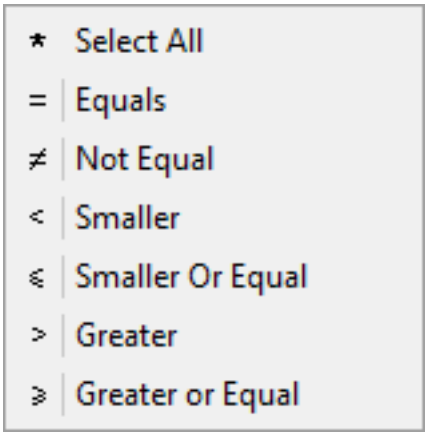
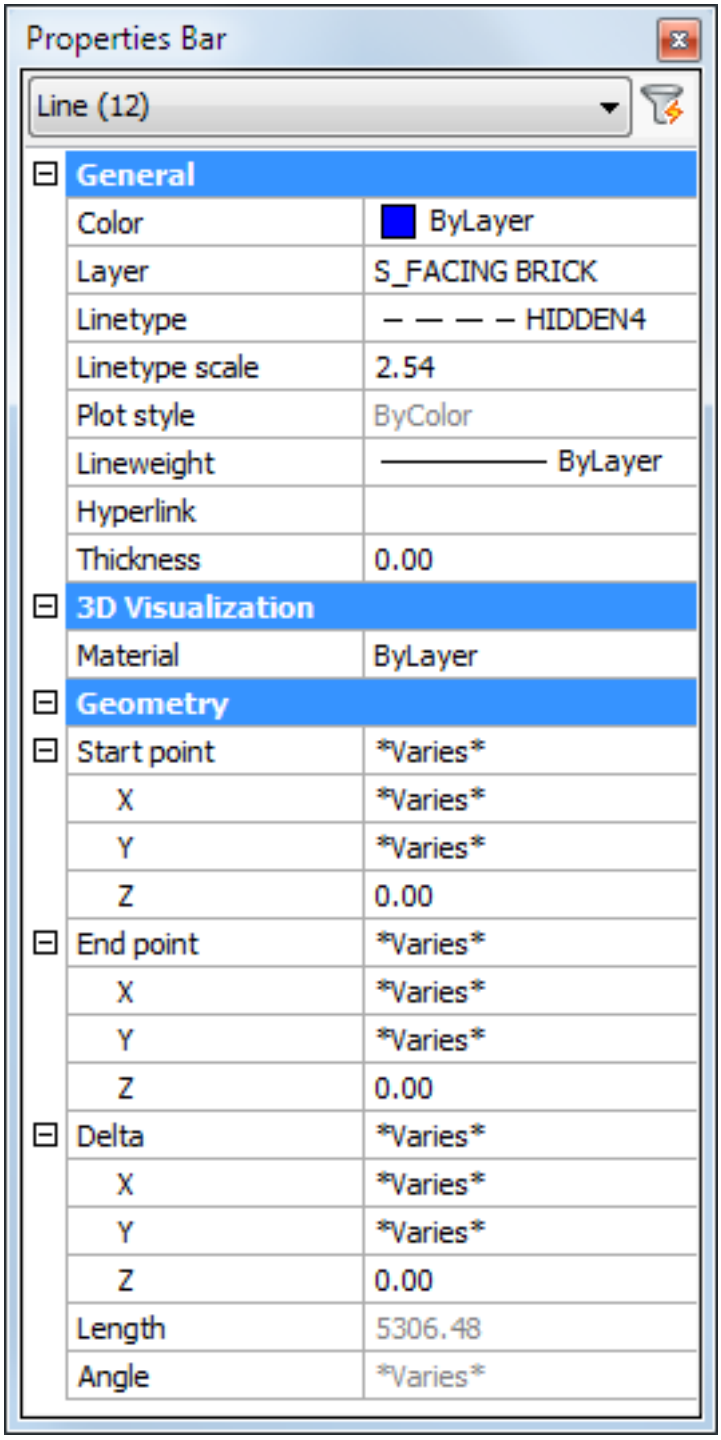


Figure 1.1049



1.23.8 The Structure Panel

The dockable (1.12.7)**Structure** panel displays a structured tree view of the drawing content. This structured view is fully configurable and configurations can easily be saved and recalled. The *.cst configuration files are stored in the Support folder (1.13.2) by default. Entities selected in the structure tree can optionally be simultaneously highlighted and selected in the drawing, and vice versa, when selecting entities in the drawing they optionally can become selected and highlighted in the structure tree.

To open the Structure panel

1. To open the **Structure** panel do one of the following:
   - Place the cursor over a toolbar or ribbon panel, then right click and choose **Structure** in the context menu.
   - Type **STRUCTUREPANEL**in the command bar, then press Enter.
2. Click the down-arrow button and load a configuration file in the list; e.g. **bim**. (\vref {fig:C:/Dev/Task-2448/images/staticb
   The BIM model is organized in a tree-structure: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/U
3. Do the following:
   - Click the expand/collapse icon of a node to expand or collapse the node.
   - Right click a node then choose **Expand All** or **Collaspe All** in the context menu to expand or collapse all branches. If you do this on the drawing name the whole tree structure expands or collapses.
   - Select an entity in the tree. Depending on the selected option in the current configuration file, entities are either selected or highlighted in the model. e.g. if you click a story, all entities on that story are selected. You can then use IsolateObjects (6.17.21) ( ) or HideObjects (6.16.11) ( ) to isolate or hide the selected story in the model.
   - Right click an entity, then choose **Zoom** in the context menu, to zoom onto the entities.
   - Select an entity in the model: the entity is selected in the tree.

Figure 1.1050

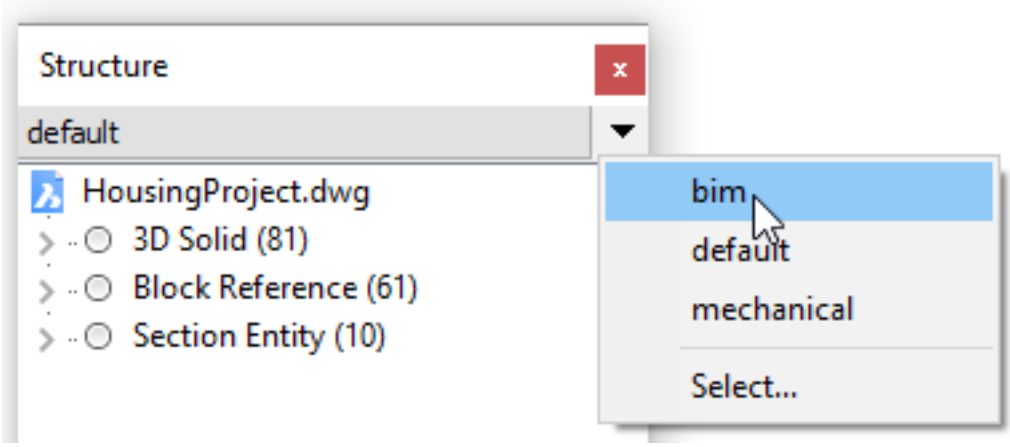
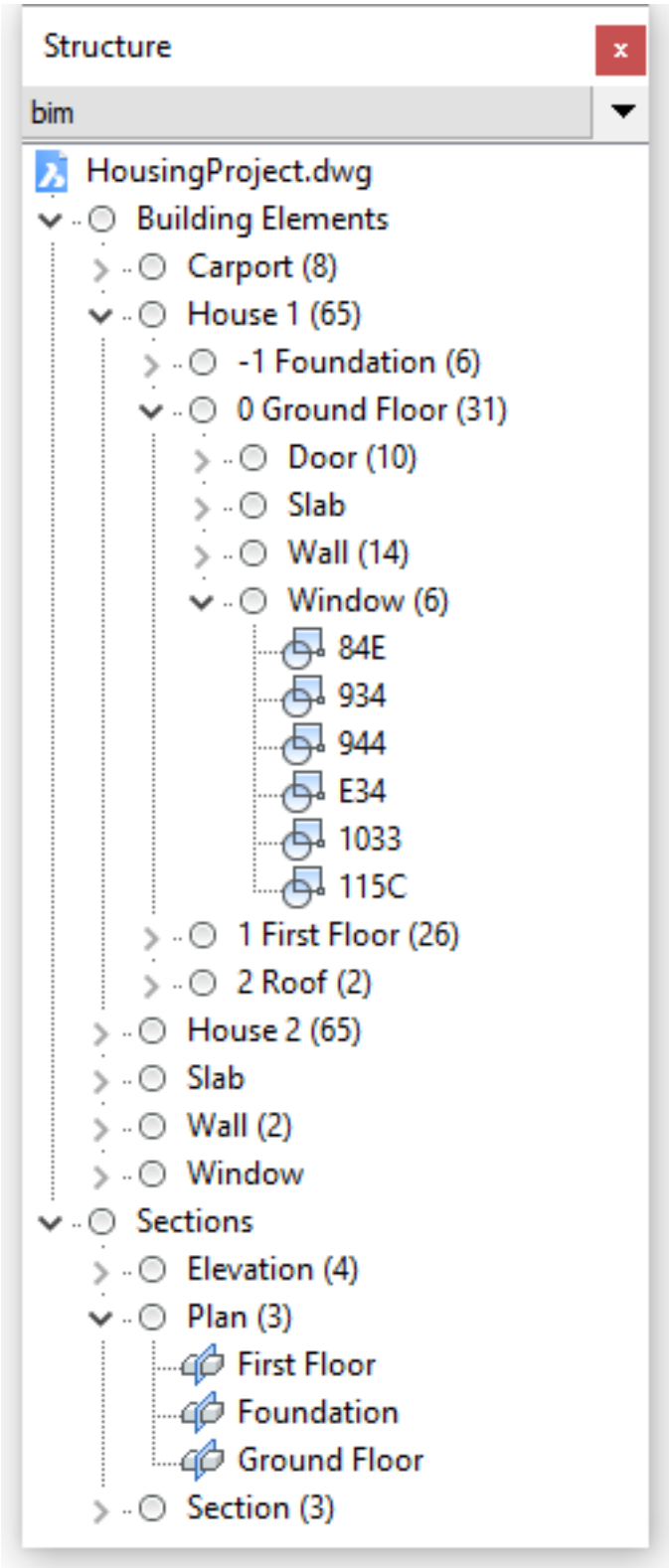


Figure 1.1051



To configure the tree-structure on the Structure panel

The tree structure on the **Structure** panel is a hierarchical set of rules. Each rule has a **grouping property** and a **sorting property**. The structure configurations are saved to *.cst files. By default *.cst files are saved in the **Support** subfolder of theRoamable Root (1.13.2) folder (e.g. **C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V17x64\en_US**). The name of the *. cst configuration file is saved in the drawing through the StructureTreeConfig (1.13.11) user preference. BricsCAD searches for *.cst configuration files in the folders defined by the SRCHPATH (1.13.11) configuration variable.

1. Right click a node, then choose **Configure** in the context menu. The **Configure Structure Tree** dialog box opens: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ARTGroupSortDialog.png})
2. To add a rule, do the following:
   - Select an existing rule, then click the **Add** icon (+) or right click and choose**Add Rule** in the context menu. The new rule is added below the selected rule.
   - Select the newly created group, then click the name or right click and choose **Rename** in the context menu and type a new name for the rule.
   - Select the **Group** node of the rule, then click the **Add** icon (+) or right click and choose **Add Grouping Property** in the context menu. The **Select Property** dialog box displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/01ConfigureStructureTrSelectPropertyDialog.png})
   - Select a property, then double click or click the **OK** button. Optionally, type a search string in the **Search** field, then click the **Up** (↑) or **Down** (↓) arrow buttons to look up the properties containing the search string.
   - Repeat the previous step to add more grouping properties.
   - Select the Sort node, then click the **Add** icon (+) or right click and choose **Add sorting property** in the context menu. The **Select Property** dialog box displays.
   - Select a property, then double click or click the **OK** button. Optionally, type a search string in the **Search** field, then click the **Up** (↑) or **Down** (↓) arrow buttons to look up the properties containing the search string.
   - Repeat the previous step to add more grouping properties.
3. Click the **Show/Skip** tab: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModShowSkipDialog.png}) Select an option and the select the entity types to show or to exclude.
4. Click the **Options** tab: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyiOptionsDialog.png})
   - Choose what happens in the model when entities are selected in the structure tree: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/01ConfigureStructureTrOptionsModelSelection.png})
   - Select what happens in the structure tree when an entity is selected/deselected in the model: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/01ConOptionsTreeSelection.png})
5. (option) To move an item in the tree: select the item, then click the **Move Up** (↑) or **Move Down** (↓) button.
6. Do one of the following:
   - Click the **OK** button to save the configuration and close the dialog box.
   - Click the **File** menu on the dialog box and choose an option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/FileOptions.png}) Open...: opens a file dialog to select another *.cst configuration file. Save : save the current configuration and keep the file dialog open. Save As... : save the current configuration under a different name and keep the file dialog open.

Figure 1.1052

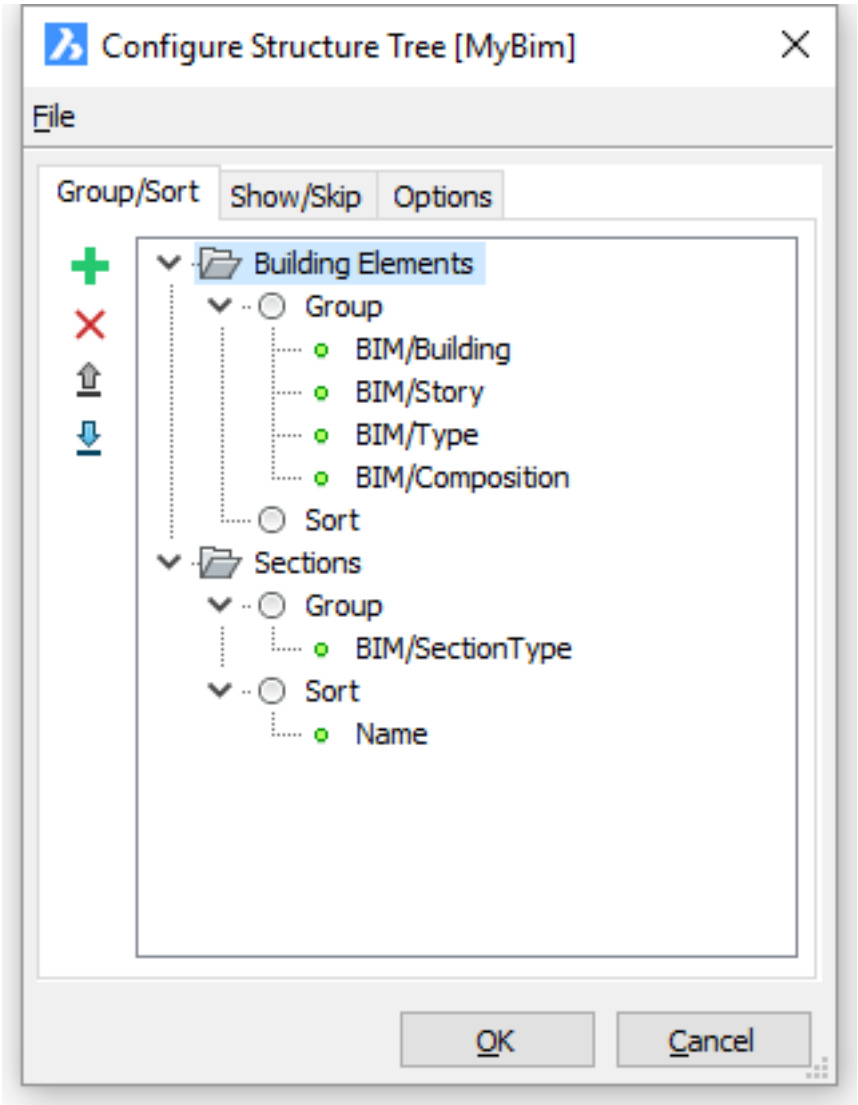


Figure 1.1053

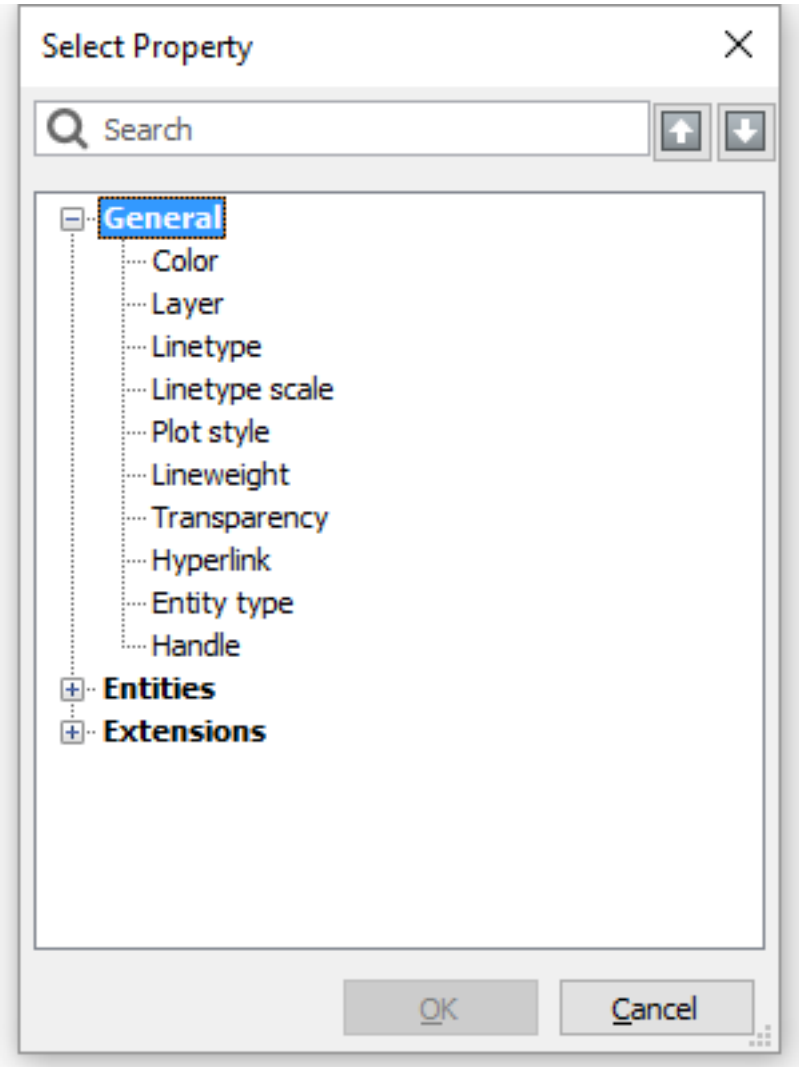


Figure 1.1054

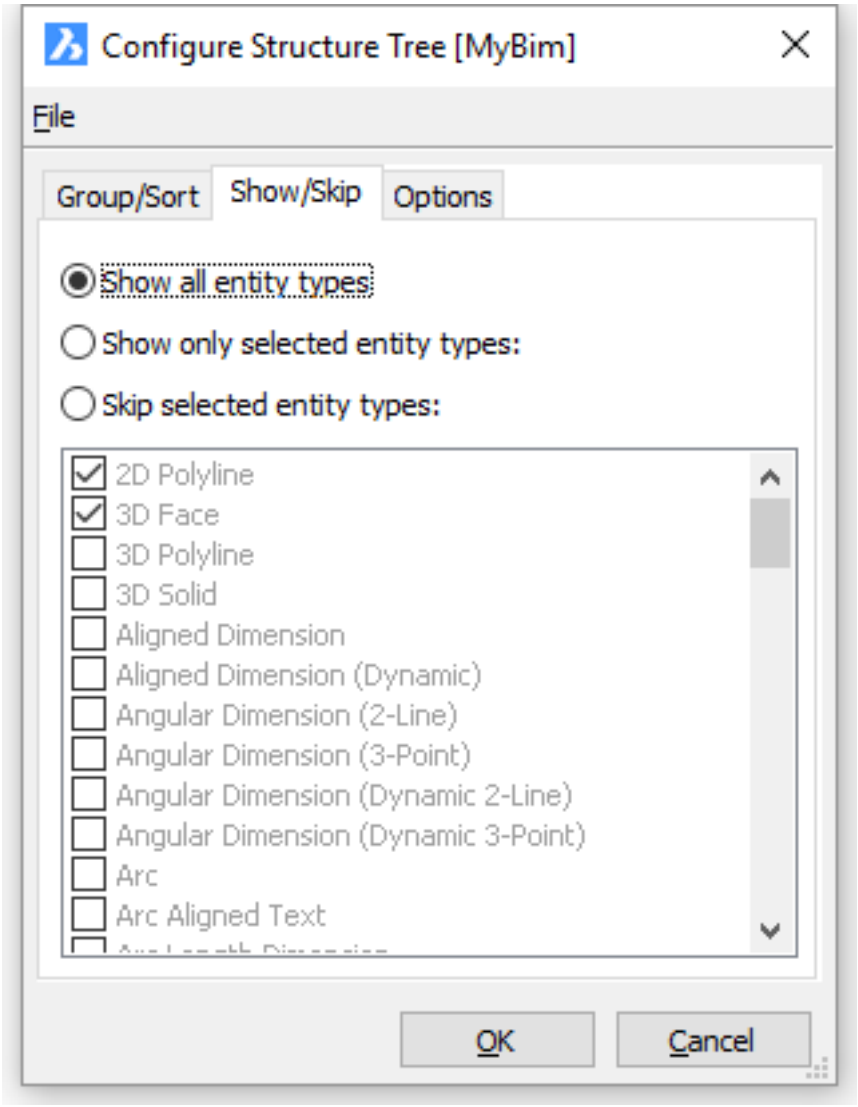


Figure 1.1055

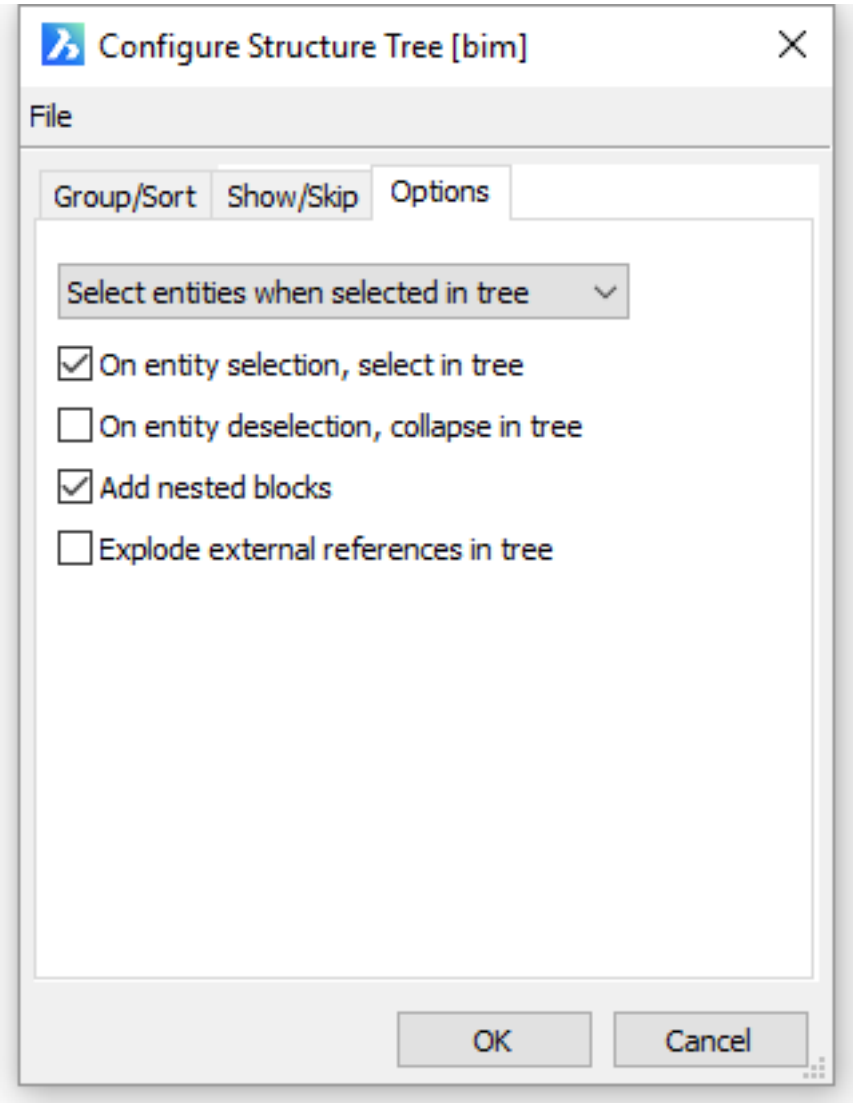


Figure 1.1056

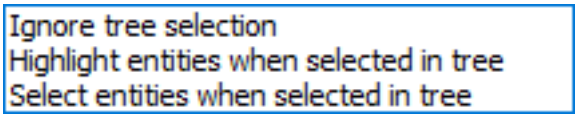


Figure 1.1057

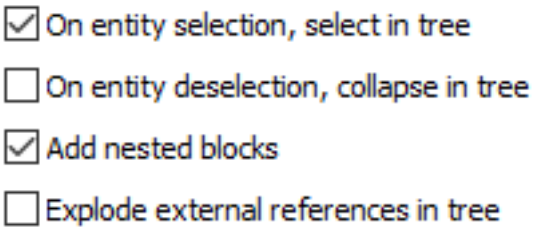
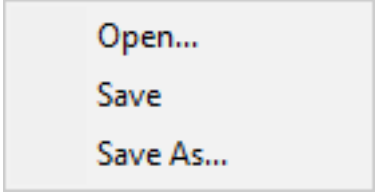


Figure 1.1058



1.23.9 Adjusting the Draw Order

Commands: DRAWORDER (6.10.38), HATCHTOBACK (6.16.6), TEXTTOFRONT (6.29.14) If new entities overlap existing entities, they will display and print on top of the previously drawn entities. The **Draw Order** tool can change the order in which overlapping entities are displayed and printed. You can move entities to the front or the back, or above or under another entity.

Using Draw Order

1. Do one of the following:

   - Click the **Draw Order** tool button ( ) on the **Draw Order** toolbar.
   - Choose **Draw Order** in the **Tools** menu.
   - Type **draworder** in the command bar.

   You are prompted: Select entities to change the draw order:

2. Select the entities (1.23.6), then right click or press enter. **You are prompted:** Change draw order: Above/Under/Clear/Fro
3. Choose the appropriate draw order tool option.
4. Depending on your choice:

   - If you have chosen **Above** or **Under** , the command bar reads: Select reference entity: Identify the reference entity. The display of the selection set is changed with respect to the reference entity.
   - If you have chosen **Front** or **Back** , the display of the selection set is changed accordingly.

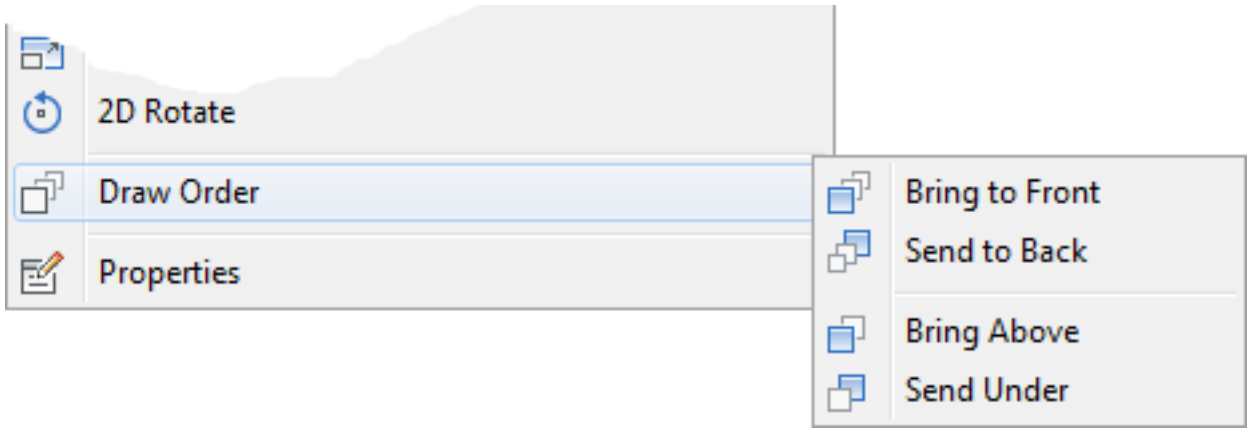
Using Draw Order tools

1. Select the entities (1.23.6) you want to change the draw order of.
2. Do one of the following:

   - Click the appropriate tool button on the **Draw Order** toolbar.
   - Right click, then choose **Draw Order** in the context menu and select the appropriate tool. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/02DrawOrderContext.pr

Tool Button	Tool Name	Description
	Bring To Front	Places the selection set on top.
	Bring Above	Places the selection set above the reference entity. (*)
	Send Under	Places the selection set under the reference entity. (*)

Figure 1.1059



	Send to Back	Moves the selection set to the back.
	Hatch to Back	Sends all hatch entities to the back.
	Text and Dimensions to Front	Brings all texts and/or dimension entities to front. Options: **Text / Dimensions / Both** can be chosen either in the command bar or in the prompt menu. The default option is **Both** .

(*) **You are prompted** to select the reference entity.

1.23.10 Manipulator

Command: MANIPULATE (6.21.2) The manipulator is a versatile grip tool to swiftly move (6.21.2), rotate (6.21.2), scale (6.21.2) and mirror (6.21.2)selected entities. Holding down the Ctrl-key when starting the action keeps the original entities, thus creating a copy. Upon selecting an entity, the manipulator is displayed at a user-controlled location and orientation. The location and orientation are controlled in a similar way as for a dynamic UCS. When multiple entities are selected simultaneously, the manipulator initially is placed at their geometric center. When a selection is built up step by step, the initial location of the manipulator is preserved. The manipulator can easily and directly be dragged to any desired location or orientation using its handles. Depending on the operation type and if dynamic dimensions (1.15.9) are active, a dynamic entry field is available to set the distance, angle or scale factor.

Adaptive Snap Step

The snap step size automatically adapts to the current screen zoom factor if SNAPTYPE= 2. The AdaptiveGrid-StepSize SETTING controls the smallest available step size. The adaptive step size is used by Manipulator Ruler, which enables to quickly modify entities using precise values, without having to key in numbers.

The manipulator settings

To display the manipulator settings in the Settings (1.13.11) dialog:

1. Right click the manipulator.
2. Choose **Manipulator Settings** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/ManipulatorSettings.p

Name	Description
Manipulator	Controls the display of the manipulator widget. 0: The manipulator is not displayed. 1: Display the manipulator whenever entities are selected. 2: Display the manipulator is the left mouse button was pressed longer than the current value of MANIPULATORDURATION.
Manipulator duration	Specifies how long the left mouse button should be pressed during entity selection to trigger the display of the manipulator. The value is expressed in milliseconds.
Size of manipulator	Specifies the size of the manipulator relative to the default. Values between 0.5 and 2.0 are accepted.
Adaptive grid step size	Sets the snap spacing in pixels fo the Adaptive Grid Snap mode of the SNAPTYPE system variable. Values between 2 and 100 are accepted.

Figure 1.1060

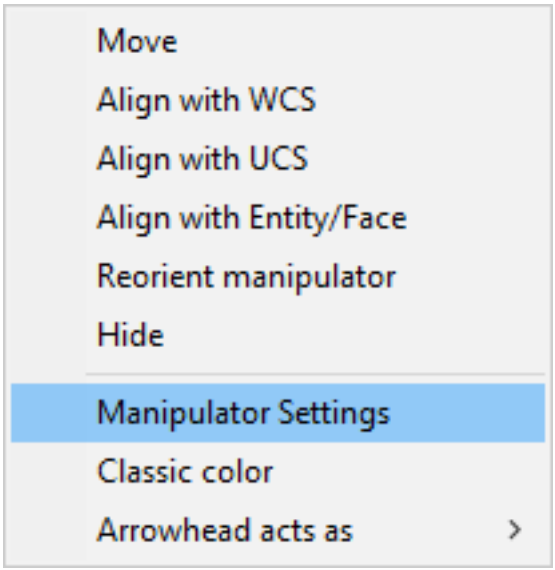
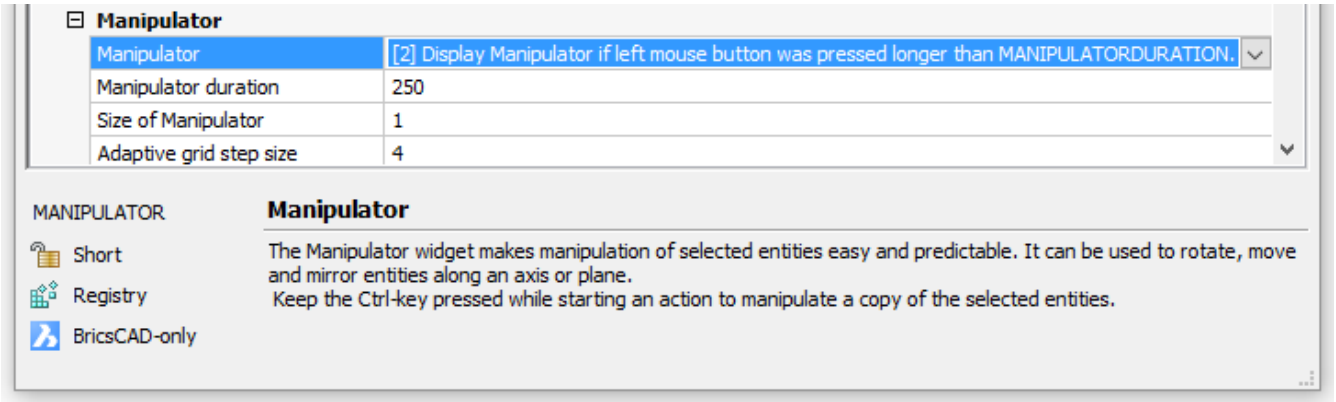


Figure 1.1061



The manipulator layout

Check the **Classic Color** option in the manipulator context menu for the classic layout. Uncheck the option for the default layout.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/Manipulator_Default.png})	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/Manipulator_Classic.png})
Default	Classic

4 handles

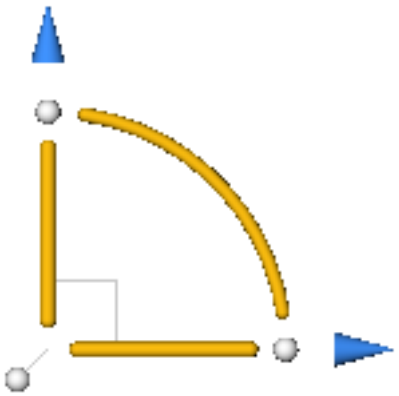
- **1**:anchor handle
- **2**: axis handle

12 command trigger zones:

- **3**: axis
- **4**: arrowhead
- **5**: plane swatch
- **6**: rotation arc

When the current view is parallel to one of the main axes of the coordinate system, as in a plan view, the manipulator is simplified to a 2D version. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/

Figure 1.1062



The location and orientation of the manipulator

The initial location and orientation of the 3D manipulator tripod depends on:

- The entity type.
- The point you click on the entity.

Entity Type Selection point Manipulator Anchor point Orientation Line Near endpoint At endpoint X-axis through the other endpoint Near midpoint At midpoint X-axis through the nearest endpoint Polyline Line Segment Near vertex At vertex X-axis through the other vertex of the selected line segment Near segment midpoint At the segment midpoint X-axis through the nearest vertex Polyline Arc Segment On the arc segment At the arc center X-axis through the nearest vertex Arc Random At the arc center X-axis through the nearest endpoint Circle Random At the circle center X-axis Parallel with the X-axis of the current coordinate system Spline Random At the nearest endpoint Through the other endpoint Region On a line segment At the nearest endpoint of the line segment X-axis through the other endpoint On a arc segment At the center of the arc segment X-axis through the nearest endpoint of the arc segment Text, MText Block, Xref Random At the origin of the entity X-axis at the rotation angle of the entity. 3D Solid* 3D Solid Face* On a face of the solid At the nearest vertex X-axis along the nearest edge XY-plane in the selected face. Multiple Entities Random At the geometric center of the selection set** Parallel with the axes of the current coordinate system * See Selection Modes (1.23.5) to learn how to select a 3D solid face or the entire solid. ** The position of the manipulator does not change when more entities are added to the selection set.

To display the manipulator when multiple entities are selected

Do one of the following:

- Select the entities one by one or use window selection, then click to select the last entity
- Set the value of the MANIPULATOR (1.23.10) system variable to 1: **Display the manipulator whenever entities are selected**.
- Compose the selection set, then launch the MANIPULATE (6.21.2) command.

To relocate the manipulator

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/ManipulatorDefault.png})

To move the manipulator Do one of the following:

- Long-click anywhere on the manipulator and drag it to a new location.
- Drag the manipulator by its anchor handle (1). Specify the new location using entity snaps. The anchor handle stands out to avoid overlap with entity grips.
- Place the cursor over the manipulator then right click and choose **Move** in the context menu. Specify the new location using entity snaps.

To rotate the manipulator Drag one of its axis handles (2) to rotate the manipulator. Pick a point or type a value in the dynamic entry field to specify the rotation angle.

To reorient the manipulator

1. Place the cursor over the manipulator, then right click and choose **Reorient** in the context menu. **You are prompted:** Specify origin of manipulator:
2. Pick a point. The anchor point of the manipulator is placed at the specified point. **You are prompted:** Point on X-axis or <Accept>:
3. Pick a point. The manipulator rotates about its X-axis. **You are prompted:** Point on the XY-plane with positive Y value or <Accept>:
4. Pick a point to define new position of the manipulator.

To align the manipulator with a coordinate system Move the cursor over the manipulator, then right click and select an option in the context menu:

- Align with WCS : the manipulator axes are forced parallel with the corresponding axes of the World Coordinate System.
- Align with UCS : if a UCS is currently the active coordinate system, the manipulator axes are forced parallel with the corresponding axes of the UCS.

Figure 1.1063

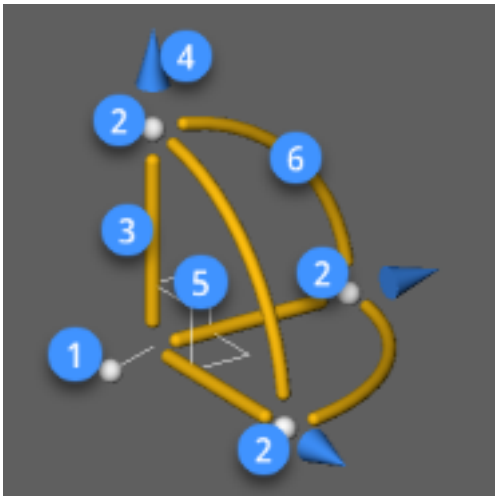
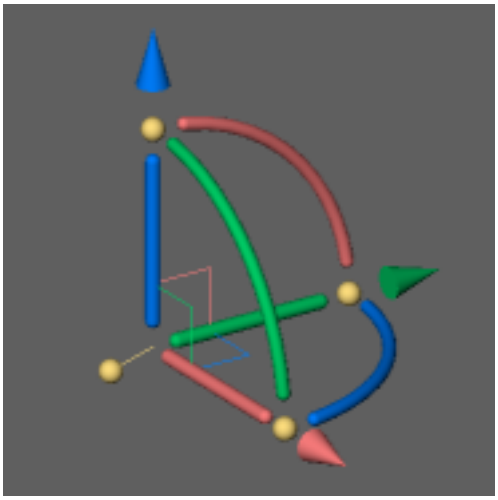


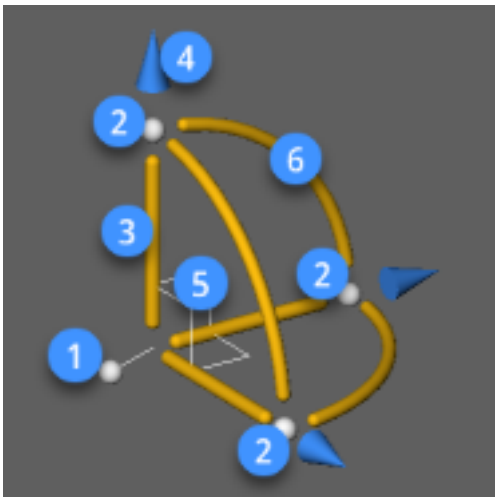
Figure 1.1064



To align the manipulator with the face of a solid Dynamic UCS (1.15.18) (DUCS) must be On for this operation to succeed.

1. Place the cursor over the manipulator then right click and choose **Move** in the context menu.
2. Hover over the face of a solid. The XY-plane of the manipulator aligns with the solid face under the cursor.
3. Do one of the following:
   - Snap to a point on the selected face or on one of its edges.
   - Hit the Shift-key to lock the XY-plane to the selected face, then snap to a point outside the face.

Figure 1.1065



1.23.11 Moving Entities

Commands: MOVE (6.21.30), CUTCLIP (6.9.59), PASTECLIP (6.24.10), PASTEORIG (6.24.41) and PASTEBLOCK (6.24.9) The **Move** command moves entities in the drawing. The **Cutclip** command copies entities to the **Clipboard** , and then erases them from the drawing. The **Pasteclip** command pastes entities from the **Clipboard** into the current drawing (short for ”paste clipboard”). The **Pasteorig** command pastes entities from the **Clipboard** into another drawing, using the coordinates of the source drawing. The **Pasteblock** command pastes entities from the **Clipboard** as a block into the current drawing. Use **Cutclip** then either **Pasteclip, Pasteorig** or **Pasteblock** to move entities to another drawing. The Copy option (1.23.4) of the Grips Editing (1.23.4) commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

NOTE	When DRAGSNAP (1.15.15) is ON, dragged entities display at the current entity snap location.

Moving entities in a drawing

The default method is to create a selection set and then specify a starting point (base point0 and an endpoint (displacement point) to define the relocation of the entities. You can also move the entities using a direction vector. Some entities can be moved using grips. The grip you select depends on the type of entity. For example, to move a line entity, select the midpoint grip. To move a curved entity, such as an arc, circle, or ellipse, select the center point grip. Not all entities can be moved using grips.

Using the Move command

1. Do one of the following:

   - Click the **Move** tool button ( ) on the **Modify** toolbar.
   - Choose **Move** in the **Modify** menu.
   - Type **move** or **M** in the command bar.

   You are prompted: Select entities to move:

2. Select the entities (1.23.6), then right click or press Enter. **You are prompted:** Vector/<Base point>:

3. Specify the base point. The selection set is now attached to the cross hairs. **You are prompted:** Displacement point.

4. Do one of the following to specify the displacement point.

   - Click the displacement point.
   - Use Direct Distance Entry (1.15.3): type the displacement distance, then press Enter. The distance is measured in the cursor direction. Use ORTHO (1.15.2) or POLAR TRACKING (1.15.16) to constrain the movement of the cross hairs.

The selection set is moved.

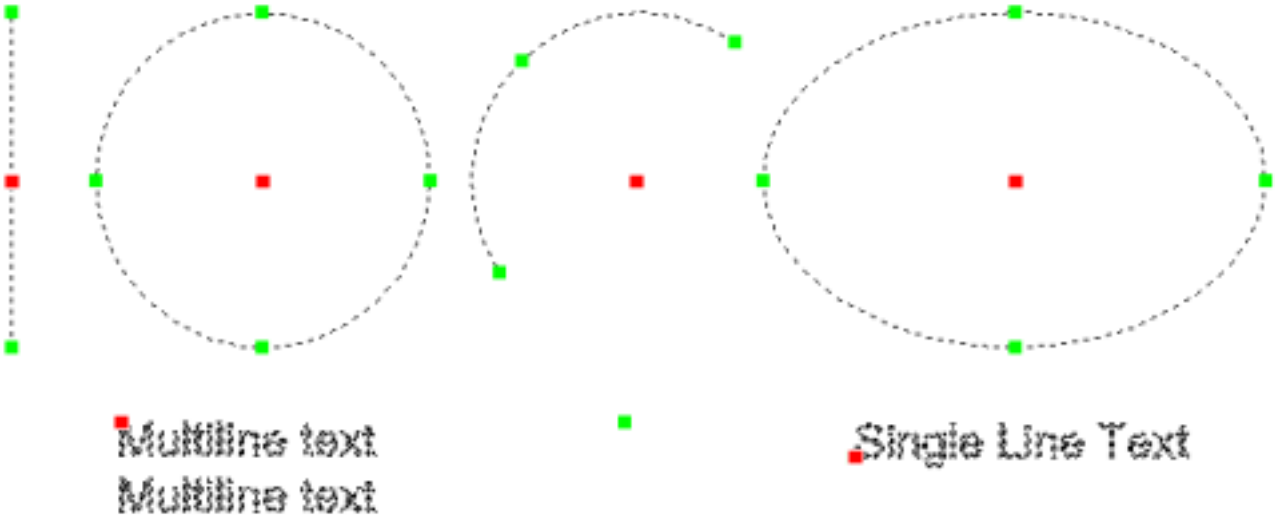
Moving entities using grips

1. Select the entity.

2. Click the grip. The selected grip displays in the **Selected Grip Color** (defined by the **GRIPHOT** system variable, default = RED). The entity moves with the cross hairs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

3. Click to relocate the entity.

Figure 1.1066



NOTE

The following entities can be moved using grips:

- lines, infinite lines and rays
- circles and circular arcs
- ellipses and elliptical arcs
- 3D solids
- text and multi-line text
- blocks

Moving entities between drawings

Entities can be moved between drawings in one of three ways: Use **CUTCLIP** to cut the entities from the source drawing, then either **PASTECLIP, PASTEORIG** or **PASTEBLOCK** to paste the entities into the target drawing.

NOTE

Use Ctrl + Tab (press and hold the Ctrl key, then press the Tab key) to cycle between open drawings.

Using Cut and Paste to move entities between drawings

1. In the source drawing, select the entities (1.23.6) you want to move.
2. Do one of the following:
   - Right click, then choose **Cut** in the context menu.
   - Choose **Cut** in the **Edit** menu.
   - Press **Ctrl + X** (press and hold the Ctrl key, then press X).

   The entities are deleted in the source drawing and copied to the Clipboard.
3. Switch to the target drawing.
4. Do one of the following:
   - Right click, then choose **Paste** in the context menu.
   - Choose **Paste** in the **Edit** menu.
   - Press **Ctrl + V** (press and hold the Ctrl key, then press V).

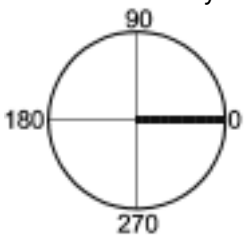
   The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.
5. Do one of the following:

The entities are deleted in the source drawing and copied to the Clipboard.

3. (option) Switch to the target drawing.
4. Choose **Paste as a Block** in the **Edit** menu or press Ctrl+Shfit+V. The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.
5. Specify the insertion point.

1.23.12 Rotating Entities

Commands: ROTATE (6.26.34) and ROTATE3D (6.26.35) You can rotate entities about a specified point at a specified rotation angle or by an angle referenced to a base angle. The default method rotates the entities using a relative rotation angle from their current orientation. By default, angles start at 3 o'clock and increase in a counter-clockwise direction. If you want to rotate in a clockwise direction you can enter a negative angle by using a minus sign.



NOTES

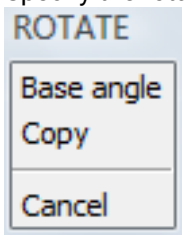
- The **Angular Base** is controlled by the **ANGBASE** system variable, which in turn refers to the current UCS.
- The **Angle Direction** is controlled by the **ANGDIR** system variable, which sets the positive angle direction from angle 0 with respect to the current UCS.

The Copy option (1.23.4) of the Grips Editing (1.23.4) commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

Rotating a selection set

1. Do one of the following:
   - Click the **Rotate** tool button ( ) on the **Modify** toolbar.
   - Choose **Rotate** in the **Modify** menu.
   - Type **rotate** or **RO** in the command bar.

   You are prompted: Select entities to rotate:
2. Select the entities (1.23.6), then right click or press Enter. **You are prompted:** Rotation point:
3. Specify the rotation point. **You are prompted:** Copy/Base angle/<Rotation angle>: A prompt menu displays:



4. (option) Choose **Copy** in the prompt menu or type **C** , then press Enter to create a rotated copy of the selection set.
5. Specify the rotation angle. The selection set is rotated.

NOTE Use the **Center** option of the Array (1.23.19) command if you want to keep the original entities.

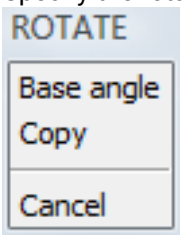
Rotating a selection set in reference to a base angle

1. Do one of the following:
   - Click the **Rotate** tool button (on the **Modify** toolbar.

- Choose **Rotate** in the **Modify** menu.
- Type **rotate** or **RO** in the command bar.

You are prompted: Select entities to rotate:

2. Select the entities (1.23.6), then right click or press Enter. **You are prompted:** Rotation point:
3. Specify the rotation point. **You are prompted:** Copy/Base angle/<Rotation angle>: A prompt menu displays:



4. (option) Choose **Copy** in the prompt menu or type **C**, then press Enter to create a rotated copy of the selection set.
5. Do one of the following:

- Choose **Base angle** in the prompt menu.
- Type **B** in the command bar, then press Enter.

You are prompted: Base angle <0>:

6. To specify the base angle do one of the following:

- Type the base angle in the command bar.
- Click the rotation point again, then click a second point.

You are prompted: New Angle.

7. Do one of the following:

- Type the new angle in the command bar.
- Click a point to define the new angle.

Rotating in 3D

1. Do one of the following.

- Click the **3D Rotate** tool button ( ) on the **Modify** toolbar.
- Choose **3D Rotate** in the **Modify** menu.
- Type **rotate3d** in the command bar.

You are prompted: Select entities to rotate:

2. Select the entities (1.23.6), then right click or press Enter. **You are prompted:** Select axis by [Entity/Last/View/Xaxis/Yaxis <2points>:
3. Press Enter to define the rotation axis by specifying two points.
4. Specify the first rotation axis point.
5. Specify the second rotation axis point. **You are prompted:** Rotation angle or [Base angle/Copy]:
6. Specify the rotation angle. The selection set is rotated clockwise, looking in the rotation axis direction.

1.23.13 Mirroring Entities

Commands: MIRROR (6.21.18) and MIRROR3D (6.21.19) The **Mirror** command creates a mirror image of a selection set. The selection is mirrored about a mirror line, which you define by specifying two points. You can choose to either delete or retain the original entities. The **Mirror3d** commands creates a mirror image of selected entities in three-dimensional space. In this case the selection is mirrored about a mirror plan, which can be defined by either specifying three points, selecting an existing two-dimensional planar entity, aligning the plane parallel to the xy, yz, or xz plane of the current UCS or aligning the plane with the current view. You can choose to either delete or retain the original entities. The Copy option (1.23.4) of the Grips Editing (1.23.4) commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

NOTE	Whether text is mirrored or not by the **Mirror** command is controlled by the **Mirror Text** (**MIRRTEXT**) system variable.

Mirroring entities about a line

1. Do one of the following:

   - Click the **Mirror** tool button ( ) on the **Modify** toolbar.
   - Choose **Mirror** in the **Modify** menu.
   - Type **mirror** or **MI** in the command bar.

   You are prompted: Select entities to mirror:

2. Select the entities (1.23.6) then right click or press Enter. **You are prompted:** Start of mirror line.
3. Specify the start point of the mirror line.
4. Specify the endpoint of the mirror line. **You are prompted:** Delete the original entities? <N>
5. Do one of the following:

   - Press Enter to keep the original entities.
   - Type **Y** , then press Enter to delete the original entities.
   - Choose **Yes-Delete entities** in the context menu.

Mirroring entities about a plane

1. Do one of the following:

   - Click the **3D Mirror** tool button ( ) on the **Modify** toolbar.
   - Choose **3D Mirror** in the **Modify** menu.
   - Type **mirror3d** in the command bar.

   You are prompted: Select entities:

2. Select the entities (1.23.6) then right click or press Enter. **You are prompted:** Define mirror plane by: Entity/Last/View/Zaxis/XY/YZ/ZX/<3points>:
3. Specify the first point of the mirror plane.
4. Specify the second point of the mirror plane.
5. Specify the third point of the mirror plane. **You are prompted:** Delete the original entities? <N>
6. Do one of the following:

   - Press Enter to keep the original entities.
   - Type **Y**, then press Enter to delete the original entities.
   - Choose **Yes-Delete entities** in the context menu.

1.23.14 Aligning Entities

Command: ALIGN (6.4.6) The **Align** command lets you reposition a selection set with respect to an existing entity in the drawing. The selection set is moved and rotated in a single action. If necessary you can even scale the selection to fit in its new location. The command can be used both in a 2D and 3D environment.

Tutorial: How to Align Entities Click here (1.4.9) to watch

Aligning an entity in 2D

1. Do one of the following:

   - Click the **Align** tool button ( ) on the **Modify** toolbar.
   - Choose **Align** in the **Modify** menu.
   - Type **align** in the command bar, then press Enter.

   You are prompted: Select entities:

2. Select the entities (1.23.6), then right click or press Enter. **You are prompted:** Specify first source point:

3. Snap to the first source point in the selection set (1). **You are prompted:** Specify first destination point:

4. Snap to the first destination point on the reference entity (2). A witness line is drawn between the source point and the target point. **You are prompted:** Specify the second source point: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/02Align1.png})

5. Snap to the second source point in the selection set (3). **You are prompted:** Specify second destination point:

6. Snap to the second destination point on the reference entity (4). A witness line is drawn between the source point and the target point. **You are prompted:** Specify the third source point:

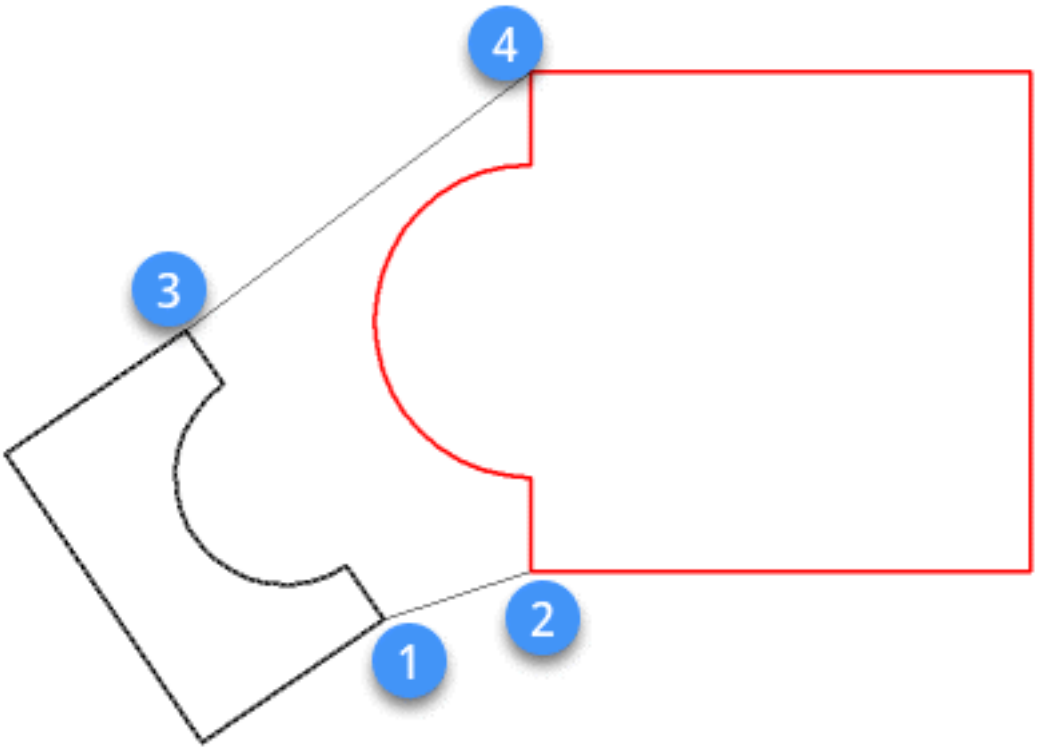
7. Right click to skip the third source point. **You are prompted:** Scale objects based on alignment points [Yes/No] <No>:

8. Do one of the following:

   - Press Enter if you don't want to scale the selection set.
   - Type Y and press Enter to scale the selection set.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/02Align2.p
Result of the Align procedure with scaled selection set.

Figure 1.1067



Aligning an entity in 3D

1. Do one of the following:

   - Click the **Align** tool button ( ) on the **Modify** toolbar.
   - Choose **Align** in the **Modify** menu.
   - Type **align** in the command bar, then press Enter.

   You are prompted: Select entities:

2. Select the entities (1.23.6), then right click or press Enter. **You are prompted:** Specify first source point:
3. Snap to the first source point in the selection set (1). **You are prompted:** Specify first destination point:
4. Snap to the first destination point on the reference entity (2). A witness line is drawn between the source point and the target point. **You are prompted:** Specify the second source point: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/02Align3D1.png})
5. Snap to the second source point in the selection set (3). **You are prompted:** Specify second destination point:
6. Snap to the second destination point on the reference entity (4). A witness line is drawn between the source point and the target point. **You are prompted:** Specify the third source point:
7. Snap to the third source point in the selection set (5). **You are prompted:** Specify third destination point:
8. Snap to the third destination point on the reference entity (6). A witness line is drawn between the source point and the target point. The entity is moved. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGu

Figure 1.1068

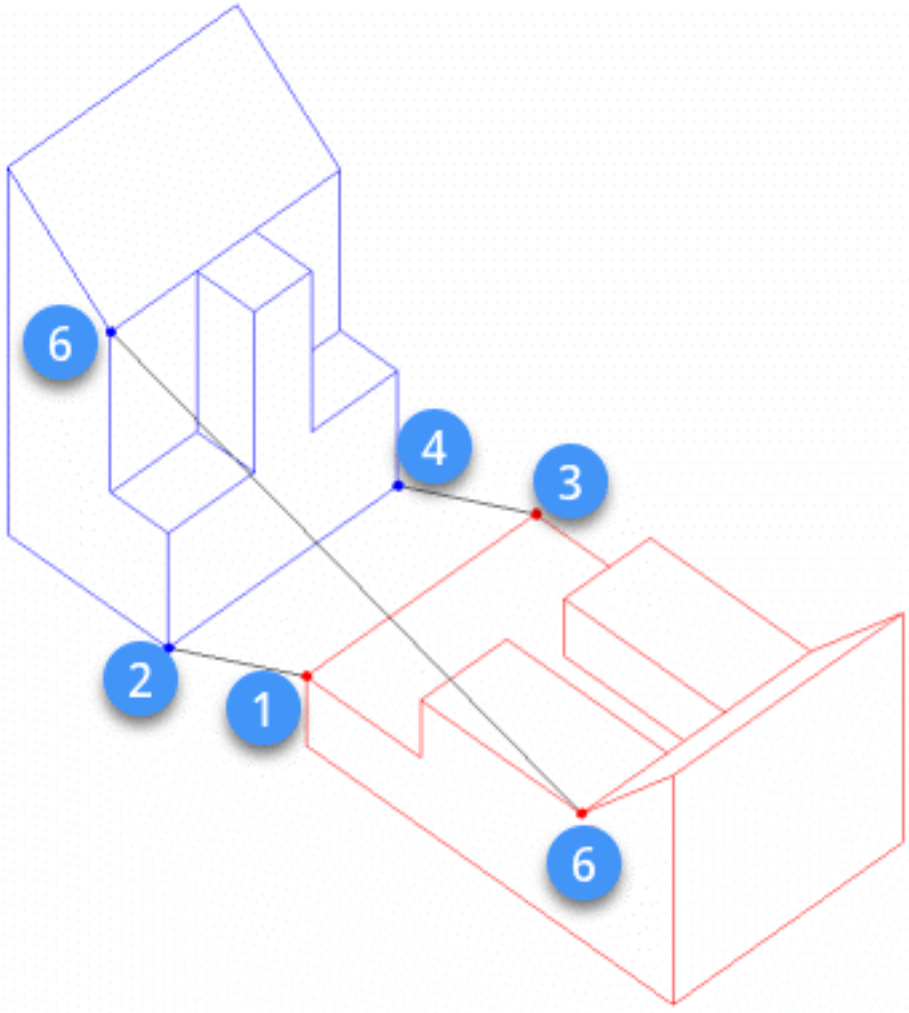


Figure 1.1069

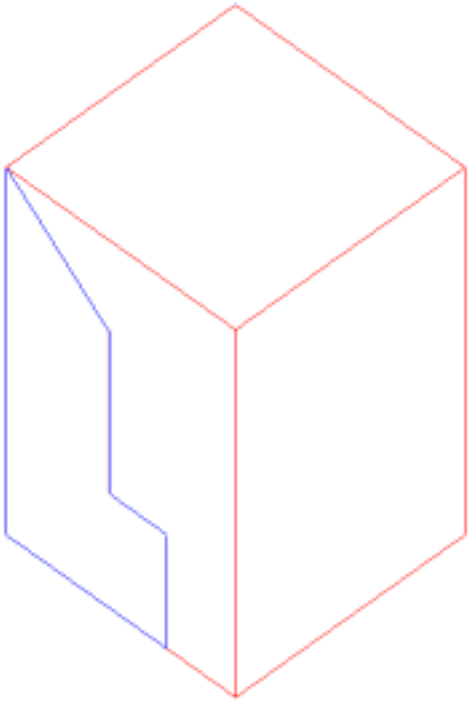
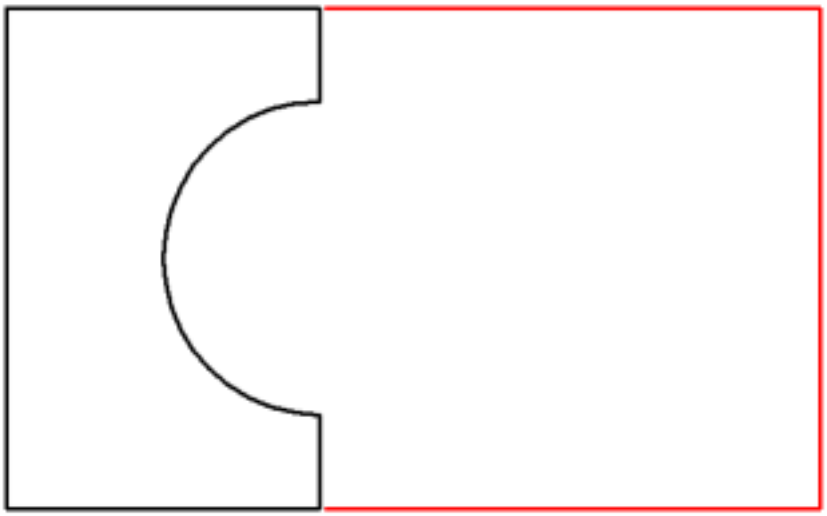


Figure 1.1070



1.23.15 Changing Entities

Command: CHANGE (6.9.10) The **Change** command lets you change the insertion point and orientation of texts and block inserts. The **Properties** option of the command can also change the layer, line type, line type scale, line weight, etc. but the BricsCAD Properties bar (1.12.4) is much more versatile to do such things.

To change text entities

1. Type **change** in the command bar, then press Enter. **You are prompted:** Select entities to change:
2. Select the text entities (1.23.6), then right click or press Enter. **You are prompted:** Change: Entities/Properties/<Change point>:
3. Right click or press Enter to accept the default option. **You are prompted:** New point for text, or Enter for no change: The first text entity is attached to the cursor.
4. Specify the new insertion point for the text or right click to accept the current location. **You are prompted:** New text style <current style>:
5. Type a new text style and press Enter or right click to keep the current style. **You are prompted:** New height <current height>:
6. Type a new height and press Enter or right click to keep the current height. **You are prompted:** New rotation angle <current angle>:
7. Type a new angle and press Enter or right click to keep the current angle. **You are prompted:** New text <current text>:
8. Type the new text and press Enter or right click to keep the current text.
9. (option) If multiple text entities were selected in step 2, steps 3 through 8 are repeated for each text.

NOTE	The **Change** command cannot change Multi-Line text (Mtext) entities.

To relocate blocks

1. Type **change** in the command bar, then press Enter. **You are prompted:** Select entities to change:
2. Select the blocks (1.23.6), then right click or press Enter. **You are prompted:** Change: Entities/Properties/<Change point>:
3. Right click or press Enter to accept the default option. **You are prompted:** New point for block, or Enter for no change: The first block is attached to the cursor.
4. Specify the new insertion point or right click to accept the current orientation. **You are prompted:** New rotation angle <current angle>:
5. Do one of the following:
   - Right click to accept the current rotation angle.
   - Type a new rotation angle in the command bar, then press Enter.
   - Click to specify the new rotation angle graphically.
6. (option) If multiple blocs were selected in step 2, steps 3 through 5 are repeated for each block.

1.23.16 Copying Entities in a Drawing

Commands: COPY (6.9.50), COPYCLIP (6.9.52), COPYBASE (6.9.51),PASTECLIP (6.24.10) and PASTEBLOCK (6.24.9)
You can duplicate entities within the current drawing using the **Copy** command. The default method is to create a selection set and then specify a starting point (base point) and an endpoint (second point) for the copy. You can also specify the displacement as a direction vector (X,Y,Z). Depending on the **COPYMODE** system variable you can create multiple copies of the selection set or just one. **Copybase** and **Pasteblock** pastes the selection as a block. The Copy option (1.23.4) of the Grips Editing (1.23.4) commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

Tutorial: Copying between drawings Click here (1.4.10) to watch

To copy a selection set

1. Do one of the following:

   - Click the **Copy** tool button ( ) on the **Modify** toolbar.
   - Choose **Copy** in the **Modify** menu.
   - Type **copy** or **CO** in the command bar.

   You are prompted: Select entities to copy:

2. Select the entities (1.23.6), then right click or press Enter. If **Copy Mode** is **Multiple** : **You are prompted:** Enter base point: mOde/<Displacement>: If **Copy Mode** is **Single** : **You are prompted:** Enter base point: mOde/Multiple/<Displacement>:

3. Specify the base point. The selection set is now attached to the cross hairs. **You are prompted:** Enter second point: <Enter to use the base point as displacement>:

4. Do one of the following:

   - Click the second point.
   - Use Direct Distance Entry (1.15.3): type the displacement distance, then press Enter. The distance is measured in the cursor direction. Use ORTHO (1.15.2) or POLAR TRACKING (1.15.16) to constrain the movement of the cross hairs.
   - Press <Enter>: the first point is interpreted as a relative X,Y,Z displacement. E.g. if the coordinates of the base point are 2,3,0, the selection set is copied 2 units in the X direction and 3 units in the Y direction from their current position.

The selection set is copied. If **Copy Mode** is **Single** , the copy command is completed. If **Copy Mode** is **Multiple** , **you are prompted** to create the next copy. See To make multiple copies.

To make multiple copies

1. Do one of the following:

   - Click the **Copy** tool button ( ) on the **Modify** toolbar.
   - Choose **Copy** in the **Modify** menu.
   - Type **copy** or **CO** in the command bar.

   You are prompted: Select entities to copy:

2. Select the entities (1.23.6), then right click or press Enter. If **Copy Mode** is **Multiple** : **You are prompted:** Enter base point: mOde/<Displacement>: If **Copy Mode** is **Single** : **You are prompted:** Enter base point: mOde/Multiple/<Displacement>:

3. (Option) If **Copy Mode** is **Single** , do one of the following:

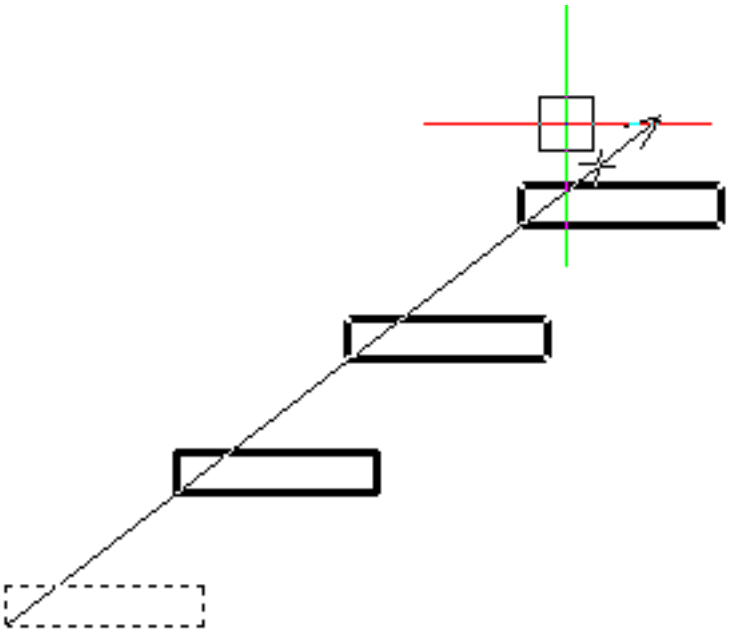
   - Choose **Multiple** in the prompt menu
   - Type **M** , then press Enter

   You are prompted: Enter base point: mOde/<Displacement>:

4. Specify the base point. The selection set is now attached to the cross hairs. **You are prompted:** Enter second point: <Enter to use the base point as displacement>:

5. Do one of the following to specify the second point.

Figure 1.1071



- Click the second point.
- Type the displacement distance, then press Enter. The distance is measured in the cursor direction. Use ORTHO (1.15.2) or POLAR TRACKING (1.15.16) to constrain the movement of the cross hairs.

The first copy of the selection set is created. **You are prompted:** Enter second point: Undo/Repeat/Exit:

6. (option) Repeat step 5 to create additional copies. Right click or press Enter to conclude the command.
7. (option) Choose the **Repeat** option. **You are prompted:** Select end point (<ENTER> to key in number of repeats) The number of repeats increases when you move the cursor away from the source selection set. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/03CopyRep
8. Do one of the following:
   - Click when the desired number of copies displays.
   - Press Enter or right click, then type the desired number of copies.

To paste a selection set as a block

1. Do one of the following:
   - Choose **Copy with Base Point** in the **Edit** menu.
   - Type **copybase** in the command bar, then press Enter.
   - Press and hold the **Ctrl** and **Shift** keys, then press **C** .

   You are prompted: Select base point:
2. Specify the base point (origin point). **You are prompted:** Select entities to copy to the clipboard:
3. Select the entities.
4. Press Enter to conclude the selection of entities.
5. Do one of the following:
   - Choose **Paste as Block** in the **Edit** menu.
   - Type **pasteblock** in the command bar, then press Enter.
   - Press and hold the **Ctrl** and **Shift** keys, then press **V** .

   The selection set is attached to the cursor. The command window reads: Select insertion point.
6. Specify the insertion point. The selection is pasted as a block.

7. (option) Repeat steps 5 and 6 to paste another copy as a block.

NOTE	If you paste multiple copies of the same selection set, each block is named differently.

1.23.17 Copying Entities Between Drawings

Commands: COPYCLIP (6.9.52), COPYBASE (6.9.51), PASTECLIP (6.24.10), PASTEORIG (6.24.41) and PASTEBLOCK (6.24.9) Entities can be copied between drawings in several ways:

- **Copyclip** and **Pasteclip** pastes the selection using a default base point.
- **Copyclip** and **Pasteorig** pastes the selection using the coordinates of the source drawing.
- **Copybase** and **Pasteclip** pastes the selection using a user-defined base point.
- **Copybase** and **Pasteblock** : pastes the selection as a block in the target drawing.

NOTE	Use Ctrl + Tab (press and hold the Ctrl key, then press the Tab key) to cycle between open drawings.

Using Copy and Paste to copy entities between drawings

1. In the source drawing, select the entities (1.23.6) you want to Copy.
2. Do one of the following:
   - Right click, then choose **Copy** in the context menu.
   - Choose **Copy** in the **Edit** menu.
   - Press **Ctrl + C** (press and hold the Ctrl key, then press C).

   The entities are copied to the Clipboard.
3. Switch to the target drawing.
4. Do one of the following:
   - Right click, then choose **Paste** in the context menu.
   - Choose **Paste** in the **Edit** menu.
   - Press **Ctrl + V** (press and hold the Ctrl key, then press V).

   The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.
5. Do one of the following:
   - Specify the displacement point.
   - Press Enter to paste the selection set using the coordinates of the source drawing.

Using Copy and Pasteorig to copy entities between drawings

1. In the source drawing, select the entities (1.23.6) you want to Copy.
2. Do one of the following:
   - Right click, then choose **Copy** in the context menu.
   - Choose **Copy** in the **Edit** menu.
   - Press **Ctrl + C** (press and hold the Ctrl key, then press C).

   The entities are copied to the Clipboard.
3. Switch to the target drawing.
4. Do one of the following:
   - Choose **Paste to Original Coordinates** in the Edit menu.
   - Type **pasteorig** in the command bar, then press Enter.

   The selection is pasted using the coordinates of the source drawing.

Using Copybase and Paste to copy entities between drawings

1. In the source drawing do one of the following:

   - Choose **Copy with Base Point** in the **Edit** menu.
   - Type **copybase** in the command bar, then press Enter.
   - Press and hold the **Ctrl** en **Shift** keys, then press **C** .

   You are prompted: Select base point:

2. Specify the base point (origin point). **You are prompted:** Select entities to copy to the clipboard:
3. Select the entities.
4. Press Enter to conclude the selection of entities.
5. Switch to the target drawing.
6. Do one of the following:

   - Choose **Paste** in the **Edit** menu.
   - Press **Ctrl + V** (press and hold the Ctrl key, then press V).

   The selection set is attached to the cursor. **You are prompted:** Select insertion point:

7. Specify the insertion point to paste the selection set.
8. (option) Repeat step 7 to create more copies of the selection set.

Tutorial: Copying between drawings Click here (1.4.10) to watch

Paste a selection as a block in the target drawing

1. In the source drawing do one of the following:

   - Choose **Copy with Base Point** in the **Edit** menu.
   - Type **copybase** in the command bar, then press Enter.
   - Press and hold the **Ctrl** en **Shift** keys, then press **C** .

   You are prompted: Select base point:

2. Specify the base point (origin point). **You are prompted:** Select entities to copy to the clipboard:
3. Select the entities.
4. Press Enter to conclude the selection of entities.
5. Switch to the target drawing.
6. Do one of the following:

   - Choose **Paste as Block** in the **Edit** menu.
   - Type **pasteblock** in the command bar, then press Enter.
   - Press and hold the **Ctrl** and **Shift** keys, then press **V** .

   The selection set is attached to the cursor. The command window reads: Select insertion point.

7. Specify the insertion point. The selection is pasted as a block.
8. (option) Repeat steps 5 and 6 to paste another copy as a block.

NOTE	If you paste multiple copies of the same selection set, each block is named differently.

1.23.18 Making Parallel Copies

Command: OFFSET (6.23.3) The **Offset** command creates a copy of linear entities and aligns them parallel to the original entities at a specified distance. You can make parallel copies of arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays and infinite lines. Depending on the selected options in the **Selection Modes** toolbar, the **Offset** command allows to create a parallel copy of:

- an edge of a 3D solid if the **Select Edges** option ( ) is set.
- the edges of a 3D solid face if the **Select Faces** option ( ) is set.
- detected boundaries if the **Select Detected Boundaries** option ( ) is set.

Making parallel copies of curved entities creates larger or smaller curves, depending on which side of the original entity you place the copy. For example, placing a parallel copy of a circle outside the circle creates a larger concentric circle; positioning the copy inside the circle creates a smaller concentric circle.

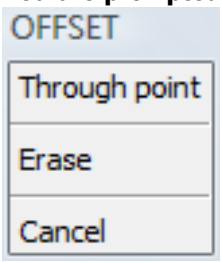
NOTE	The **OFFSETGAPTYPE** settings variable controls how potential gaps between segments are treated when closed polylines are offset.

Making a parallel copy at a specified distance

1. Do one of the following:

   - Click the **Offset** tool button ( ) on the **Modify** toolbar.
   - Choose **Offset** in the **Modify** menu.
   - Type **offset** in the command bar.

   You are prompted: Offset: Through point/Erase/<Distance> <current distance>: A prompt menu displays:

2. (Option) Choose **Erase** in the prompt menu or type **E**, then press Enter. Respond **Yes** at the command prompt to erase the source entity.

3. Do one of the following:

   - Press Enter to accept the current distance.
   - Type a new distance in the command bar.
   - Define a new distance by specifying two points.

   You are prompted: Select Entity.

4. Select the entity. **You are prompted:** Both sides/<Side for parallel copy>:

5. Click the side for the parallel copy. The parallel copy is created. **You are prompted:** Select Entity.

6. Do one of the following:

   - Repeat steps 4 and 5 to create more parallel copies.
   - Right click or press Enter to stop.

Making a parallel copy through a point

1. one of the following:

   - Click the **Offset** tool button ( ) on the **Modify** toolbar.
   - Choose **Offset** in the **Modify** menu.

- Type **offset** in the command bar.

You are prompted: Offset: Through point/<Distance> <current distance>:

2. Choose **Through point** in the context menu or type **T**, then press Enter. **You are prompted:** Select entity:
3. Select the entity. **You are prompted:** Through point:
4. Specify the point to create the parallel copy through. The parallel copy is created. **You are prompted:** Select entity:
5. Do one of the following.
   - Repeat steps 3 and 4 to create more parallel copies.
   - Right click or press Enter to stop.

Making a parallel copy of a 3D solid edge

Make sure the Quad (1.12.5) is active and the **Select edges** button ( ) in the **Selection Modes** toolbar is pressed.

1. Move the cursor over the edge of the solid. When the edge highlights, choose **Parallel** ( ) in either the **2D Editing** or the **Direct Modeling** command group in the **Quad** . The parallel copy is created dynamically.
2. Do one of the following:
   - Type a distance in the dynamic entry field.
   - Snap to a point.

Making a parallel copy a 3D solid face

Make sure the Quad (1.12.5) is active and the **Select Faces** button ( ) in the **Selection Modes** toolbar is pressed.

1. Move the cursor over the face of the solid. When the face highlights, choose **Parallel** ( ) in either the **2D Editing** or the **Direct Modeling** command group in the **Quad** . The parallel copy is created dynamically.
2. Do one of the following:
   - Type a distance in the dynamic entry field.
   - Snap to a point.

Making a parallel copy of a detected boundary

Make sure the Quad (1.12.5) is active and the **Select Detected Boundaries** button ( ) in the **Selection Modes** toolbar is pressed.

1. Move the cursor inside the boundary. When the boundary highlights, choose **Parallel** ( ) in either the **2D Editing** or the **Direct Modeling** command group in the **Quad** . The parallel copy is created dynamically.
2. Do one of the following:
   - Type a distance in the dynamic entry field.
   - Snap to a point.

1.23.19 Arraying Entities

Commands: ARRAY (6.4.16) , -ARRAY (6.4.15), ARRAYCLASSIC (6.4.17), ARRAYRECT (6.4.25), ARRAYPATH (6.4.21), ARRAYPOLAR (6.4.23),ARRAYEDIT (6.4.20), 3DARRAY (6.3.2) The **Array** commands copy a selection set in a rectangular or polar (circular) pattern or along a linear entity. For a rectangular array you must specify the number of rows and columns and also the spacing between subsequent rows and columns. To create a polar array you will be prompted to specify the center point of the array, the rotation step and the number of items in the array or the angle to fill. You can choose to rotate the selection set about the center point or to keep its original orientation. The **ArrayClassic** command creates a static polar or rectangular array of entities through a dialog box. The **-Array** command creates a static polar or rectangular array of entities. The **3DArray** command constructs static 3D rectangular arrays and rotated polar arrays The **Array** command distributes entity copies in an evenly spaced associative rectangular, polar or path array. The **ArrayRect** command associatively distributes entity copies into any number of rows, columns, and levels. The **ArrayPolar** command associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. The **ArrayPath** command associatively distributes entity copies evenly along a path into multiple rows, and levels. The **ArrayEdit** command allows editing associative arrays and their source entities.

Creating a static rectangular array

1. Type **arrayclassic** in the command bar, then press Enter. A dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/03ArrayRectangularDialog.p
2. Under **Array Type** choose **Rectangular**.
3. Click the **Select Entities** button ( ). The **Array** dialog box closes temporarily to let you select entities.
4. Select the entities (1.23.6), then right click or press Enter. The **Array** dialog box reopens.
5. Fill out the **Settings** fields or press the **Pick Points** buttons ( ) to define the settings graphically. The **Array** dialog box closes temporarily to let you pick point in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h
Row offset (A) and column offset (B) can be defined by the spacing rectangle (red).
6. Click the OK button to create the array.

NOTE	Positive values in steps 4 and 5 are measured along the positive X- and Y-axis of the current UCS. Negative values are measured in the opposite direction.

Figure 1.1072

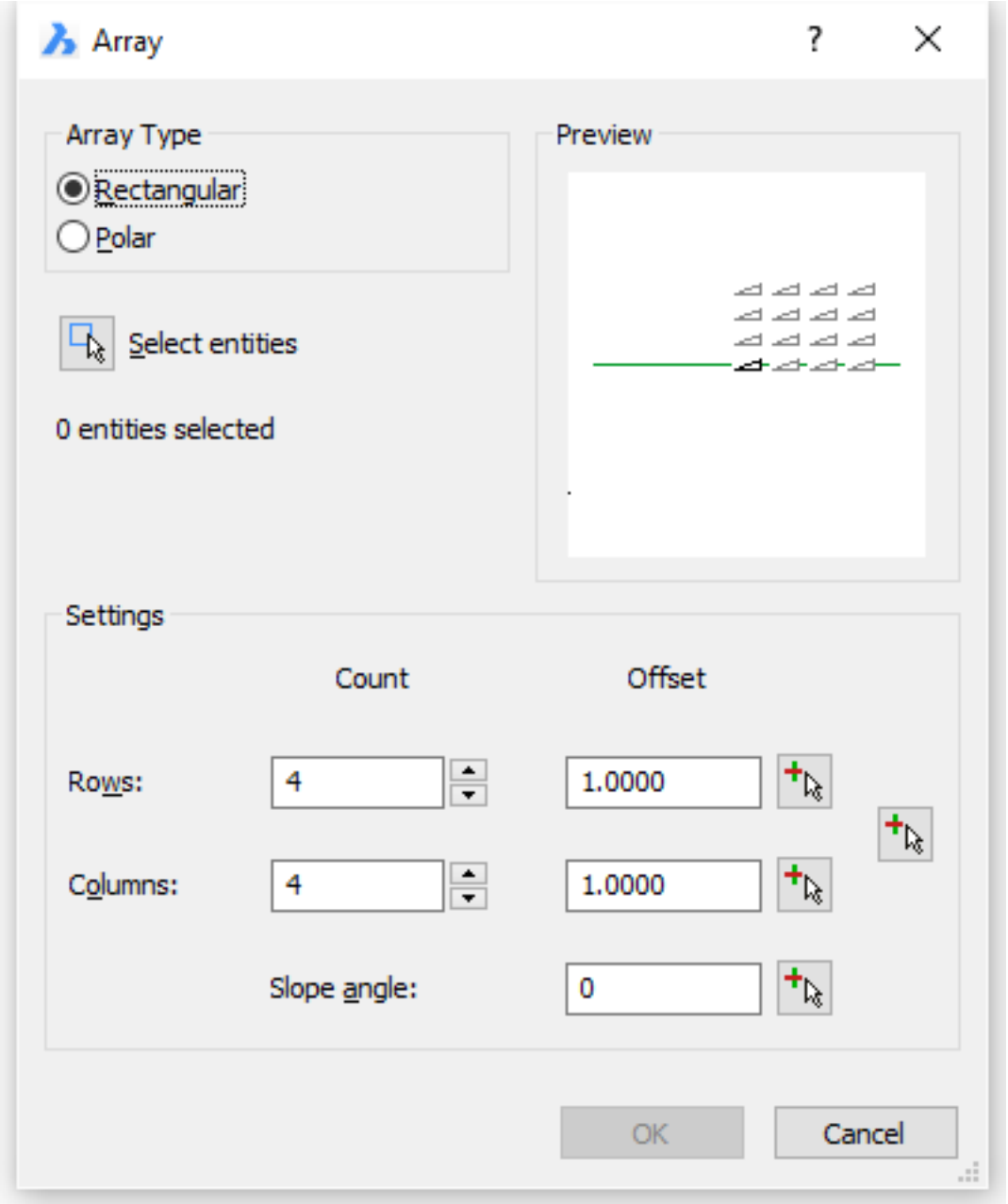
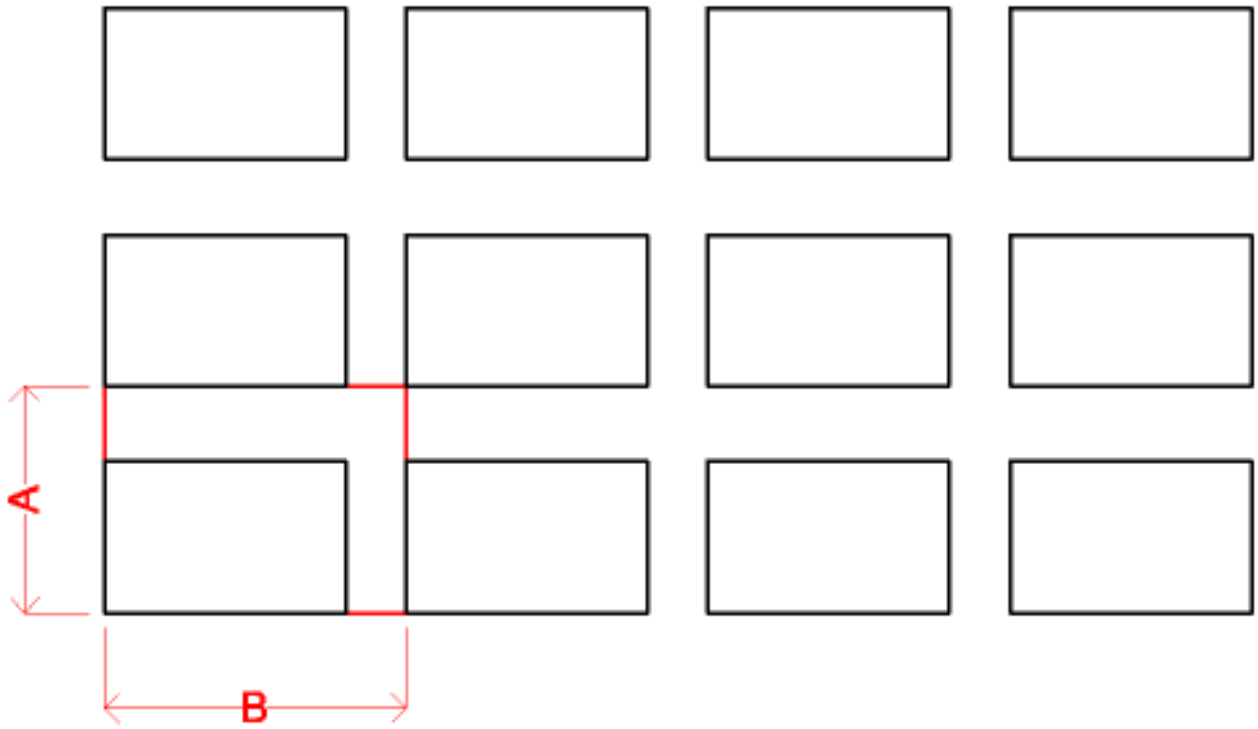


Figure 1.1073



Creating a static polar array

1. Type **arrayclassic** in the command bar, then press Enter. A dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/03ArrayPolarDialog.png})
2. Under **Array Type** choose **Polar** .
3. Click the **Select Entities** button ( ). The **Array** dialog box closes temporarily to let you select entities.
4. Select the entities (1.23.6), then right click or press Enter. The **Array** dialog box reopens.
5. Choose a **Calculate** option: **Angle between**, **Angle to fil** l or **Number of items**. The settings field of the selected option is dimmed.
6. Fill out the **Settings** fields or press the **Pick Points** buttons ( ) to define the settings graphically. The **Array** dialog box closes temporarily to let you pick point in the drawing.
7. (option) Check the **Rotate items as copied** option. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 **Entities rotated (left) or not (right) about the center point of the array.**
8. (option) Define an **Entity base point**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sou **Polar array with base point (dot) and center point (cross).**

Figure 1.1074

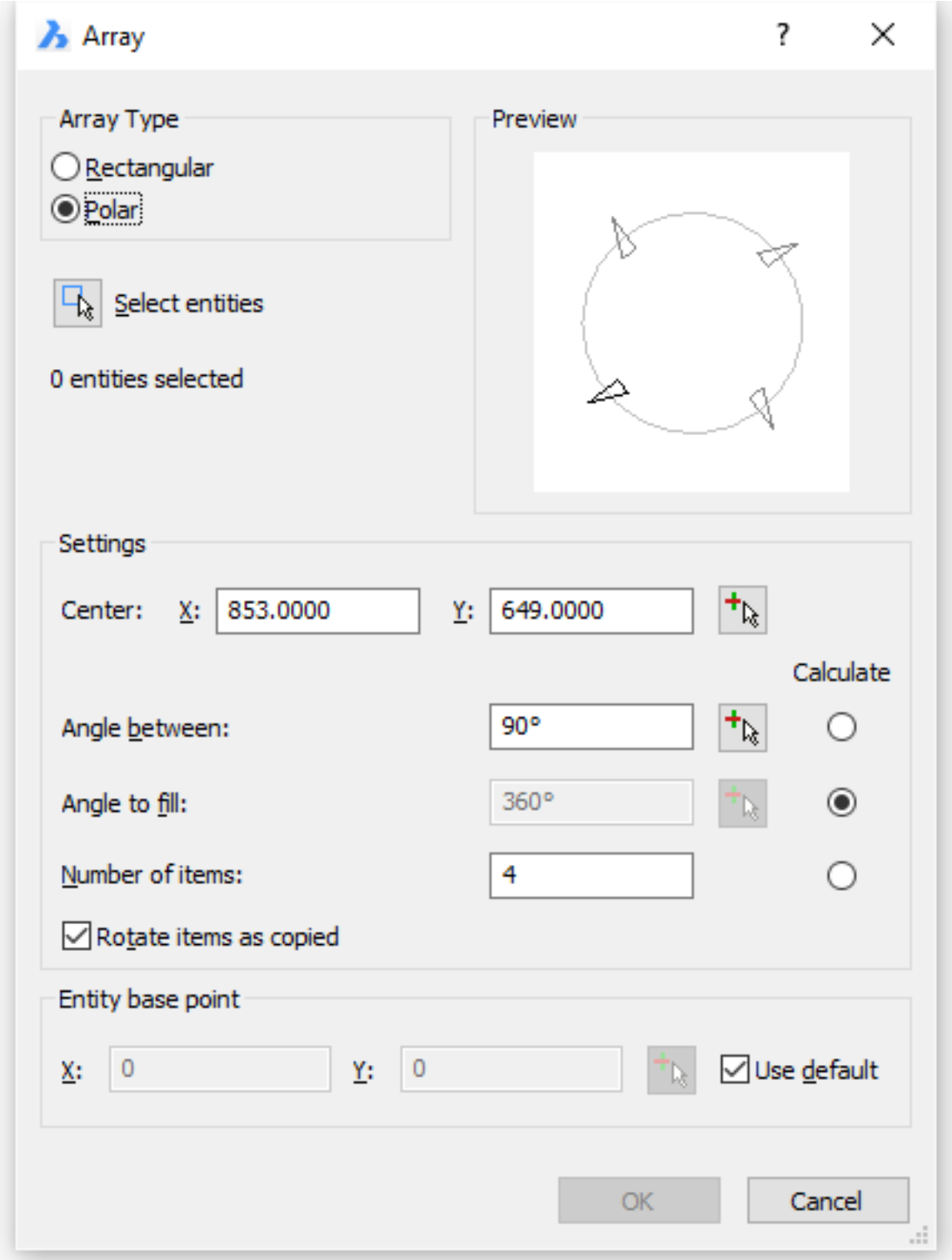


Figure 1.1075

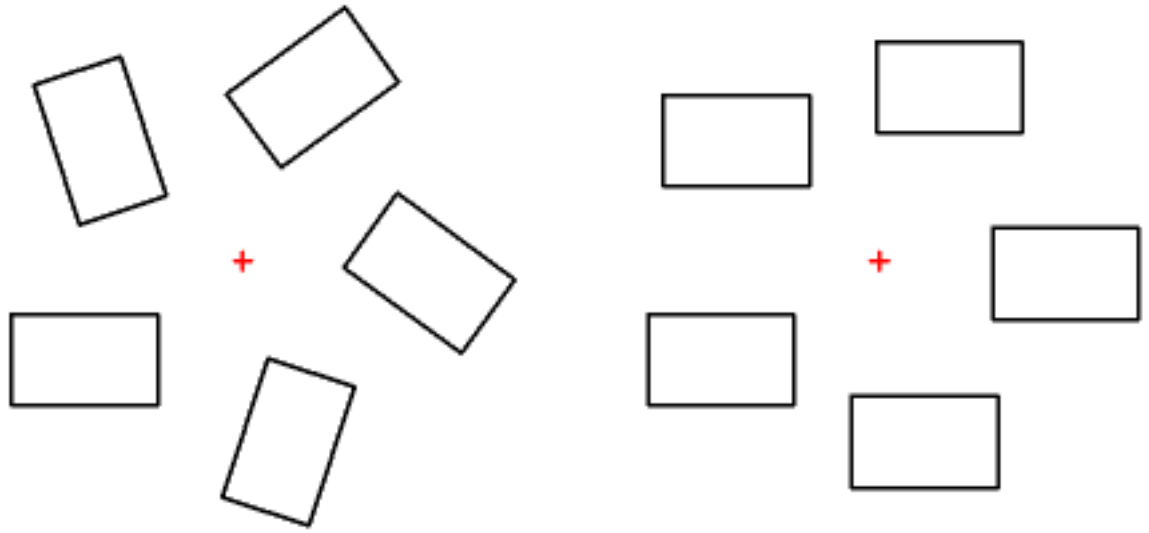
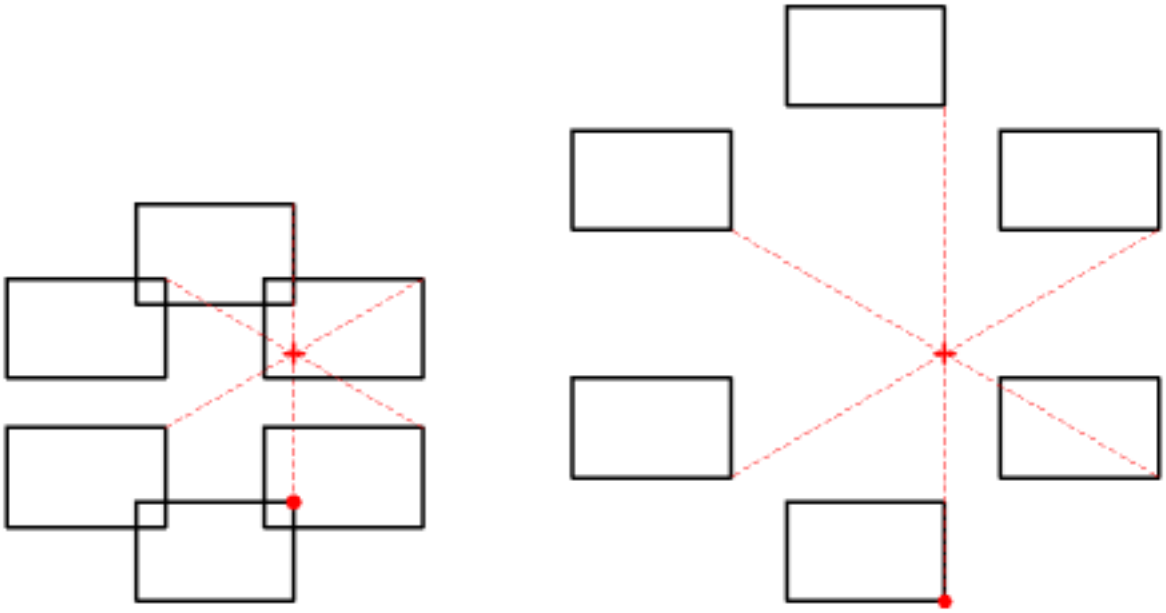


Figure 1.1076



Creating a static 3D rectangular array

1. Do one of the following:

   - Click the **3D Array** tool button ( ) on the **Modify** toolbar.
   - Choose **3D Array** in the **Modify** menu.
   - Type **3darray** in the command bar, then press Enter.

   You are prompted: Select entities to array:

2. Select the entities (1.23.6), then right click or press Enter. **You are prompted:** Type of array: Polar/<Rectangular>:
3. Choose **Rectangular** in the prompt menu or type **R** in the command bar and press Enter. **You are prompted:** Number of rows in the array <1>:
4. Type the number of rows in the command bar, then press Enter. **You are prompted:** Number of columns <1>:
5. Type the number of columns in the command bar, then press Enter. **You are prompted:** Number of levels <1>:
6. Type the number of levels in de command bar, then press Enter. **You are prompted:** Vertical distance between rows:
7. To define the distance between rows of the array, do one of the following:

   - Type the distance in the command bar and press Enter.
   - Specify the distance by clicking two points.

   You are prompted: Horizontal distance between columns:

8. To define the distance between the columns of the array, do one of the following:

   - Type the distance in the command bar and press Enter.
   - Specify the distance by clicking two points.

   You are prompted: Depth between levels:

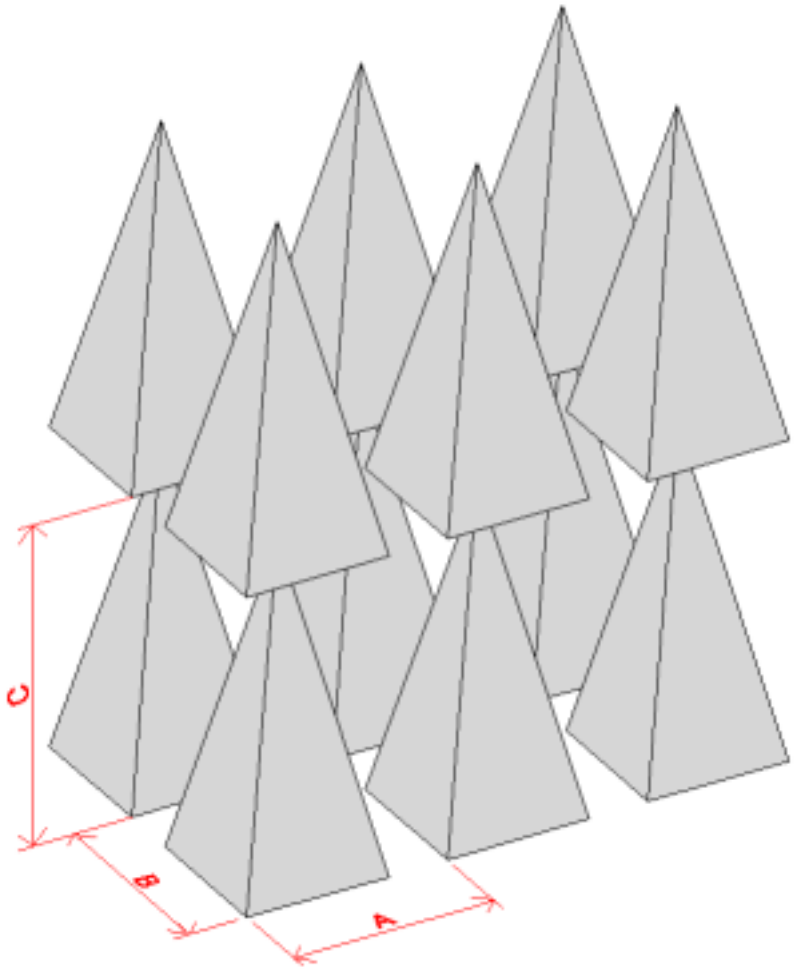
9. To define the distance between the levels of the array, do one of the following:

   - Type the distance in the command bar and press Enter.
   - Specify the distance by clicking two points.

The array is created. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntit
Rectangular 3D Array:row spacing A, column spacing B, level spacing C

NOTE	Positive values in steps 7, 8 and 9 are measured along the positive X-, Y- and Z-axis of the current UCS. Negative values are measured in the opposite direction.

Figure 1.1077



Creating a static 3D polar array

1. Do one of the following:

   - Click the **3D Array** tool button ( ) on the **Modify** toolbar.
   - Choose **3D Array** in the **Modify** menu.
   - Type **3darray** in the command bar, then press Enter.

   You are prompted: Select entities to array:

2. Select the entities (1.23.6), then right click or press Enter. **You are prompted:** Type of array: Polar/<Rectangular>:

3. Choose **Polar** in the prompt menu or type **P** in the command bar and press Enter. **You are prompted:** ENTER to specify angle between items/<Number of items to array>:

4. Type the number of items you want in the array, then press Enter. **You are prompted:** Angle to array (+ for ccw, - for cw) <360>:

5. Do one of the following:

   - Type the angle to array, then press Enter.
   - Press Enter to create a 360° array.

   You are prompted: Rotate entities around the array? No/<Yes>

6. Do one of the following:

   - Press Enter to rotate the entities about the array axis.
   - Type N and press Enter to keep the original orientation of the entities.

   You are prompted: Center of polar array.

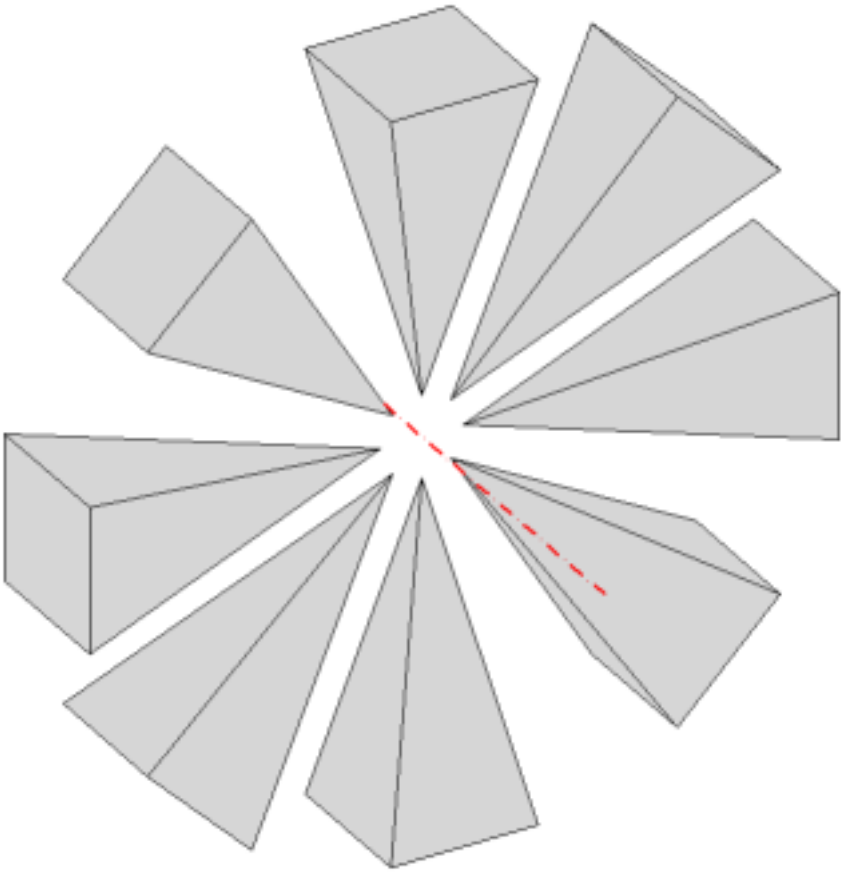
7. Specify the first point of the array axis. **You are prompted:** Specify second point along central axis of array:

8. Specify the second point of the array axis.

The array is created. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntit
Polar 3D array about a horizontal axis (red).

Tutorial: Creating and Editing Associative Arrays Click here (1.5.4) to watch

Figure 1.1078



1.23.20 Extending Entities

Command: EXTEND (6.13.19) and TRIM (6.29.26) The **Extend** command lets you extend entities to a boundary, which is defined by one or more other entities. If the **Edge Mode** setting (**EDGEMODE**) is **On** , you can extend entities to an implied edge of the boundary entities. If the boundary entity is not in the same plane as the entity you want to extend, the **Projection Mode** setting (**PROJMODE**) lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view plane
- True 3D mode (No projection).

When extending entities, first select the boundary edges, and then specify the entities to extend, selecting them either one by one or using the fence selection method. The following entities can be extended: arcs, lines, two-dimensional polylines, rays. Boundary entities can be: arcs, circles, ellipses, lines, splines, polylines, rays, infinite lines, layout viewports.

To extend entities

1. Do one of the following:

   - Click the **Extend** tool button ( ) on the **Extend/Stretch** flyout of **Modify** toolbar.
   - Choose **Extend** in the **Modify** menu.
   - Type **extend** or **EX** in the command bar, then press Enter.

   You are prompted: Select boundary entities for extend <ENTER to select all>:

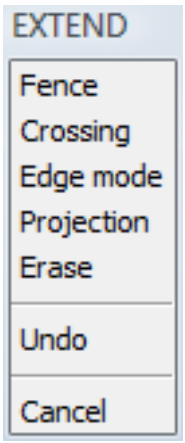
2. Select the boundary entities, then press Enter or right click. **You are prompted:** Fence/Crossing/Edge mode/Projection/eE entity to extend or shift-select to trim>: A prompt menu displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/

3. Click the entity you want to extend near the end that can make the extension. The entity is extended. **Or** Press and hold the shift key to trim (1.23.21) the entity.

4. Repeat step 3 to extend more entities.

5. Right click or press Enter to conclude the **Extend** command.

Figure 1.1079



NOTES

- Entities which are selected when you launch the **Extend** command will be used as boundary entities.
- If you select an entity near the end that cannot make an extension to one of the boundary entities, the **Extend** command is aborted.

1.23.21 Trimming Entities

Command: TRIM (6.29.26) and EXTEND (6.13.19) The Trim command lets you clip or trim entities by cutting entities. If the **Edge Mode** (**EDGEMODE**) system variable is **On** , you can trim entities by an implied edge of the cutting entities. If the cutting entity is not in the same plane as the entity you want to trim, the **Projection Mode** (**PROJMODE**) system variablele defines how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view plane
- True 3D mode (No projection).

When trimming entities, you first select the cutting edges, and then specify the entities to trim, selecting them either one by one or using the fence selection method. The following entities can be trimmed: lines, two- and three dimensional polylines, arcs, circles, ellipses, elliptical arcs, splines, rays and infinite lines. Cutting entities can be: lines, splines, polylines, arcs, circles, elliptical arcs, ellipses, rays, infinite lines, layout viewports. Entities in a block can be selected as cutting entities.

To trim entities

1. Do one of the following:

   - Click the **Trim** tool button ( ) on the **Modify** toolbar.
   - Choose **Trim** in the **Modify** menu.
   - Type **trim** or **TR** in the command bar, then press Enter.

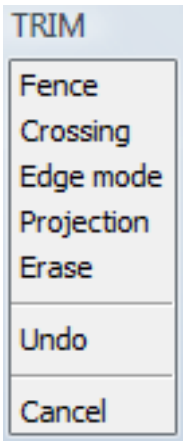
   You are prompted: Select cutting entities for trim <ENTER to select all>:

2. Select the cutting entities, then press Enter or right click. **You are prompted:** Fence/Crossing/Edge mode/Projection/eRa
   entity to trim or shift-select to extend>: A prompt menu displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/

3. Click the entity you want to trim, the part of the entity that you click will be removed. The entity is trimmed. **Or** Press and hold the shift key to extend (1.23.20) the entity.
4. Repeat step 3 to trim more entities.
5. Right click to conclude the **Trim** command.

NOTES	Entities which are selected when you launch the **Trim** command will be used as cutting entities.

Figure 1.1080



1.23.22 Changing the length of an entity

Command: LENGTHEN (6.20.21) The **Lengthen** command lets you change the length of lines, open polylines and arcs. You can also modify the included angle of arcs.

Change the length of an entity dynamically

1. Do one of the following:

   - Click the **Lengthen** tool button ( ) on the **Modify** toolbar.
   - Choose **Lengthen** in the **Modify** menu.
   - Type **lengthen** in the command bar, then press Enter.

   You are prompted: Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEnti

2. (option) Click an entity. The current length of the selected entity displays in the command bar. In case an arc is selected, the current length and the included angle display in the command bar.

3. Do one of the following:

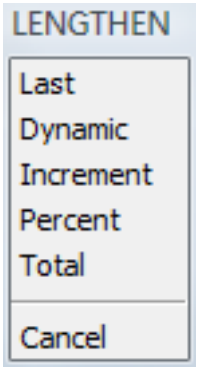
   - Choose **Dynamic** in the prompt box.
   - Type **DY** in the command bar, then press Enter.

   You are prompted: Mode/<Select entity to change>:

4. Click the entity near the end you want to change. The length of the entity changes dynamically.

5. Click to define the new length.

Figure 1.1081



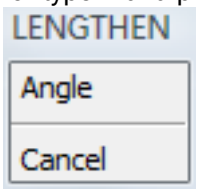
Modify the included angle of an arc

1. Do one of the following:

   - Click the **Edit Length** tool button ( ) on the **Modify** toolbar.
   - Choose **Edit Length** in the **Modify** menu.
   - Type **editlen** in the command bar, then press Enter.

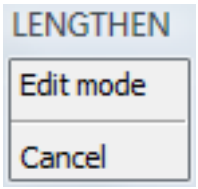
   You are prompted: Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>:

2. Choose **Total** in the prompt menu or type **T** and press Enter. **You are prompted:** Angle/<Enter total Length (0.00)>: A prompt menu displays:

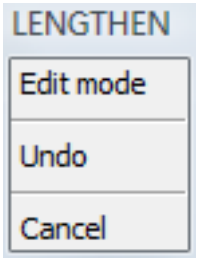


3. Choose **Angle** in the prompt menu or type **A** and press Enter. **You are prompted:** Enter total angle <00° 0’ 0”>:

4. Type the new angle in the command bar and press Enter. **You are prompted:** Mode/<Select entity to change>: A prompt menu displays:



5. Click the arc at the end you want to lengthen or shorten. The arc is modified. **You are prompted:** Mode/Undo/<Select entity to change>: A prompt menu displays:



6. Do one of the following:

   - Select another arc.
   - Choose **Undo** in the prompt menu or type **U** and press Enter to undo the previous action.
   - Choose **Edit mode** in the prompt menu or type **M** and press Enter to choose a different **Edit Length** command option.
   - Choose **Cancel** in the prompt menu to conclude the **Edit Length** command.

NOTE	In the Properties bar (1.17.16) you can edit the **start angle** , **end angle** and**radius** of an arc.

1.23.23 Stretching Entities

Command: STRETCH (6.27.49) The size and shape of entities can be changed by stretching them. You select an area in your drawing using either a rectangular window or a polygon, then you specify a base point and a displacement point. All points and nodes inside the selected area will be moved over the specified distance. As a result, entities that cross the window or polygon boundary are stretched; those completely within the window or polygon are simply moved. The Copy option (1.23.4) of the Grips Editing (1.23.4) commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

To stretch entities

1. Do one of the following:

   - Click the **Stretch** tool button ( ) on the **Extend/Stretch** flyout of the **Modify** toolbar.
   - Choose **Stretch** in the **Modify** menu.
   - Type s**tretch** or **S** in the command bar and press Enter.

   You are prompted: Select entities to stretch by crossing-window or crossing-polygon: A prompt menu displays: (\vref{fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART

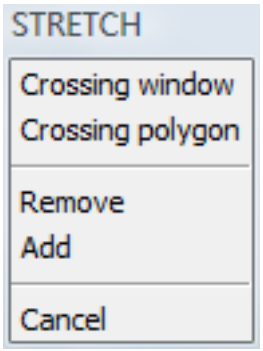
2. (option) Choose **Crossing window** in the **Stretch** prompt menu, then define the stretch area by a rectangular window.
3. (option) Choose **Crossing polygon** in the **Stretch** prompt menu, then define the stretch area by a polygon.
4. (option) Repeat steps 2 and 3 to expand the stretch area.
5. (option) Choose **Remove** in the **Stretch** prompt menu to select entities in the stretch area that must not be stretched.
6. (option) Choose **Add** in the **Stretch** prompt menu to add previously remove entities to the selection of entities that must be stretched.
7. Right click to conclude the selection of entities. **You are prompted:** Base point of displacement.
8. Specify the base point. The selection stretches dynamically. **You are prompted:** Second point of displacement.
9. Do one of the following to specify the second displacement point.

   - Click the displacement point.
   - Use Direct Distance Entry (1.15.3): type the displacement distance, then press Enter. The distance is measured in the cursor direction. Use ORTHO (1.15.2) or POLAR TRACKING (1.15.16) to constrain the movement of the cross hairs.

The selection is stretched.

NOTE	To add or remove entities from the selection set in steps 5 and 6 you can use any selection method: picking, window inside or crossing window.

Figure 1.1082



Stretching entities using grips

1. Click the entity you want to stretch. The entity grips display.
2. Click a grip to activate it. The grip is attached to the drawing cursor.
3. Press and hold the Shift key to select multiple grips, then release the Shift key and move one of the selected grips.
4. Click to relocate the grip. The grip is released from the drawing cursor.

NOTES

- Use Ortho Mode (1.15.2), Polar Tracking (1.15.16) or Snap Tracking (1.15.17) to constrain the movement of the drawing cursor.
- If you select two (or more) entities with coinciding grips, the shared grips move simultaneously.

1.23.24 Scaling Entities

Command: SCALE (6.27.6) The **Scale** command resizes a selection set in relation to a base point. You can specify the scale factor by selecting a base point and a length or by typing an explicit scale factor in the command bar. The scale factor can also be defined with respect to a base scale factor. E.g. when the base scale factor is 2 and the new scale is 3, the new size is 3/2 of the original. The base scale and the new scale can also be defined graphically in the drawing. The Copy option (1.23.4) of the Grips Editing (1.23.4) commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

To scale a selection set

1. Do one of the following:

   - Click the **Scale** tool button () on the **Modify** toolbar
   - Choose **Scale** in the **Modify** menu.
   - Type **scale** or **SC** in the command bar.

   You are prompted: Select entities to scale:

2. Select the entities (1.23.6), then right click or press Enter. **You are prompted:** Base point:

3. Specify the base point for the scaling. The selection set scales dynamically. **You are prompted:** Base scale/<Scale Factor>:

4. Type the scale factor in the command bar and press Enter. The selection set is scaled.

NOTE	The length of the vector between the base point and the current cursor position is used as the dynamic scale factor in step 3.

Scaling an entity using a base or reference scale

1. Select the entity.

2. Do one of the following:

   - Click the **Scale** tool button () on the **Modify** toolbar
   - Choose **Scale** in the **Modify** menu.
   - Type **scale** or **SC** in the command bar.

   You are prompted: Base point:

3. Specify the base point (1). **You are prompted:** Base Scale/<Scale Factor>: (\vref {fig:C:/Dev/Task-2448/images/staticbri

4. Choose **Base** in the prompt menu of type **B** or **R** and press Enter. **You are prompted:** Base scale <1>:

5. Click point 1, then point 2 to define the base scale. The pentagon scales dynamically.

6. Click point 3. The edge of the pentagon now equals the side of the square. (\vref {fig:C:/Dev/Task-2448/images/staticbri

Figure 1.1083

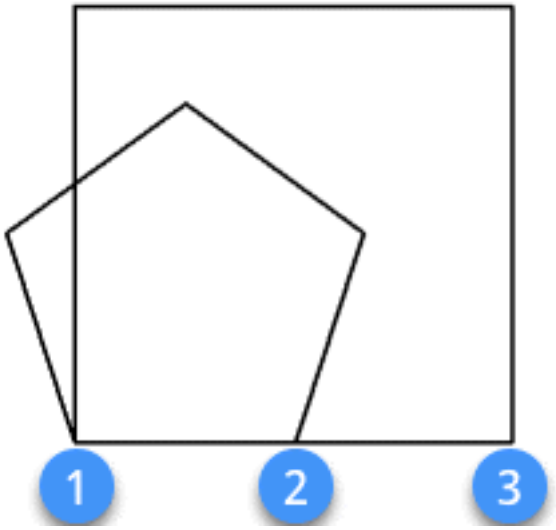
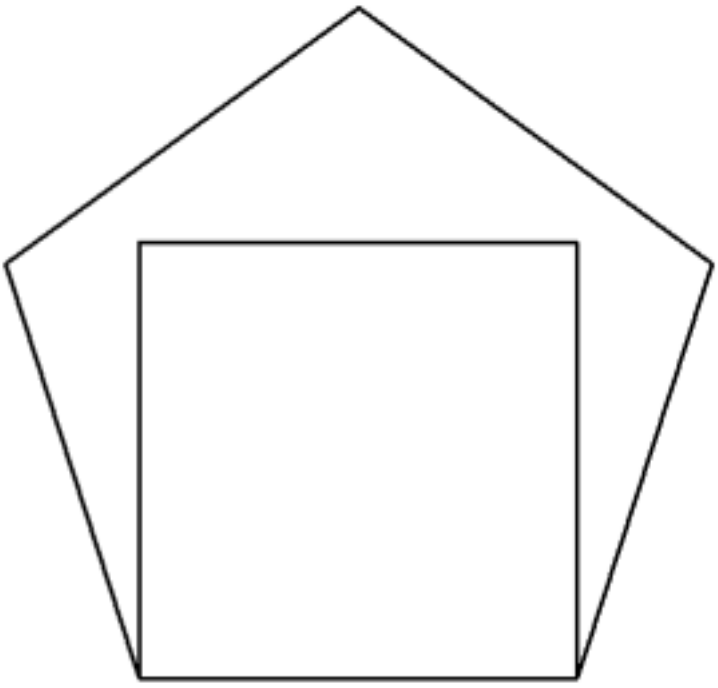


Figure 1.1084



1.23.25 Joining Entities

Command: JOIN (6.18.1) The **Join** command joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints. The type of the resulting entity depends on the types of the input entities and on their coplanarity.

Tutorial: Joining Multiple Entities into One Click here (1.3.19) to watch

To join colinear lines

When you join colinear lines, the farthest endpoints remain at their existing locations; BricsCAD draws a new line between these points.

1. Do one of the following:

   - Click the **Join** tool button ( ) on the **Modify** toolbar.
   - Choose **Join** in the **Modify** menu.
   - Type **join** in the command bar, then press Enter.

   You are prompted: Select source entity or multiple entities to join at once:

2. Select the lines. **You are prompted:** Select entities to join:

3. Right click or press Enter. The lines are joined.

To join two arcs

Coplanar arcs, sharing midpoint and radius, are joined counterclockwise, therefore the result depends on the selecting order.

1. Do one of the following:

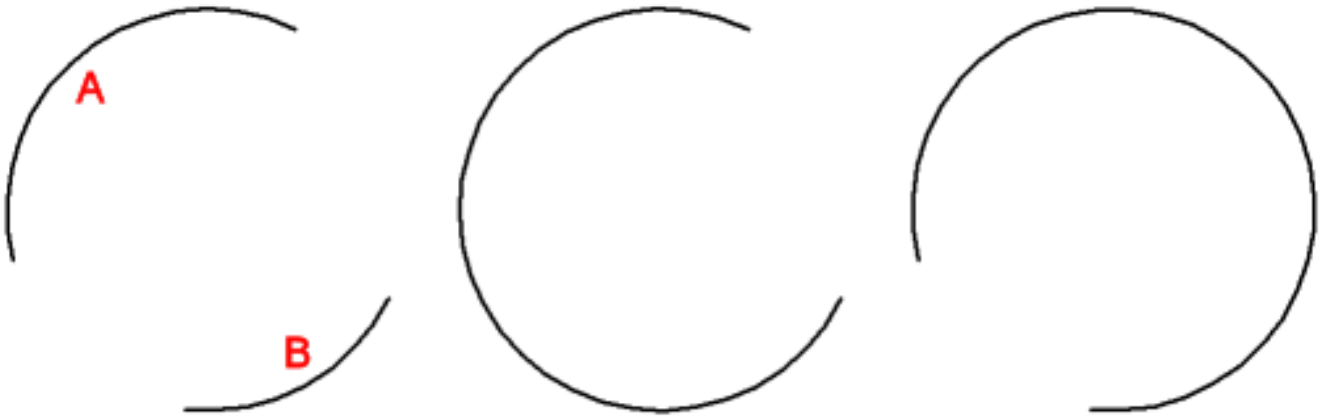
   - Click the **Join** tool button ( ) on the **Modify** toolbar.
   - Choose **Join** in the **Modify** menu.
   - Type **join** in the command bar, then press Enter.

   You are prompted: Select source entity or multiple entities to join at once:

2. Select the first arc. **You are prompted:** Select entities to join:

3. Select the second arc and press Enter. **You are prompted:** 2 entities joined into 1 arc. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/05JoinArcs.png}) **Joining two arcs: start situation (left) and result when selecting A first (middle) or B first (right).**

Figure 1.1085



NOTE	If more arcs share the same center point and radius, you can select them in step 3. Press Enter after selecting the last arc.

1.23.26 Breaking Entities

Command: BREAK (6.6.18) The **Break** command remove a portion of an entity, thus breaking it into two parts. You can break arcs, circles, ellipses, lines, polylines, rays and infinite lines. Breaking a circle converts it to an arc. A ray is broken into a ray and a line, an infinite line is broken into two rays. When breaking entities, you must specify two points for the break. By default, the point you use to select the entity becomes the first break point; however, you can use the First option to select a break point different from the one that selects the entity.

To break an entity

1. Do one of the following:

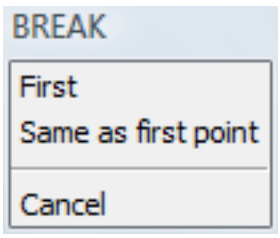
   - Click the **Break** tool button ( ) on the **Modify** toolbar.
   - Choose **Break** in the **Modify** menu.
   - Type **Break** or **BR** in the command bar, then press Enter.

   You are prompted: Select entity to break:

2. Click the entity you want to break. **You are prompted:** First break point/<Second break point>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/A

3. Click a second point on the entity. The entity is broken. The portion between the two break points is removed.

Figure 1.1086



1.23.27 Filleting Entities

Command: FILLET (6.14.7) The **Fillet** command connects two entities with an arc of a specified radius to create a rounded edge. You can fillet pairs of line segments, straight polyline segments, arcs, circles, rays, and infinite lines. You can also fillet parallel lines, rays, and infinite lines. When filleting a polyline, you can fillet multiple segments between two selected segments or you can fillet the entire polyline.

Filleting two entities or polyline segments

1. Do one of the following:

   - Click the **Fillet** tool button ( ) on the **Chamfer/Fillet** flyout of the **Modify** toolbar.
   - Choose **Fillet** in the **Modify** menu.
   - Type **fillet** or **F** in the command bar, then press Enter.

   You are prompted: Fillet (<current fillet settings>): Settings/Polyline/Radius/Trim/Undo/Multiple/<Select first entity>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sc

2. (option) Choose **Fillet Settings ...** in the prompt menu or type **S** and press Enter. Specify the **Fillet radius** in the **Settings** dialog window: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11Mod Close the Settings dialog window. **You are prompted:** Fillet (<current fillet settings>): Settings/Polyline/Radius/Trim/Und first entity>:

3. Select the first entity or polyline segment. The command window reads: Select second entity.

4. Select the second entity or polyline segment. The fillet is created.

Figure 1.1087

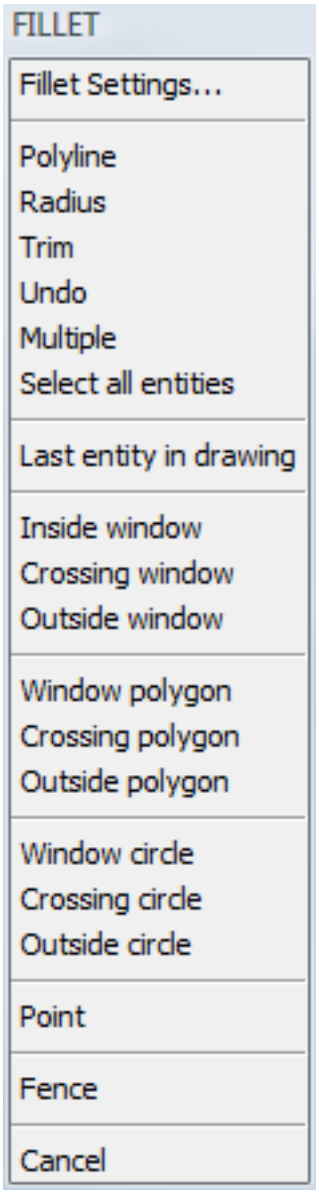
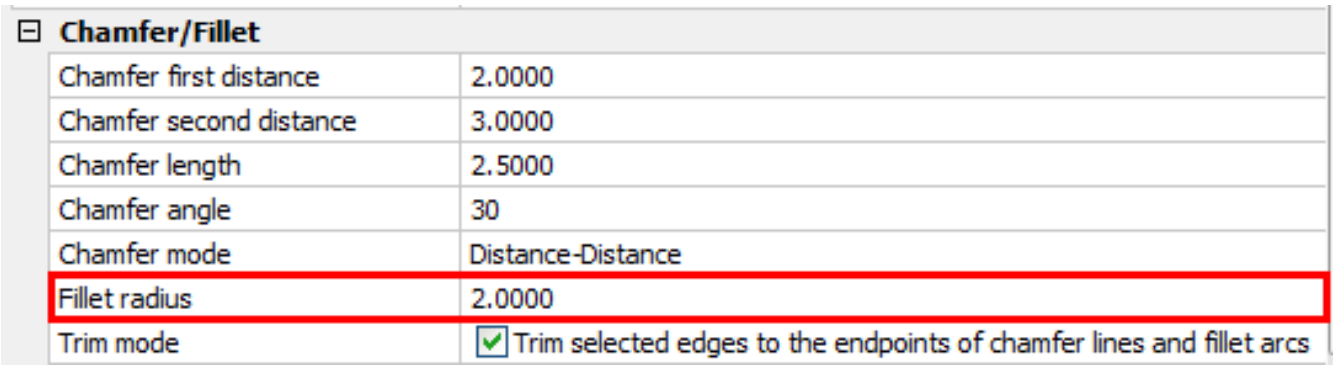


Figure 1.1088



Filleting all vertices of a polyline

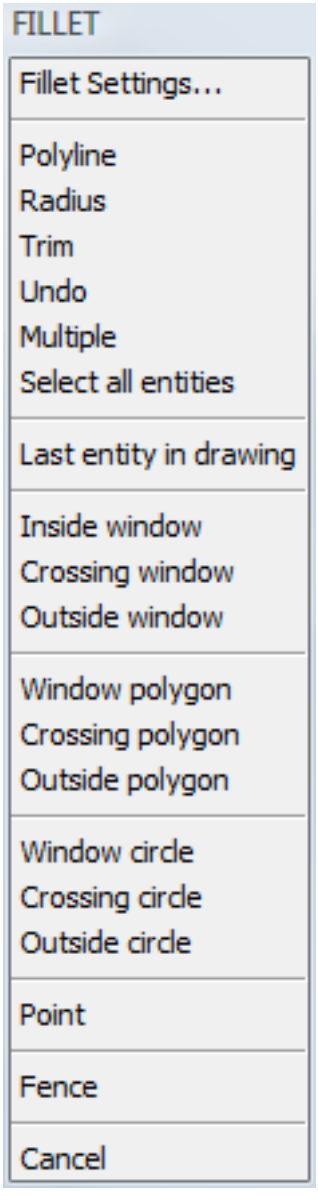
1. Do one of the following:

   - Click the **Fillet** tool button ( ) on the **Chamfer/Fillet** flyout of the **Modify** toolbar.
   - Choose **Fillet** in the **Modify** menu.
   - Type **fillet** or **F** in the command bar, then press Enter.

   You are prompted: Fillet (<current fillet settings>): Settings/Polyline/<Select first entity>: Pompt menu displays: (\vref{fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART

2. (option) Adjust the **Fillet Settings**.
3. Choose **Polyline** in the prompt menu or type **P** and press Enter. **You are prompted:** Select 2D polyline to chamfer:
4. Select a polyline. All vertices of the selected polyline are filleted.

Figure 1.1089



Filleting two parallel lines

1. Do one of the following:

   - Click the **Fillet** tool button ( ) on the **Chamfer/Fillet** flyout of the **Modify** toolbar.
   - Choose **Fillet** in the **Modify** menu.
   - Type **fillet** or **F** in the command bar, then press Enter.

   You are prompted: Fillet (<current fillet settings>): Settings/Polyline/Radius/Trim/Undo/Multiple/<Select first entity>:

2. Select the first entity (line or ray). The command window reads: Select second entity:

3. Select the second entity (line, ray or infinite line). The fillet is executed at the endpoint closest to the point were the first entity is selected. The length of the second entity is adjusted.

1.23.28 Chamfering Entities

Command:CHAMFER (6.9.9) The **Chamfer** command connects two non-parallel entities by extending or trimming them and then joining them with a line to create a beveled edge. In BricsCAD you can choose between two chamfer methods:

- **distance-distance** : specify how far to trim the entities back from their intersection
- **distance-angle** : specify the length of the chamfer and the angle it forms along the first entity.

The following entities can be chamfered: lines, polylines, rays and infinite lines. When chamfering a polyline, you can create a chamfer between two polyline segments or you can chamfer the entire polyline.

Chamfering using the distance-distance method

1. Do one of the following:

   - Click the **Chamfer** tool button ( ) on the **Chamfer/Fillet** flyout of the **Modify** toolbar.
   - Choose **Chamfer** in the **Modify** menu.
   - Type **chamfer** or **CHA** in the command bar, then press Enter.

   You are prompted: Chamfer (<current chamfer settings>): Settings/Polyline/Angle/Distance/mEthod/Trim/Undo/Multip
   first entity>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sc

2. (option) Choose **Chamfer Settings ...** in the prompt menu or type **S** and press Enter. In the **Settings** dialog window:

   - Specify the **Chamfer first distance**.
   - Set the **Chamfer second distance**.
   - Set the **Chamfer mode** to **Distance-Distance.**

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/06Cha
   Close the **Settings** dialog window. **You are prompted:** Chamfer (<current chamfer settings>): Settings/Polyline/Angle/Dis
   first entity>:

3. Select the first entity or polyline segment. The command window reads: Select second entity.
4. Select the second entity or polyline segment. The chamfer is created.

Figure 1.1090

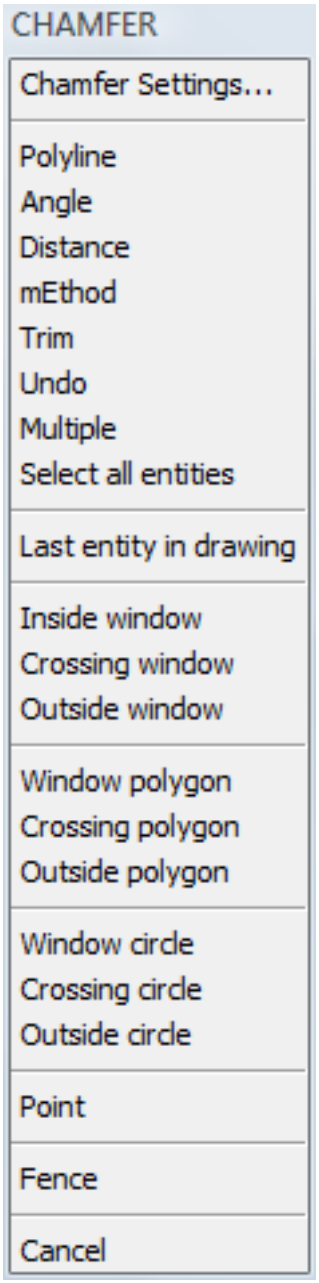
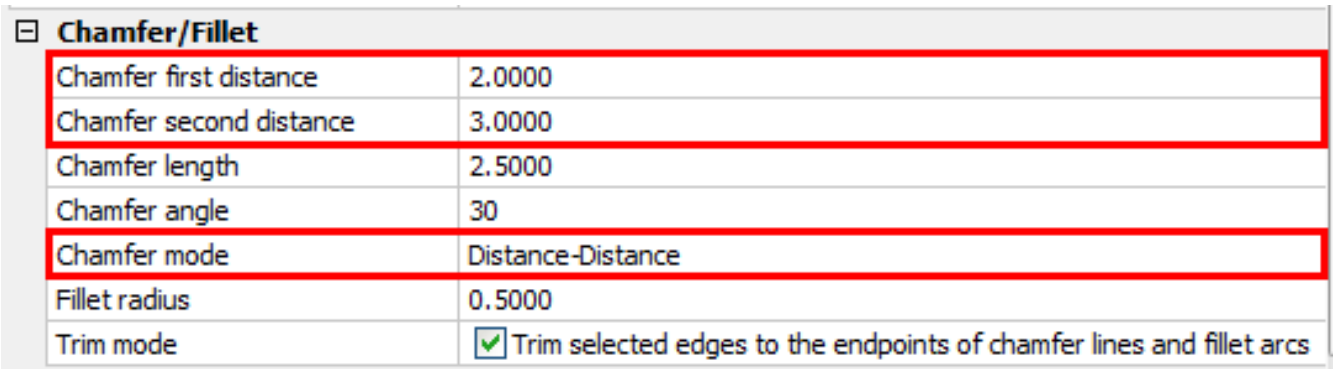


Figure 1.1091



Chamfering using the length-angle method

1. Do one of the following:

   - Click the **Chamfer** tool button ( ) on the **Chamfer/Fillet** flyout of the **Modify** toolbar.
   - Choose **Chamfer** in the **Modify** menu.
   - Type **chamfer** or **CHA** in the command bar, then press Enter.

   You are prompted: Chamfer (<current chamfer settings>): Settings/Polyline/<Select first entity>: A prompt menu displays.

2. Choose **Chamfer Settings ...** in the prompt menu or type **S** and press Enter.

3. In the **Settings** dialog window:

   - Specify the **Chamfer length**.
   - Set the **Chamfer angle**.
   - Set the **Chamfer mode** to **Length-Angle.**

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/06Ch

4. Close the Settings dialog window. **You are prompted:** Chamfer (<current chamfer settings>): Settings/Polyline/<Select first entity>:

5. Select the first entity or polyline segment. The command window reads: Select second entity.

6. Select the second entity or polyline segment. The chamfer is created.

Figure 1.1092

Chamfer/Fillet	
Chamfer first distance	0.5000
Chamfer second distance	0.5000
Chamfer length	2.5000
Chamfer angle	30
Chamfer mode	Length-Angle
Fillet radius	0.5000
Trim mode	☑ Trim selected edges to the endpoints of chamfer lines and fillet arcs

Chamfering all vertices of a polyline

1. Do one of the following:

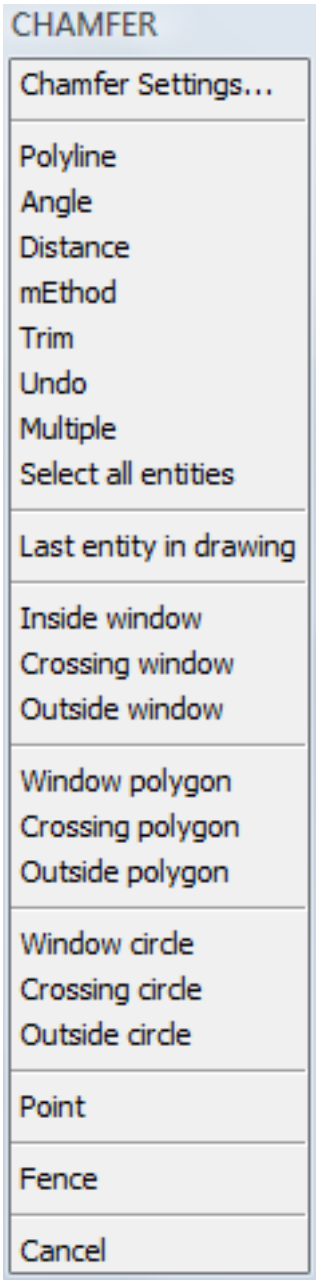
   - Click the **Chamfer** tool button ( ) on the **Chamfer/Fillet** flyout of the **Modify** toolbar.
   - Choose **Chamfer** in the **Modify** menu.
   - Type **chamfer** or **CHA** in the command bar, then press Enter.

   You are prompted: Chamfer (<current chamfer settings>): Settings/Polyline/<Select first entity>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEnti

2. (option) Adjust the **Chamfer Settings**.
3. Choose **Polyline** in the prompt menu or type **P** and press Enter. **You are prompted:** Select 2D polyline to chamfer:
4. Select a polyline. All vertices of the selected polyline are chamfered.

NOTE	When the chamfer method is **distance-angle** , the direction of the polyline defines which is the first entity of a vertex. See drawing rectangles (1.17.19) for more information about the direction of closed polylines.

Figure 1.1093



1.23.29 Converting an entity into a polyline

Command: PEDIT (6.24.20) Lines and arcs can be converted into single-segment polyline.

Converting an entity into a polyline

1. Do one of the following:

   - Click the **Edit Polyline** tool button ( ) on the **Modify** toolbar.
   - Choose **Edit Polyline** in the **Modify** menu.
   - Type **editpline** or **pedit** in the command window, then press Enter.

   You are prompted: Select polyline to edit. Select a line or arc. **You are prompted:** The entity selected is not a polyline. Turn it into one? <Y> A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU

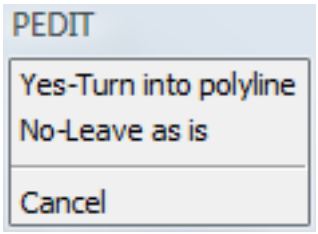
2. Do one of the following:

   - Press Enter to accept the default option.
   - Choose **Yes-Turn into polyline** in the prompt menu.

3. Press Enter to conclude the **Edit Polyline** command.

NOTE	The **Explode** tool turns a single segment polyline back into a line or arc.

Figure 1.1094



1.23.30 Opening and closing polylines

Command: PEDIT (6.24.20) When you close a polyline, a straight polyline segment is drawn from the last vertex of the polyline to the first vertex. Opening a polyline removes the closing segment. When you select a polyline for editing, the prompt menu displays either the **Open** or **Close** option, depending on whether the polyline you select is closed or open. The following entities are created as closed polylines:

- rectangles (1.17.19)
- polygons (1.17.20)
- donuts (1.17.22)
- revision clouds (1.17.24)

You can open or close a polyline using the **Edit Polyline** tool or in the BricsCAD Properties bar (1.12.4).

To close or open a polyline

1. Do one of the following:

   - Click the **Edit Polyline** tool button ( ) on the **Modify** toolbar.
   - Choose **Edit Polyline** in the **Modify** menu.
   - Type **editpline** or **pedit** in the command window, then press Enter.

   You are prompted: Select polyline to edit.

2. Click the polyline you want to close or open. **You are prompted:** Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/J
   Mode/Reverse/Spline/Taper/Width/Undo/<eXit>: A prompt menu displays.

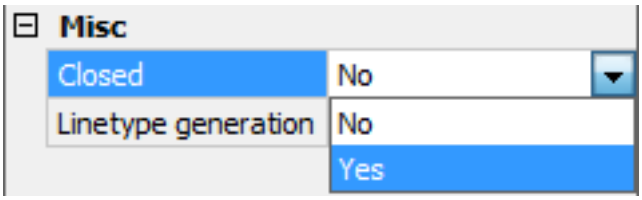
3. To close (or open) a polyline do one of the following:

   - Type **C** (or **O**) in the command bar, then press Enter.
   - Choose **Close** (or **Open**) in the prompt menu.

Opening and closing a polyline using the BricsCAD Properties bar

1. Select the polyline. The current properties of the selected polyline display in the BricsCAD Properties bar (1.12.4).
2. Select **Closed** in the **Misc** settings category in the **BricsCAD Properties** bar.
3. Click the down arrow and select **Yes** (or **No**). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Usr

Figure 1.1095



1.23.31 Joining Polylines

Commands: PEDIT (6.24.20) and JOIN (6.18.1) Using the **Join** option of the **Edit Polyline** tool you can add an arc, a line or a polyline entity to an existing open polyline, forming one continuous polyline entity. To join an entity to a polyline, that entity must already share an endpoint with an end vertex of the selected polyline. When you join an entity to a polyline, the **width** of the new polyline segment depends on the width of the original polyline and the type of entity you are joining to it:

- A line or an arc inherits the width from the polyline segment to which it is joined.
- A polyline joined to a tapered polyline retains its own width values.
- A polyline joined to a uniform-width polyline inherits the width from the polyline to which it is joined.

The **Join** command joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints. The type of the resulting entity depends on the types of the input entities and on their coplanarity.

To join an arc, line, or polyline to an existing polyline

1. Do one of the following:

   - Click the **Edit Polyline** tool button ( ) on the **Modify** toolbar.
   - Choose **Edit Polyline** in the **Modify** menu.
   - Type **editpline** or **pedit** in the command window, then press Enter.

   You are prompted: Select polyline to edit.

2. Select the parent polyline. **You are prompted:** Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>: A prompt menu displays.
3. Choose **Join** in the prompt menu or type **J** and press Enter. **You are prompted:** Select entities to join:
4. Select the entities to join. The number of already selected entities displays in the command bar.
5. Right click or press Enter to stop selecting entities and add the selection to the polyline.
6. Do one of the following:

   - Continue editing the selected polyline.
   - Choose **Exit** in the prompt menu or press Enter to conclude the **Edit Polyline** tool.

NOTE	You can select multiple entities in step 3 on condition they form one chain with the parent polyline.

1.23.32 Curving and decurving polylines

Command: PEDIT (6.24.20) The **Fit** or **Spline** options of the **Edit Polyline** tool convert a multi-segment polyline into a smooth curve. The **Fit** option creates a smooth curve connecting all the vertices. The **Spline** option computes a smooth curve that is pulled toward the vertices but passes through only the first and last vertices. The **Decurve** option removes **Fit** or **Spline** curves and arcs, leaving straight segments between the vertices.

To fit a curve to a polyline

1. Do one of the following:

   - Click the **Edit Polyline** tool button ( ) on the **Modify** toolbar.
   - Choose **Edit Polyline** in the **Modify** menu.
   - Type **editpline** or **pedit** in the command window, then press Enter.

   You are prompted: Select polyline to edit.

2. Select the polyline. **You are prompted:** Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>: A prompt menu displays.

3. Do one of the following:

   - Choose **Fit** in the prompt menu of type **F** and press Enter.
   - Choose **Spline** in the prompt menu or type **S** and press Enter.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en_US/11Modify/SplineOri.png})	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en_US/11Modify/SplineFit.png})	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en_US/11Modify/SplineSpline.png})
Original polyline	After applying Fit	After applying Spline

4. Do one of the following:

   - Continue editing the selected polyline.
   - Choose **Done** in the prompt menu or press Enter to conclude the **Edit Polyline** tool.

NOTES

- Use the **Decurve** option in step 3 of the above procedure to restore the original polyline.
- The **SPLINESEGS** (Spline Segments) settings variable controls the number of segments (lines or arcs) to be generated for spline-fit polylines.

Figure 1.1096

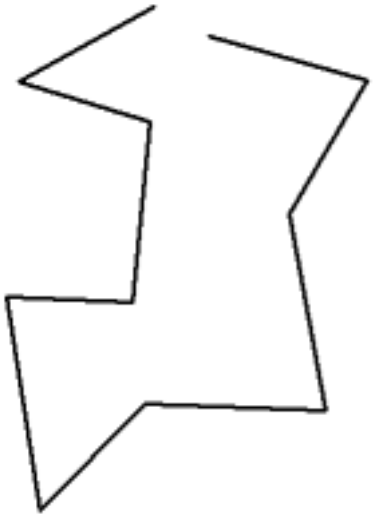


Figure 1.1097



Figure 1.1098



1.23.33 Editing polyline vertices

Commands:PEDIT (6.24.20)andPEDITEXT (6.24.21) TheEdit verticesoption of thePeditcommand modifies individual polyline vertices. When you select this option, the program switches into a special vertex editing mode and places an X on the first vertex. The X indicates the vertex you are editing. TheNextandPreviousoptions move the X to the next or previous vertex. You can edit only one vertex at a time. When editing vertices, you can modify the polyline in the following ways:

- Convert a straight polyline segment into a curve (1.23.33).
- Break a polyline into two separate polylines (1.23.33).
- Insert a new vertex in a polyline (1.23.33).
- Move a vertex in a polyline (1.23.33).
- Delete vertices in a polyline (1.23.33).
- Change the width of a polyline segment (1.23.33).

ThePeditextcommand is used in2D Editingcommand group of theQuad (1.12.5)cursor menu

Editing a polyline segment using the Quad

1. Move the cursor over the polyline. If currently ON, theTIPSwidget displays: (\vref {fig:C:/Dev/Task-2448/images/staticbri
   The available commands in the Quad apply to the entire polyline.
2. Press the Ctrl-key to switch to segment mode. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/U
   The available commands in the Quad apply to the segment under the cursor.
3. When theQuad cursor menu (1.12.5)displays, do one of the following:
   - Right click to execute the defaultQuadcommand.
   - Move the cursor to theQuadand move to theEditcommand group.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/07Qu
4. Select a command:
   - Add vertex: adds a vertex to the segment under the cursor.
   - Add vertex at end: adds as a new vertex at the end of the polyline. This option is only available if the cursor is on one of the endpoints of a polyline.
   - AdjustBulge (1.23.38): Modifies the bulge factor of the segment under the cursor.
   - Delete segment: deletes the segment under the cursor.
   - Stretch segment: moves the segment under the cursor.
   - Stretch vertex: moves the vertex closest to the cursor.

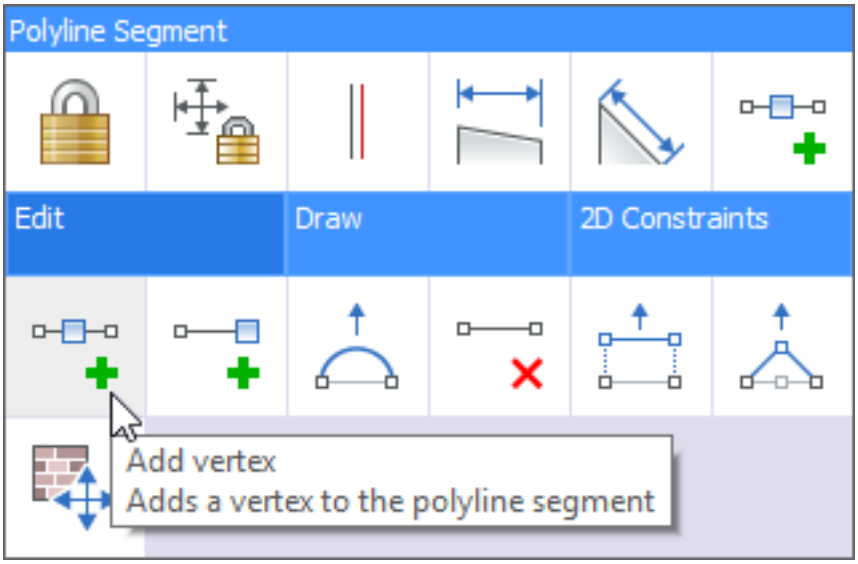
Figure 1.1099



Figure 1.1100



Figure 1.1101



Starting the polyline vertex editing mode

1. Do one of the following:

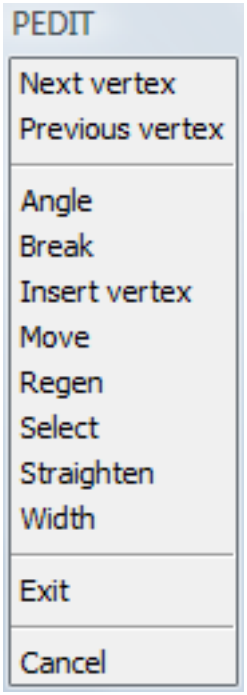
   - Click theEdit Polylinetool button ( ) on theModifytoolbar.
   - ChooseEdit Polylinein theModifymenu.
   - Typeeditplineorpeditin the command window, then press Enter.

   You are prompted: Select polyline to edit.

2. Select the polyline. **You are prompted:** Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>: A prompt menu displays.

3. ChooseEdit verticesin the prompt menu or typeEand press Enter. **You are prompted:** Next/Previous/Angle/Break/Insert/M TheEditplineprompt menu changes: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sou A triangle indicates the start point of the polyline. The first vertex of the polyline is selected.

Figure 1.1102



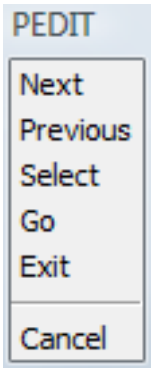
Convert a straight polyline segment into an arc

1. Start thepolyline vertex editing mode (1.23.33).
2. Select the start vertex of the segment you want to convert. **ChooseNext vertex/Previous vertex** in the prompt menu to select a vertex. TheXindicates the currently selected vertex.
3. ChooseAnglein the prompt menu or typeAand press Enter. The prompt menu closes. **You are prompted:** Included angle for segment (>0 is ccw, 0 is straight, <0 is cw) <current angle>:
4. Type a new included angle for the segment and press Enter. The direction of an arc segment is as follows:
   - positive angles: counter clockwise
   - negative angle: clockwise direction
   - 0° for straight segments
5. Choose**Exit**in the prompt menu or type**X**and press Enter to leave thePolyline vertex editing mode.
6. Do one of the following:
   - Continue editing the selected polyline.
   - ChooseExitin the prompt menu or press Enter to conclude theEdit Polylinetool.

Break a polyline into two separate polylines

1. Start thepolyline vertex editing mode (1.23.33).
2. Select the start vertex where you want to break the polyline. **ChooseNext vertex/Previous vertex** in the prompt menu to select a vertex. TheXindicates the currently selected vertex.
3. ChooseBreakin the prompt menu or typeBand press Enter. **You are prompted:** Next/Previous/Select/Go/eXit/<Next>: The prompt menu changes: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11M
4. (option) Do one of the following to select a second vertex:
   - **ChooseNext vertex/Previous vertex** in the prompt menu.
   - Choose**Select**in the prompt menu or type**S**and press Enter. **You are prompted** to select a vertex.
5. Choose </p> in the prompt menu or type**G**and press Enter. The polyline is broken into two polylines. If a second point is selected in step 4 the segment(s) between the selected vertices is (are) deleted.

Figure 1.1103



6. Choose**Exit**in the prompt menu or type**X**and press Enter to leave thePolyline vertex editing mode.
7. Do one of the following:
   - Continue editing the selected polyline.
   - Choose**Exit** in the prompt menu or press Enter to conclude theEdit Polylinetool.

Insert a new vertex in a polyline

1. Start thepolyline vertex editing mode (1.23.33).
2. Select the start vertex of the segment where you want to insert a vertex. **ChooseNext vertex/Previous vertex** in the prompt menu to select a vertex. TheXindicates the currently selected vertex.
3. Choose**Insert** vertexin the prompt menu or type**I**and press Enter. **You are prompted:** Location for new vertex:
4. Specify the location for the new vertex. The new vertex is inserted.
5. Choose**Exit**in the prompt menu or type**X**and press Enter to leave thePolyline vertex editing mode.
6. Do one of the following:
   - Continue editing the selected polyline.
   - Choose**Exit**in the prompt menu or press Enter to conclude theEdit Polylinetool.

Move a vertex in a polyline

1. Start thepolyline vertex editing mode (1.23.33).
2. Select the vertex you want to move. **ChooseNext vertex/Previous vertex** in the prompt menu to select a vertex. TheXindicates the currently selected vertex.
3. Choose**Move**in the prompt menu or type**M**and press Enter. **You are prompted:** New location for vertex.
4. Specify the new location for the selected vertex. The vertex is moved.
5. Choose**Exit**in the prompt menu or type**X**and press Enter to leave thePolyline vertex editing mode.
6. Do one of the following:
   - Continue editing the selected polyline.
   - Choose**Exit**in the prompt menu or press Enter to conclude theEdit Polylinetool.

Delete vertices in a polyline

1. Start thepolyline vertex editing mode (1.23.33).
2. Select the vertex before the first vertex you want to delete. **ChooseNext vertex/Previous vertex** in the prompt menu to select a vertex. TheXindicates the currently selected vertex.
3. Choose**Straighten**in the prompt menu or type**S**and press Enter. **You are prompted:** Straighten: Next/Previous/Select/Go/
4. Do one of the following to select the vertex after the last vertex you want to delete:

- **ChooseNext vertex/Previous vertex** in the prompt menu.
- Choose**Select**in the prompt menu or type**S**and press Enter. **You are prompted** to select a vertex.

5. Choose**Go**in the prompt menu or type**G**and press Enter. The vertices between the selected vertices are removed. A straight segment is drawn between the selected vertices.
6. Choose**Exit**in the prompt menu or type**X**and press Enter to leave thePolyline vertex editing mode.
7. Do one of the following:
   - Continue editing the selected polyline.
   - Choose**Exit**in the prompt menu or press Enter to conclude theEdit Polylinetool.

Change the width of a polyline segment

1. Start thepolyline vertex editing mode (1.23.33).
2. Select the start vertex of the segment you want to change the width of. **ChooseNext vertex/Previous vertex** in the prompt menu to select a vertex. The**X**indicates the currently selected vertex.
3. Enter starting width <current width>:
4. Do one of the following:
   - Type the new width in the command bar and press Enter.
   - Click to define the width graphically.

   You are prompted: Enter ending width <current width>:
5. Do one of the following:
   - Type the new width in the command bar and press Enter.
   - Click to define the width graphically.
6. Choose**Exit**in the prompt menu or type**X**and press Enter to leave thePolyline vertex editing mode.
7. Do one of the following:
   - Continue editing the selected polyline.
   - Choose**Exit**in the prompt menu or press Enter to conclude theEdit Polylinetool.

NOTE	The new width is applied when you conclude theEdit Polylinetool in step 6.

1.23.34 Setting the Linetype generation mode

Command: PEDIT (6.24.20) The **Linetype mode** option of the **Edit Polyline** tool lets you change the way how a dashed linetype is applied to a multi-segment polyline.

To set the Linetype mode

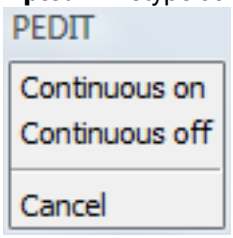
1. Do one of the following:

   - Click the **Edit Polyline** tool button ( ) on the **Modify** toolbar.
   - Choose **Edit Polyline** in the **Modify** menu.
   - Type **editpline** or **pedit** in the command window, then press Enter.

   You are prompted: Select polyline to edit.

2. Select the polyline. **You are prompted:** Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>: A prompt menu displays.

3. Choose **Linetype** mode in the prompt menu or type **L** and press Enter. **You are prompted:** Linetype continuous along polyline: ON/OFF <current setting>: The **Editpline** prompt menu changes:



4. Do one of the following:

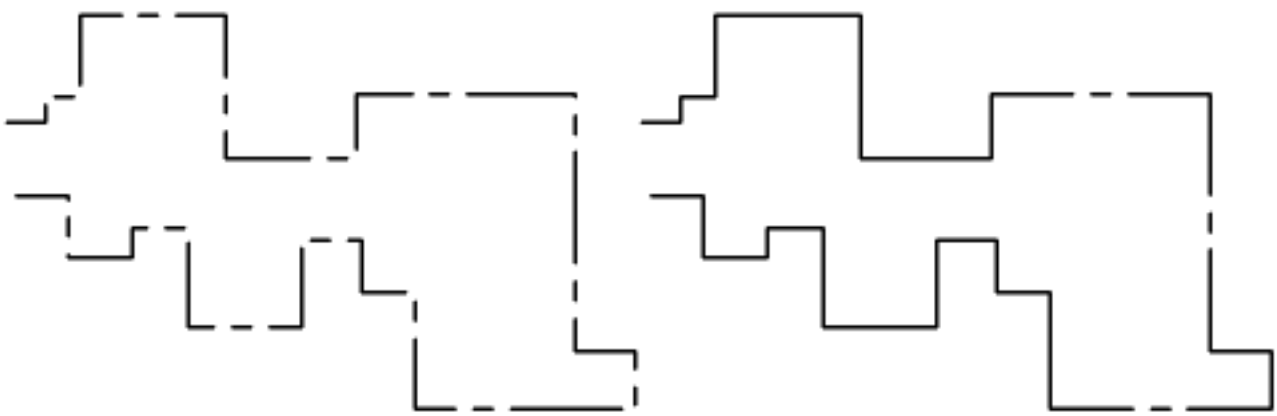
   - Choose **Continuous on** or **Continuous off** in the prompt box.
   - Type **ON** or **OF** in the command bar, then press Enter.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/00EntityCrea
   The same polyline with Continuous On (left) and Continuous Off (right)

5. Do one of the following:

   - Continue editing the selected polyline.
   - Choose **Cancel** in the prompt menu or press Enter or right click to conclude the **Edit Polyline** tool.

Figure 1.1104



1.23.35 Changing the polyline width

Command: PEDIT (6.24.20) The **Width** option of the **Edit Polyline** tool applies a uniform width to the entire entity, while the **Taper** option tapers the polyline uniformly along its entire length.

To apply a uniform width to a polyline

1. Do one of the following:

   - Click the **Edit Polyline** tool button ( ) on the **Modify** toolbar.
   - Choose **Edit Polyline** in the **Modify** menu.
   - Type **Editpline** or **Pedit** in the command window, then press Enter.

   You are prompted: Select polyline to edit.

2. Select the polyline. **You are prompted:** Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>: A prompt menu displays.

3. Choose **Width** in the prompt menu or type **W** and press Enter. **You are prompted:** Enter new width for all segments:

4. Type a new width in the command bar and press Enter. The new width is applied to the entire polyline.

5. Do one of the following:

   - Continue editing the selected polyline.
   - Choose **Cancel** in the prompt menu or press Enter or right click to conclude the **Edit Polyline** tool.

To taper a polyline uniformly along its length

1. Do one of the following:

   - Click the **Edit Polyline** tool button ( ) on the **Modify** toolbar.
   - Choose **Edit Polyline** in the **Modify** menu.
   - Type **Editpline** or **Pedit** in the command window, then press Enter.

   You are prompted: Select polyline to edit.

2. Select the polyline. **You are prompted:** Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>: A prompt menu displays.

3. Choose **Taper** in the prompt menu or type **T** and press Enter. **You are prompted:** Enter new starting polyline width <current starting width>:

4. Type a new starting width in the command bar and press Enter. **You are prompted:** Enter new ending polyline width <current ending width>:

5. Type a new ending width in the command bar and press Enter. The polyline is tapered uniformly along its length.

6. Do one of the following:

   - Continue editing the selected polyline.
   - Choose **Cancel** in the prompt menu or press Enter or right click to conclude the **Edit Polyline** tool.

1.23.36 Exploding entities

Commands: EXPLODE (6.13.13) and TXTEXP (6.29.27) The **Explode** command converts complex entities, such as blocks, polylines, solids or dimensions into their component parts. Exploding a polyline or dimension reduces it to a collection of individual line and arc entities that you can then modify individually. Blocks are converted to the individual entities, possibly including other, nested blocks that composed the original entity. Multiline texts explode into single line text entities. In general exploding entities will have no visible effect in the drawing, except for:

- If the original polyline had a width, the width information is lost when you explode it. The resulting lines and arcs follow the centerline of the original polyline.
- If you explode a block containing attributes, the attributes are lost, but the original attribute definitions remain. Colors and linetypes assigned **BYBLOCK** may appear different after exploding an entity, because they will adopt the default color and linetype.

The **TxtExp** command explodes text into polylines (short for "text explode").

To explode entities

1. Do one of the following:

   - Click the **Explode** tool button () on the **Modify** toolbar.
   - Choose **Explode** in the **Modify** menu.
   - Type **Explode** in the command bar, then press Enter.

   The command window reads: Select entities to explode.

2. Select the entities (1.23.6), then right click or press Enter.

NOTE	If you select the entities first, then launch the**Explode** tool, the selected entities are exploded immediately.

1.23.37 Creating Regions

Command: REGION (6.26.18) The **Region** command converts closed entities into a solid entity. You can create regions from closed entities, such as polylines, polygons, circles, ellipses, closed splines and donuts. Creating regions typically has no visible effect on a drawing. However, if the original entity had a width or lineweight, that information is lost when you create the region.

To create regions

1. Do one of the following:

   - Click the **Region** tool button ( ) in the **Draw** toolbar.
   - Choose **Region** in the **Draw** menu.
   - Type **Region** in the command bar, then press Enter.

   You are prompted: Select objects:

2. Select the entities (1.23.6), then right click or press Enter. The command bar displays how many regions have been created.

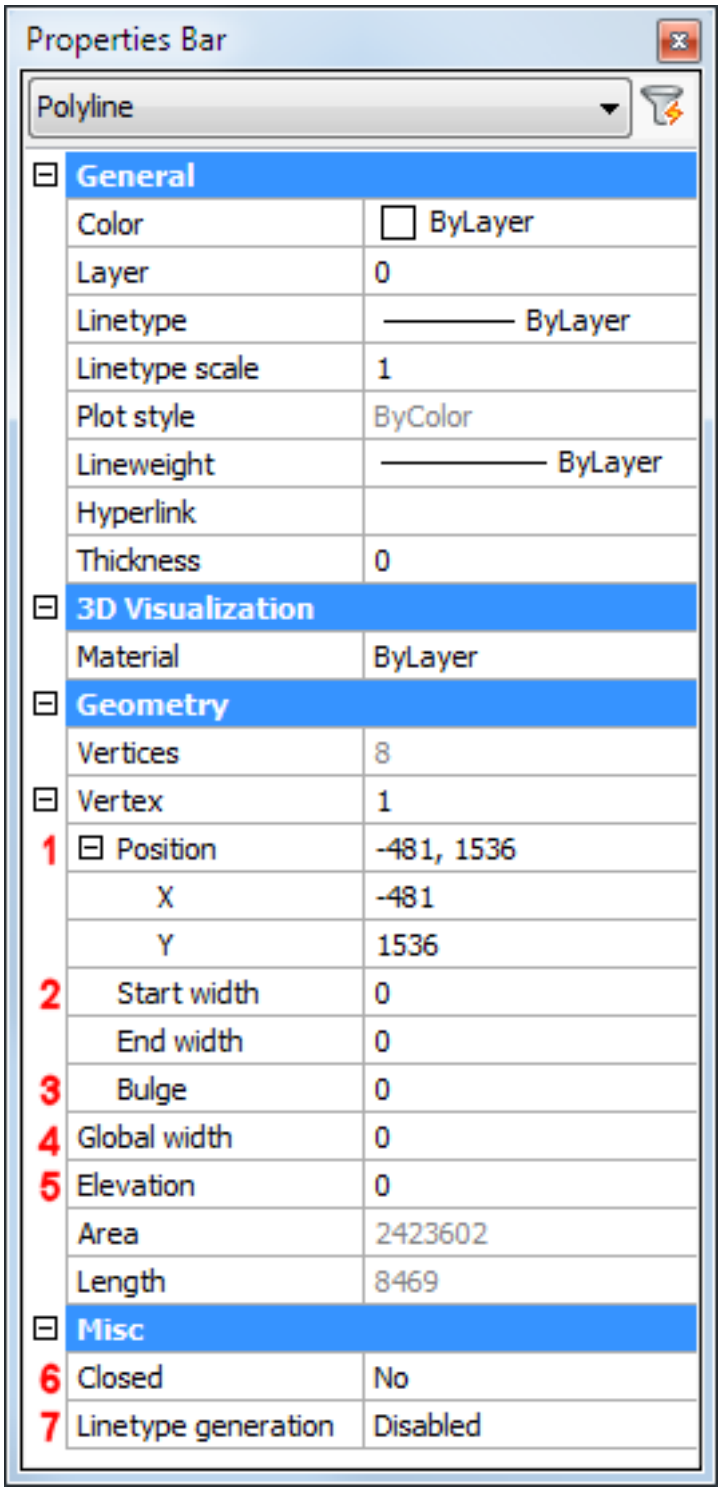
1.23.38 Editing polylines in the Properties bar

When you select polyline, its current properties display in the BricsCAD Properties bar (1.12.4). In the **BricsCAD Properties bar** the following polyline properties can be edited:

- Move vertices (1)
- Change the width of polyline segments (2)
- Convert straight segments into a curve (3)
- Change the global width (4)
- Change the Elevation (5)
- Open / close the polyline (6)
- Change the Linetype generation mode (7)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/08EditPline

Figure 1.1105



To move polyline vertices

1. Select the polyline.
2. Click the **Vertex** field in the **Properties bar**, then press the arrow buttons to select the vertex you want to move.
   A **X** indicates the currently selected vertex in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/er
3. Do one of the following:
   - Adjust the **X-** and/or **Y-field** and press Enter.
   - Click the **Vertex Position** field and press the **Position** button to move the vertex in the drawing.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/08Edi
4. Continue editing the polyline or press the Esc key to stop.

Figure 1.1106

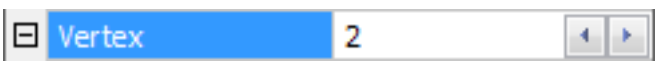
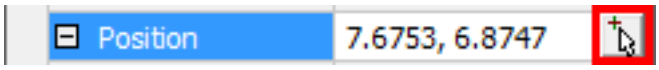


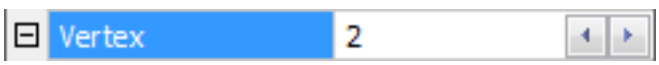
Figure 1.1107



To change the width of polyline segments

1. Select the polyline.
2. Click the **Vertex** field in the **Properties bar**, then press the arrow buttons to select the start vertex of the segment you want to modify. A **X** indicates the currently selected vertex in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/08EditPlinePropertiesBarVer
3. Type a new width in the **Start width** field and press Enter.
4. Type a new width in the **End width** field and press Enter.
5. Continue editing the polyline or press the Esc key to stop.

Figure 1.1108



To convert straight segments into a curve

1. Select the polyline.
2. Click the **Vertex** field in the **Properties bar**, then press the arrow buttons to select the start vertex of the segment you want to modify. A **X** indicates the currently selected vertex in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11ModifyingEntities/ART/08EditPlinePropertiesBarVer
3. Type a new value in the Bulge field and press Enter. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
4. Continue editing the polyline or press the Esc key to stop.

NOTE

The bulge is the tangent of one fourth the included angle for an arc segment. A positive bulge creates a counter clockwise arc, while a negative bulge creates an arc with a clockwise direction. Straight segments have a zero bulge. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/11

Bulge = 2H/D Bulge = tg α / 4

Figure 1.1109

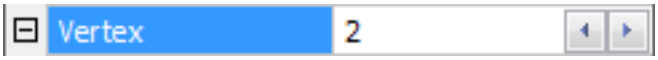


Figure 1.1110

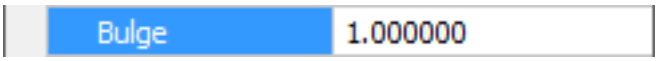
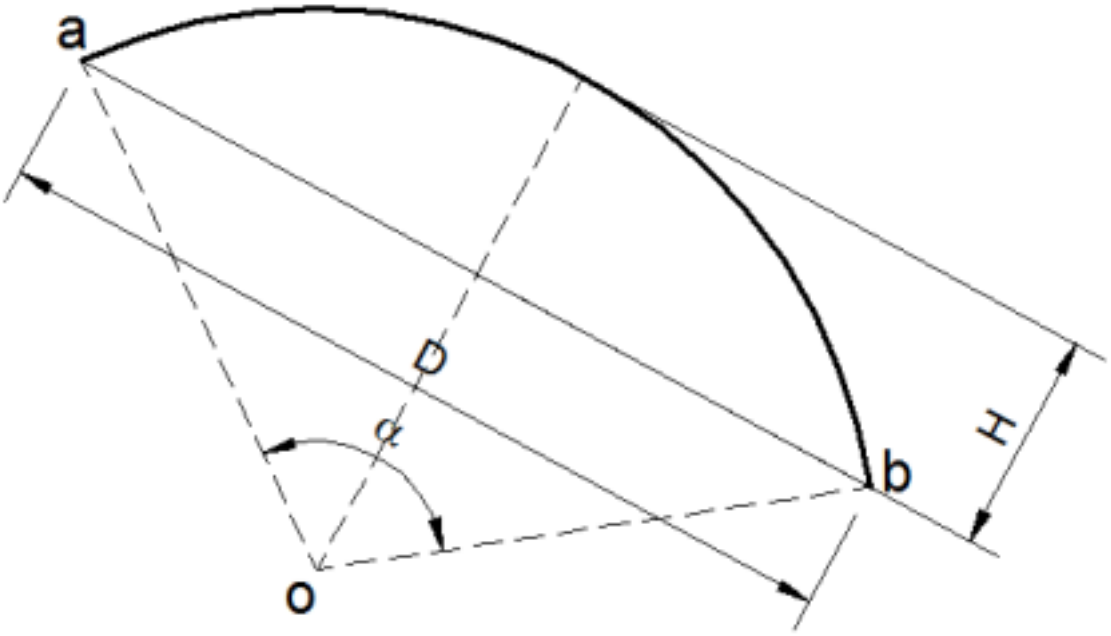


Figure 1.1111



1.23.39 Measuring Entities

Command: MEASURE (6.21.13) The **Measure** command places markers - points or blocks - at a specified interval along the length or circumference of an entity. The Measure tool starts placing markers at the closes endpoint to where you select the entity. You can measure lines, polylines, arcs, circles, ellipses, elliptical arcs and splines. You cannot measure rays and infinite lines. You measure a circle along its circumference starting from the **Angle Base** value as defined in the **Settings** dialog. If **Angle Base** is set to zero, a circle is measured starting at 3 o'clock (east).

To measure an entity using points

1. Do one of the following:

   - Click the **Measure** tool button ( ) on the **Modify** toolbar.
   - Choose **Measure** in the **Modify** menu.
   - Type **measure** in the command bar, then press Enter.

   You are prompted: Select entity to measure:

2. Click the entity you want to measure. **You are prompted:** Block/<Segment length>:

3. To define the measurement interval, do one of the following:

   - Type the measurement interval and press Enter.
   - Specify two points in the drawing.

   Points are placed along the selected entity at the specified interval.

NOTE	It might be necessary to adjust the Point display mode (1.17.25) settings to display the points correctly.

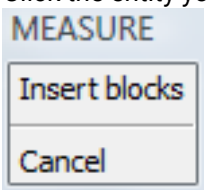
To place blocks a specified interval along an entity

1. Do one of the following:

   - Click the **Measure** tool button ( ) on the **Modify** toolbar.
   - Choose **Measure** in the **Modify** menu.
   - Type **measure** in the command bar, then press Enter.

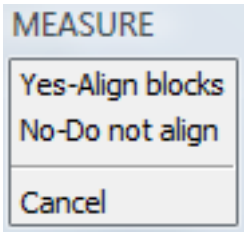
   You are prompted: Select entity to measure:

2. Click the entity you want to measure. **You are prompted:** Block/<Segment length>: A prompt menu displays:



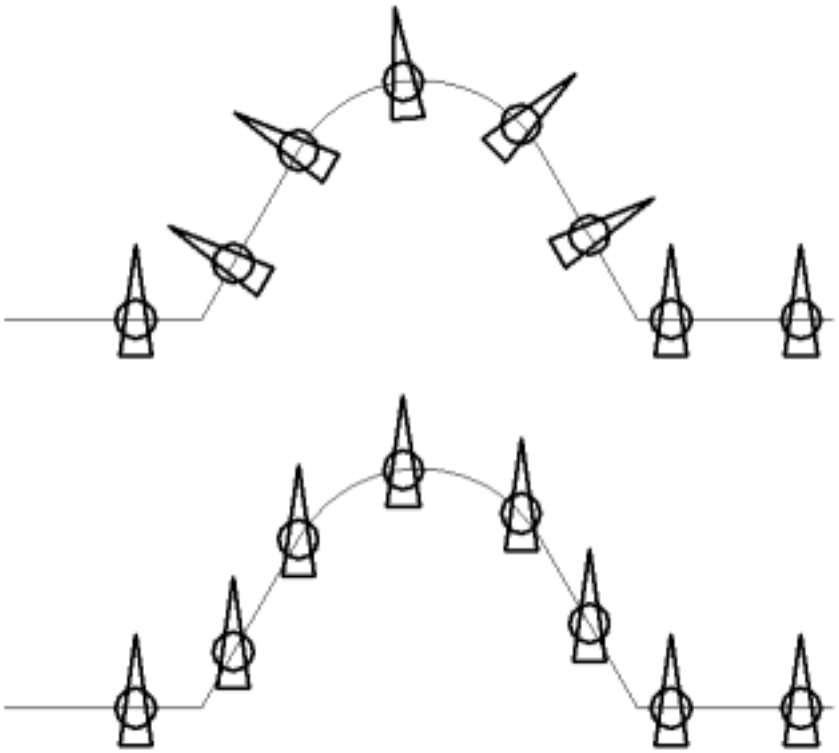
3. Choose **Insert blocks** in the prompt menu or type B and press Enter. **You are prompted:** Name of block to insert:

4. Type the name of the block in the command bar and press Enter. **You are prompted:** Align blocks with entity? <Y>: A prompt menu displays:



5. Do one of the following:

Figure 1.1112



- Choose **Yes-Align blocks** in the prompt menu or press Enter to align the blocks with the selected entity.
- Choose **No-Do not align** in the prompt menu or type N and press Enter to place the blocks not rotated.

You are prompted: Segment length:

6. To define the measurement interval, do one of the following:

   - Type the measurement interval and press Enter.
   - Specify two points in the drawing.

   The blocks are placed along the selected entity at the specified interval. (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
   Blocks aligned (top) or not aligned (bottom)

1.23.40 Dividing Entities

Command: DIVIDE (6.10.33) The **Divide** command places markers - points or blocks - along a selected entity. The markers evenly divide the entity into the specified number of equal parts. You can divide lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

To divide an entity using points

1. Do one of the following:

   - Click the **Divide** tool button ( ) on the **Modify** toolbar.
   - Choose **Divide** in the **Modify** menu.
   - Type **divide** in the command bar, then press Enter.

   You are prompted: Select entity to divide:

2. Click the entity you want to divide. **You are prompted:** Blocks/<Number of Segments>:

3. Type the number of segments in the command bar and press Enter. Points are placed along the selected entity to divide the entity.

NOTE	It might be necessary to adjust the Point display mode (1.17.25) settings to display the points correctly.

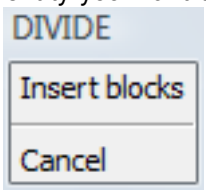
To divide an entity using blocks

1. Do one of the following:

   - Click the **Divide** tool button ( ) on the **Modify** toolbar.
   - Choose **Divide** in the **Modify** menu.
   - Type **divide** in the command bar, then press Enter.

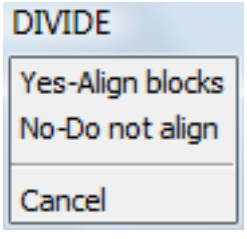
   You are prompted: Select entity to divide:

2. Click the entity you want to divide. **You are prompted:** Blocks/<Number of Segments>: A prompt menu displays:



3. Choose **Insert blocks** in the prompt menu or type B and press Enter. **You are prompted:** Name of block to insert:

4. Type the name of the block in the command bar and press Enter. **You are prompted:** Align blocks with entity? <Y>: A prompt menu displays:

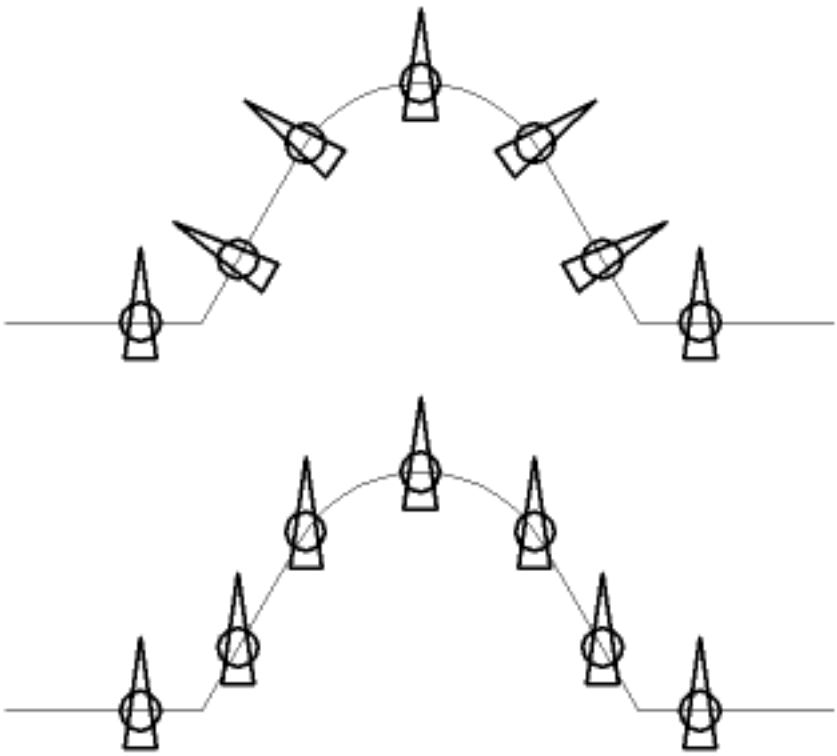


5. Do one of the following:

   - Choose **Yes-Align blocks** in the prompt menu or press Enter to align the blocks with the selected entity.
   - Choose **No-Do not align** in the prompt menu or type N and press Enter to place the blocks not rotated.

   The blocks are placed along the selected entity to divide the entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscon
   Blocks aligned (top) or not aligned (bottom)

Figure 1.1113

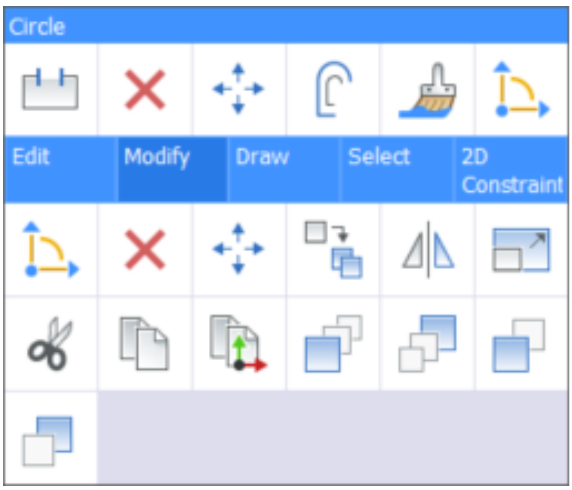


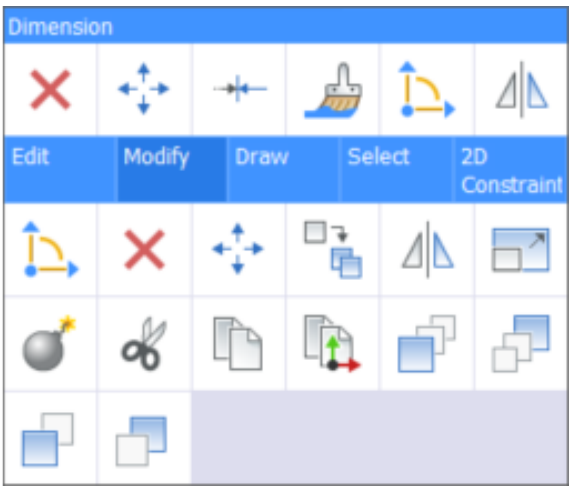
1.23.41 Entity Manipulation

Traditional Manipulation Tools

You can access most of the manipulation tools using typical methods such as the command line. They're also available on the Modify tab of the Quad when you select or hover over an entity. Unlike Edit tools which can vary significantly for different entities, the Modify tools are relatively consistent. For example, they include Erase, Move, Copy and tools to specify draw order. Some entities such as dimensions, hatches, and blocks, also include the Explode tool. Since you're probably familiar with all the traditional modify tools from your AutoCAD® experience, I'll focus on the one that's unique to BricsCAD, the Manipulator! (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BC

Figure 1.1114





Manipulator

The Manipulator is somewhat like the 3D Gizmos in AutoCAD (not available in AutoCAD LT). In BricsCAD the Manipulator is useful for both 2D and 3D entity manipulation. Therefore, it's available in all BricsCAD editions including Classic. You can access the Manipulator from the Quad when objects are selected. It's also accessible via other methods including the MANIPULATE command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Mo
With this single Manipulator tool, you can move, rotate, scale, copy and even mirror the selected entities. And, you can easily do so while constraining them along a specific axis or plane. Right-click on the Manipulator to access additional controls including a toggle to switch the Arrowhead behavior between Mirror and Scale. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/EntityManipulationManipulator2.p

Figure 1.1115

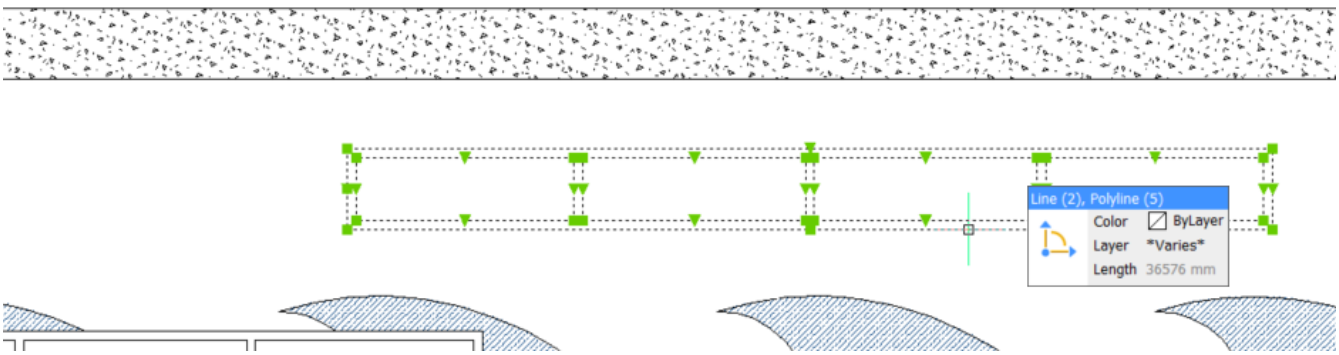
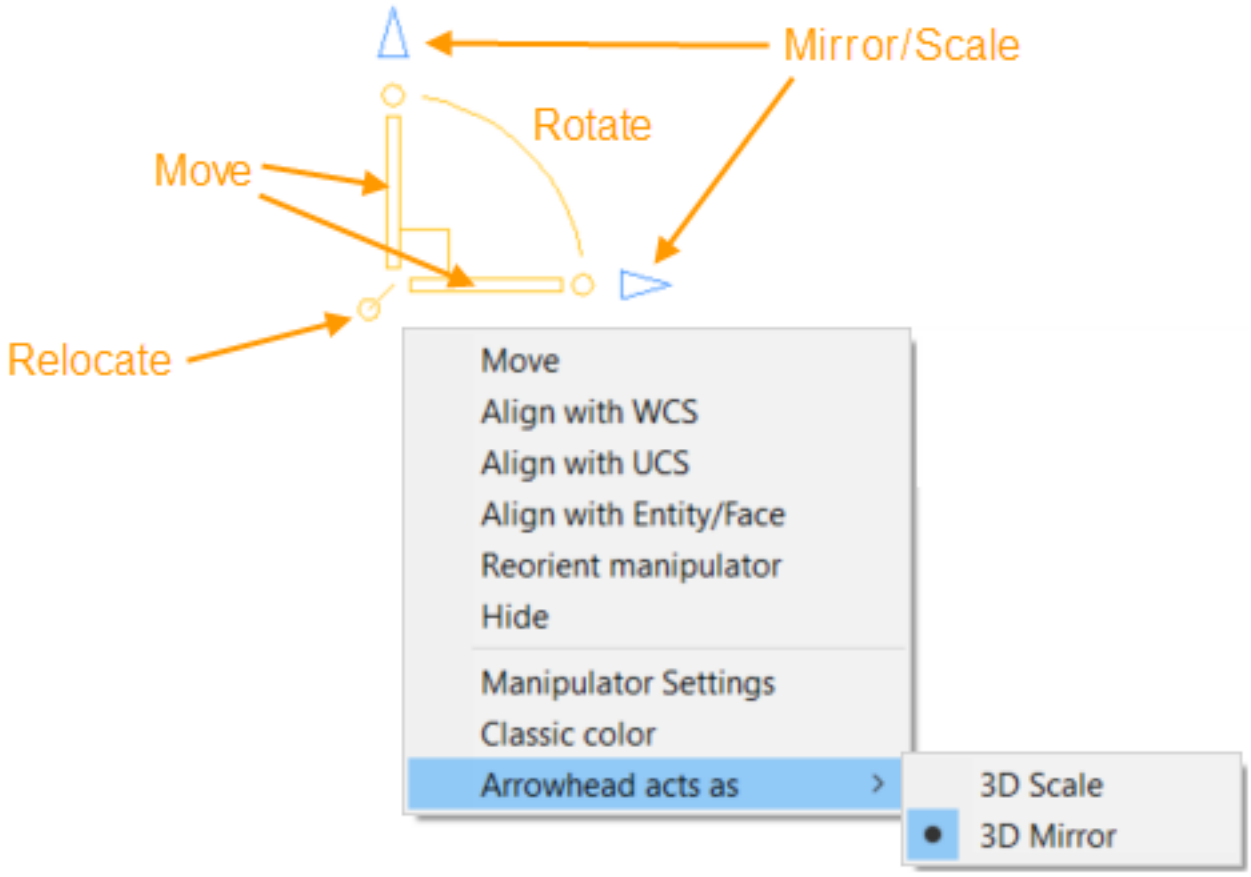
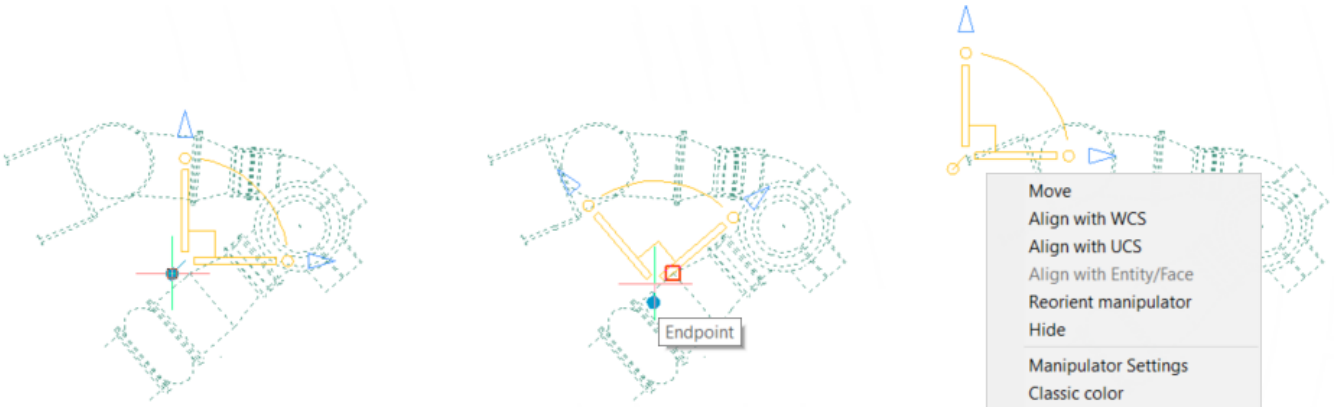


Figure 1.1116



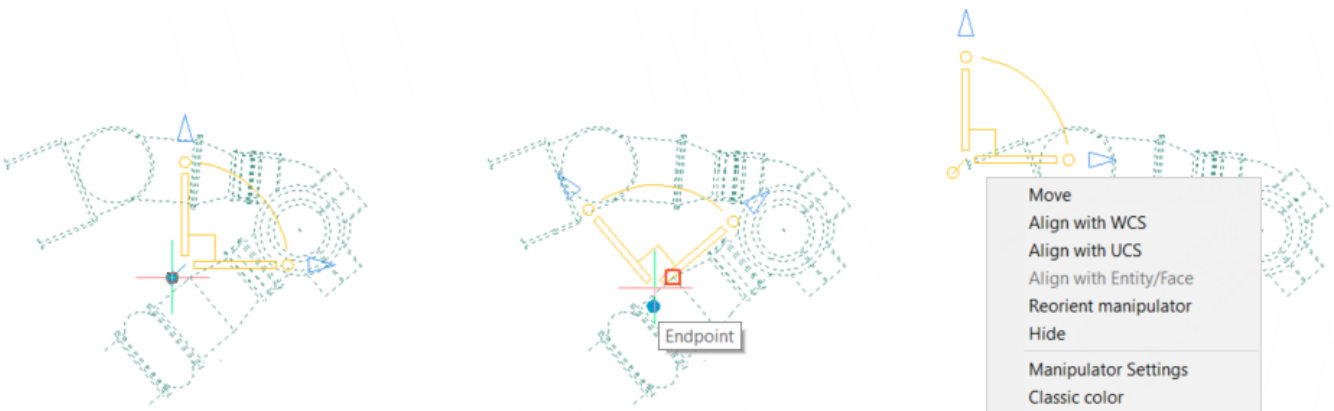
Reorient the Manipulator By default, the Manipulator displays at the center of the selection set. To move it, you can select the relocate grip and pick a new base point for the manipulation. As you move it across the drawing, it can automatically align with geometry enabling you to manipulate along any relevant axis. The default alignment is with the current UCS. Options on the right-click menu offer additional controls to reorient and align the Manipulator. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/EntityManipulationReorienti

Figure 1.1117



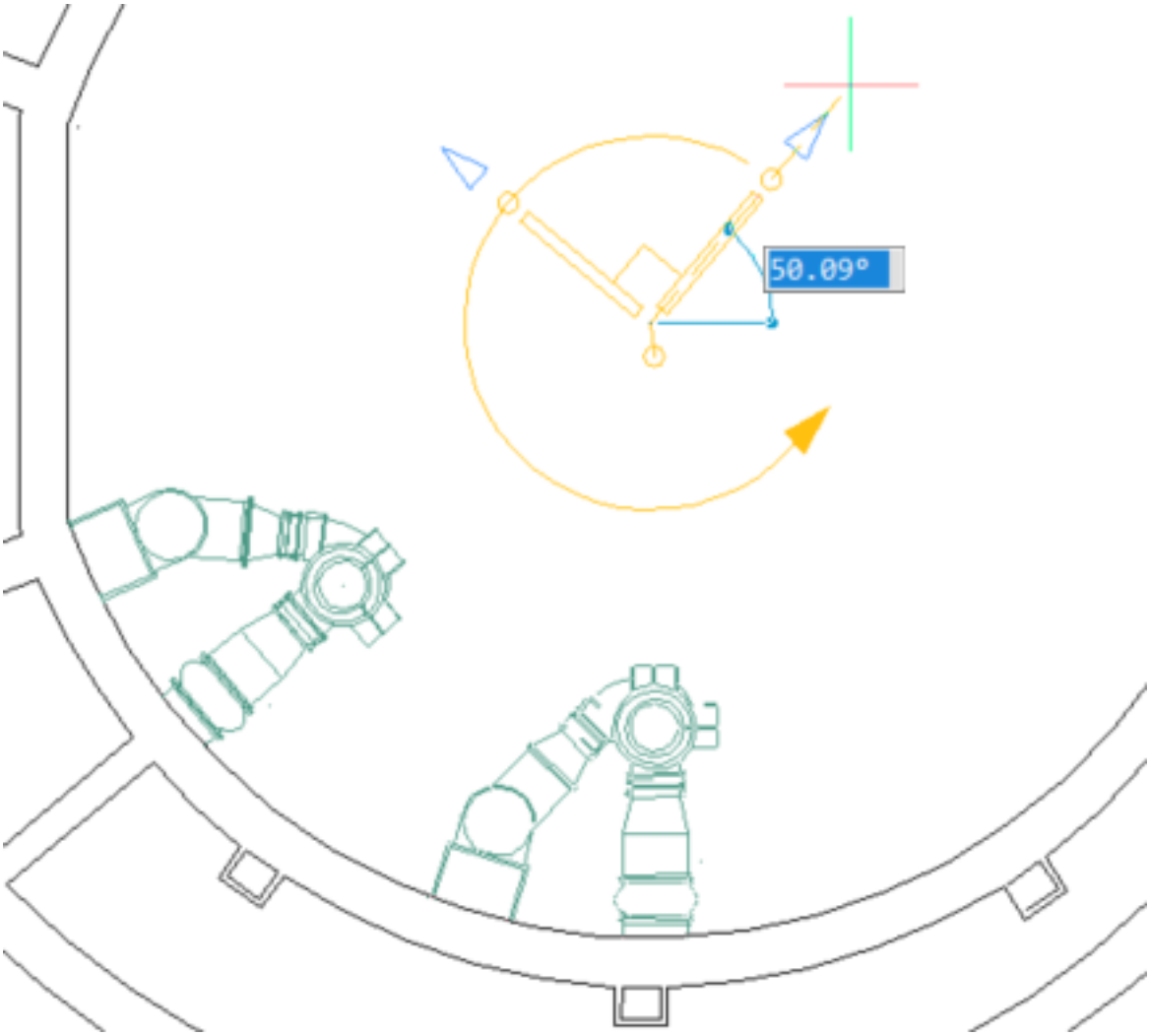
Move With the Manipulator properly oriented, you can click an axis or plane to move the selected entities. Their movement is automatically constrained to the specified axis or plane as you pick the second point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/EntityManipulationMoving.png})

Figure 1.1118



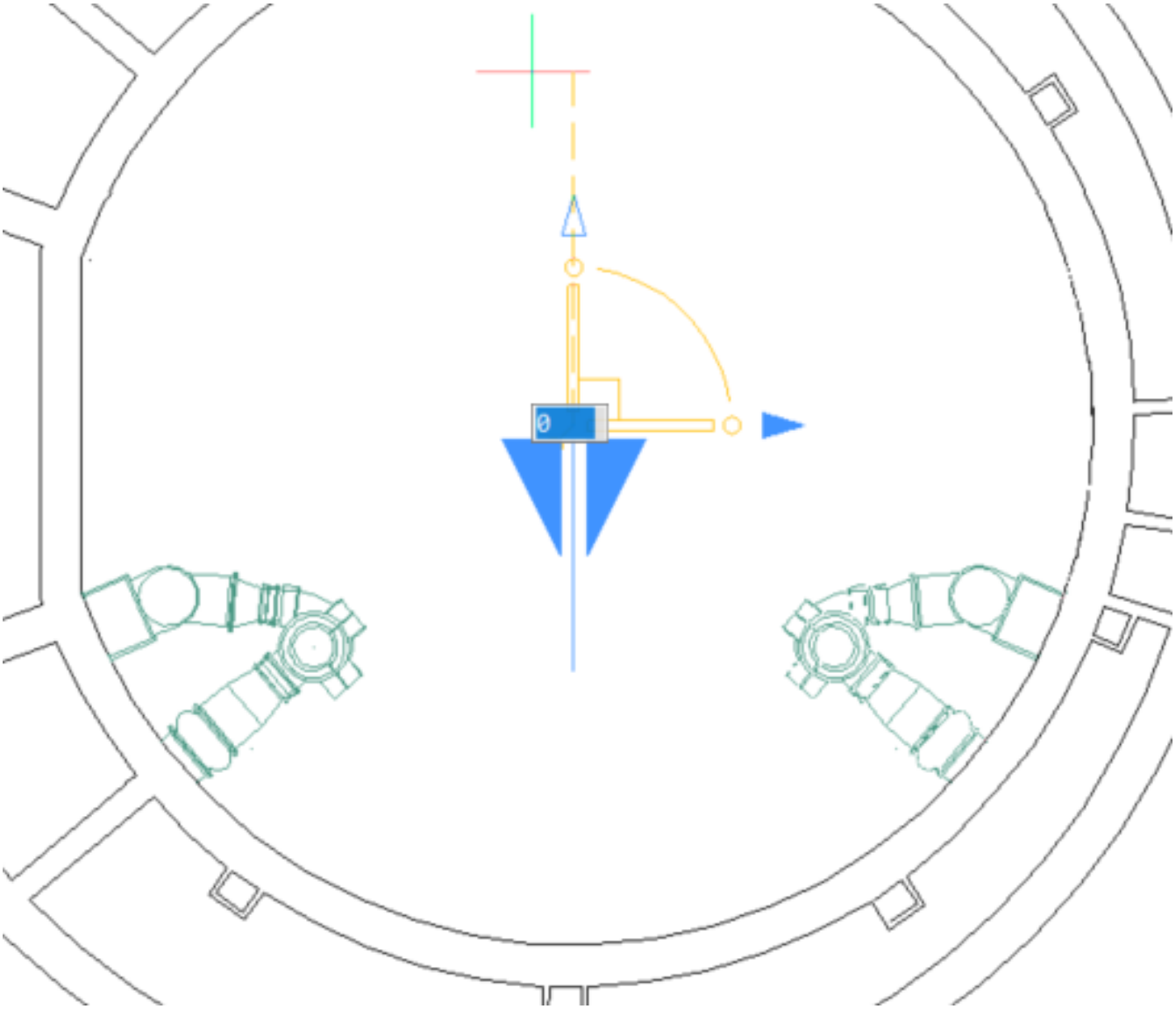
Rotate To rotate the entities around the basepoint of the manipulator, select the rotation arc. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/EntityManipulationRotating.png})

Figure 1.1119



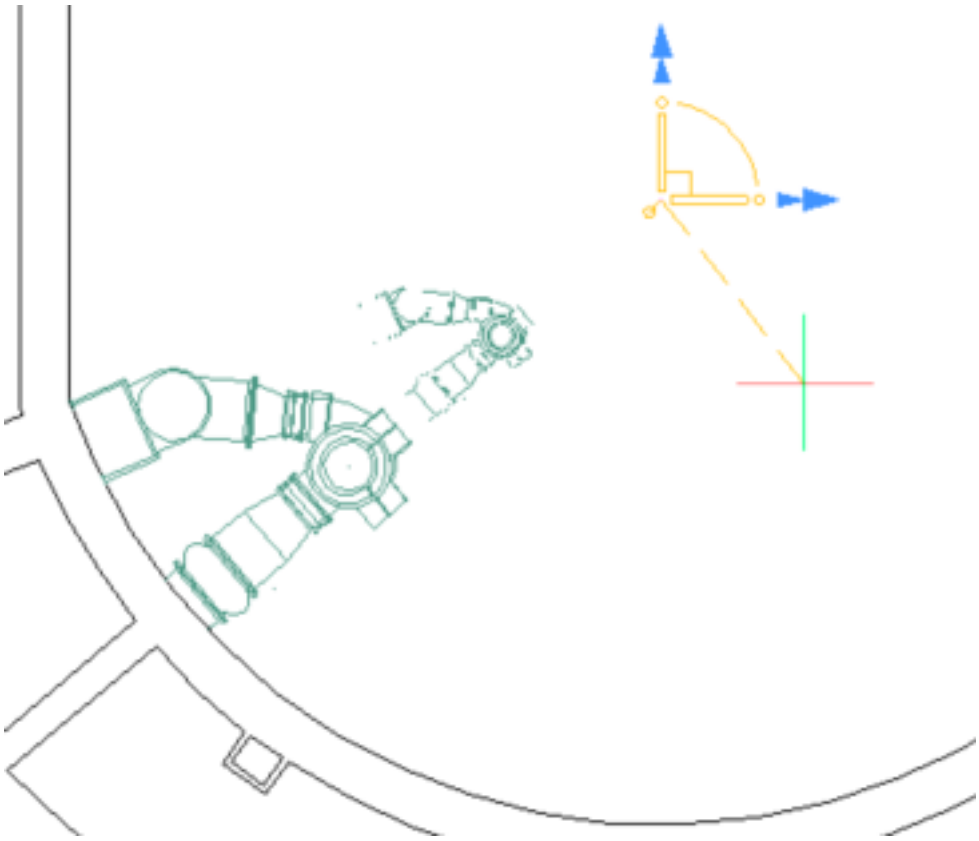
Mirror Select one of the arrowheads to mirror the entities along either axis. Mirror is the default behavior for the arrowheads. You can change their behavior to Scale using the right-click menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsy

Figure 1.1120



Scale With the arrowhead behavior set to Scale, you can select either arrow to scale the selected entities from the manipulator basepoint. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/ModifyingEntities/Enti

Figure 1.1121



Copy As you perform any of the four operations using the Manipulator (Move, Rotate, Mirror, Scale), you can create a copy of the selected entities. Simply press the Ctrl key as you select the operation.

1.24 Managing Images

1.24.1 Exploring Images

Commands: IMAGE (6.17.3), IMAGEADJUST (6.17.4), IMAGEATTACH (6.17.5), IMAGECLIP (6.17.7), IMAGEFRAME (6.17.8), IMAGEQUALITY (6.17.9) The **Image** command Inserts images in drawings through the Images Explorer. The **Imageadjust** command adjusts the properties of images through the Properties bar (1.12.4). The **Imageattach** command attaches raster images to the drawing. The **Imageclip** command clips images. The **Imageframe** command toggles the display of the frame around images. The **Imagequality** command toggles the display quality of images between draft and high. The **IMAGENOTIFY** system variable controls whether warnings for missing or modified image files are issued. The **XREFNOTIFYTIME** system variable defines the frequency to check for modified externally referenced resources: xrefs, images and pdf attachments if **XREFNOTIFY, IMAGENOTIFY** and/or **PDFNOTIFY** is ON.

Open the Images Explorer

To open the **Images Explorer** do one of the following:

- Choose **Drawing Explorer > Images** in the **Tools** menu.
- Click the **Image Management** tool button ( ) in the **Images** toolbar.
- Type **image** in the command bar, then press Enter.
- Select **Images** in the **Open Drawings** sub-window of the Drawing Explorer (1.14.2).

In the **Images Explorer** you can:

- See a preview of the attached images
- Place images ( ) or ( )
- Load / Unload inserted images
- Detach images ( )

Images Explorer display options

You can choose between **Detail View** (list), **Icon View** and **Tree View** to see the images in the current drawing.

Open the Detail View of the Images Explorer Click the **Detail View** button ( ) in the **Drawing Explorer** toolbar. The **Detail View** button is now pressed, indicating the image details are displayed. The selected image displays in the **Preview** sub-window of the **Drawing Explorer** .

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/13Images/ART/05ImagesDetailView.png}) Detail View of the images in the current drawing

Open the Icon View of the Images Explorer

Click the **Icon View** button ( ) in the **Drawing Explorer** toolbar. The **Icon View** button is now pressed, indicating the image icons are displayed. The selected image displays in the **Preview** sub-window of the **Drawing Explorer** .

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/13Images/ART/05ImagesIconView.png}) Icon View of the images in the current drawing

Open the Tree View of the Images Explorer

Click the **Tree View** button ( ) in the **Drawing Explorer** toolbar. The **Tree View** button is now pressed, indicating the image tree is displayed. The selected image displays in the **Preview** sub-window of the **Drawing Explorer** .

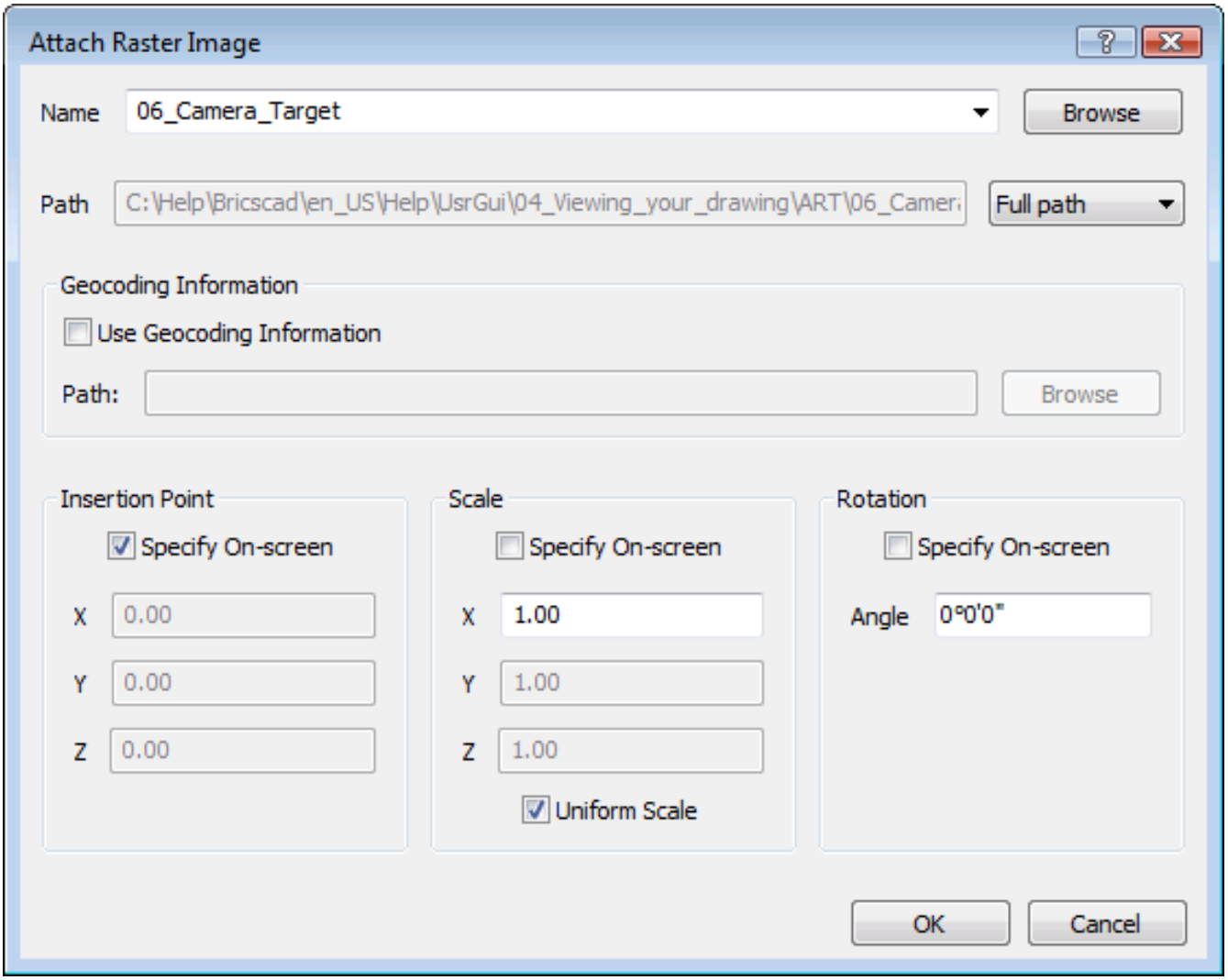
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/13Images/ART/05ImagesTreeView.png})
Tree View of the images in the current drawing

Placing images in a drawing

1. Do one of the following:
   - Launch the **Image Explorer** , then click the **New** tool button ( ) in the **Drawing Explorer - Images** toolbar.
   - Launch the **Image Explorer** , select a raster image file in the list, then click the **Insert** tool button ( ) in the **Drawing Explorer - Images** toolbar.
   - Click the **Attach Raster Image** tool button ( ) in the **Insert** toolbar.

   The **Attach Raster Image** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/
2. Do one of the following:
   - Accept the currently selected image file.
   - Click the down arrow button in the **Name** field to select a different raster image file in the list. Raster images which are already attached are listed.
   - Click the **Browse** button to select a new raster image file in the **Select Raster Image** dialog.
3. (option) Check the **Use Geocoding Information** option, then click the **Browse** button to open an appropriate world file to be used with the selected raster image. If the **Use Geocoding Information** option is checked, the **Insertion Point** , **Scale** and **Rotation** settings fields are not available.
4. (option) Do the following:
   - Specify an **insertion point** or select the **Specify on-screen** check box.
   - Specify **scale** or select the **Specify on-screen** check box.
   - Specify **rotation** angle or select the **Specify on-screen** check box.
5. Click the **OK** button.
6. If none of the **Specify on-screen** options is selected, the image is inserted at the desired insertion point, scale and rotation angle; else **You are prompted** to specify the insertion point, scale and/or rotation angle on-screen.

Figure 1.1122



Load / Unload images

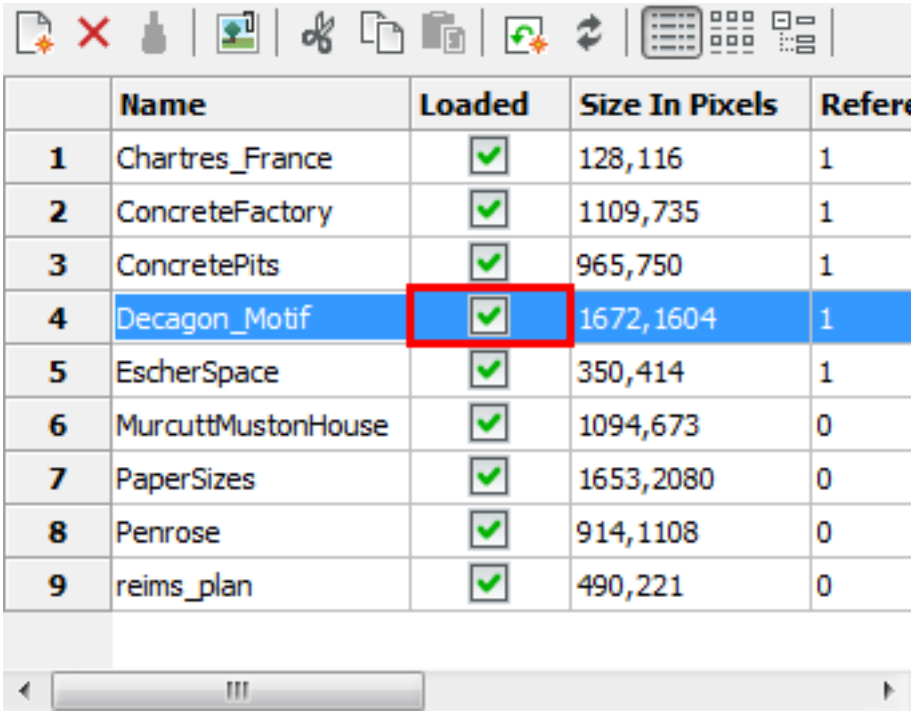
When an image is attached and then inserted in the drawing, you can unload the image to temporarily remove it. Unloaded images are still inserted in the drawing, but they no longer display. If the **Imageframe** setting is ON, the frame around the image still displays though.

Load / Unload a single image

1. Launch the **Image Explorer**.
2. For the image you want to load or unload, click the check box in the **Loaded** column. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/13Images/ART/05ImagesLoaded.png})
3. If the **Regen On/Off** button ( ) is not pressed, click the **Regen** tool ( ) in the **Images Explorer** toolbar.

NOTE	It is not necessary to select the image first in step 2.

Figure 1.1123



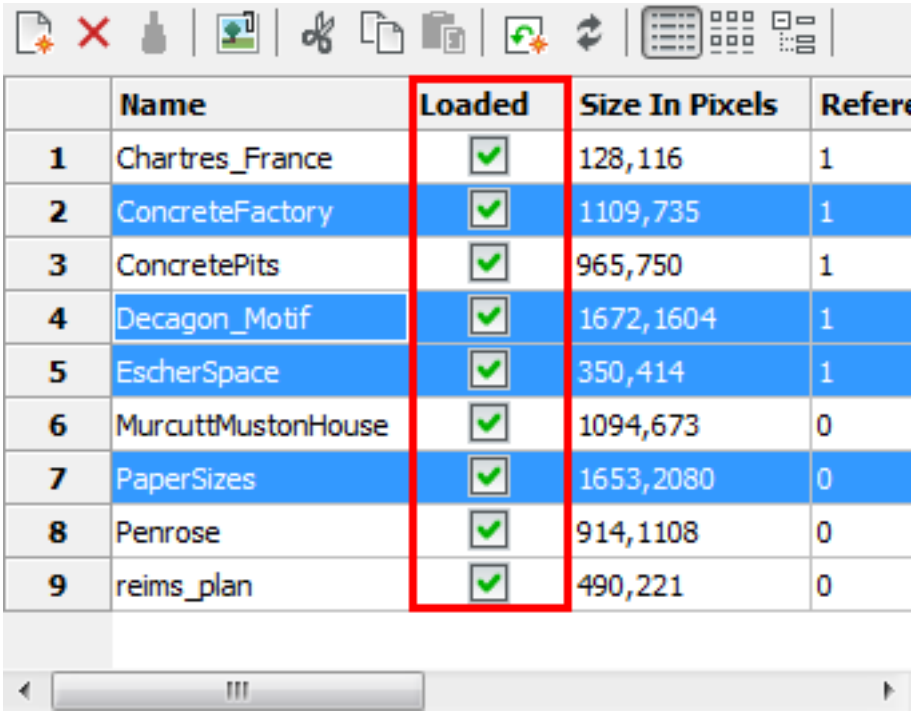
	Name	Loaded	Size In Pixels	Refere
1	Chartres_France	☑	128,116	1
2	ConcreteFactory	☑	1109,735	1
3	ConcretePits	☑	965,750	1
4	Decagon_Motif	☑	1672,1604	1
5	EscherSpace	☑	350,414	1
6	MurcuttMustonHouse	☑	1094,673	0
7	PaperSizes	☑	1653,2080	0
8	Penrose	☑	914,1108	0
9	reims_plan	☑	490,221	0

Load / Unload multiple images

1. Launch the **Image Explorer**.
2. Select the images you want to load or unload.
3. For one of the selected images, click the check box in the **Loaded** column. (\vref {fig:C:/Dev/Task-2448/images/staticbric
   All selected images will be loaded or unloaded simultaneously.
4. If the **Regen On/Off** button ( ) is not pressed, click the **Regen** tool ( ) in the **Images Explorer** toolbar.

NOTE Press and hold the Ctrl key to select multiple images.

Figure 1.1124



	Name	Loaded	Size In Pixels	Refere
1	Chartres_France	☑	128,116	1
2	ConcreteFactory	☑	1109,735	1
3	ConcretePits	☑	965,750	1
4	Decagon_Motif	☑	1672,1604	1
5	EscherSpace	☑	350,414	1
6	MurcuttMustonHouse	☑	1094,673	0
7	PaperSizes	☑	1653,2080	0
8	Penrose	☑	914,1108	0
9	reims_plan	☑	490,221	0

Removing images

Since an image is first attached, then inserted in the drawing, you can delete an image, without detaching it.

Detach images

1. Launch the **Image Explorer**.
2. Select the image(s) that you want to detach.
3. Do one of the following
4. Click the **Delete** tool button (✕) in the **Images Explorer** toolbar.
5. Right click and select **Delete** in the context menu.

NOTE Press and hold the Ctrl key to select multiple images.

Figure 1.1125

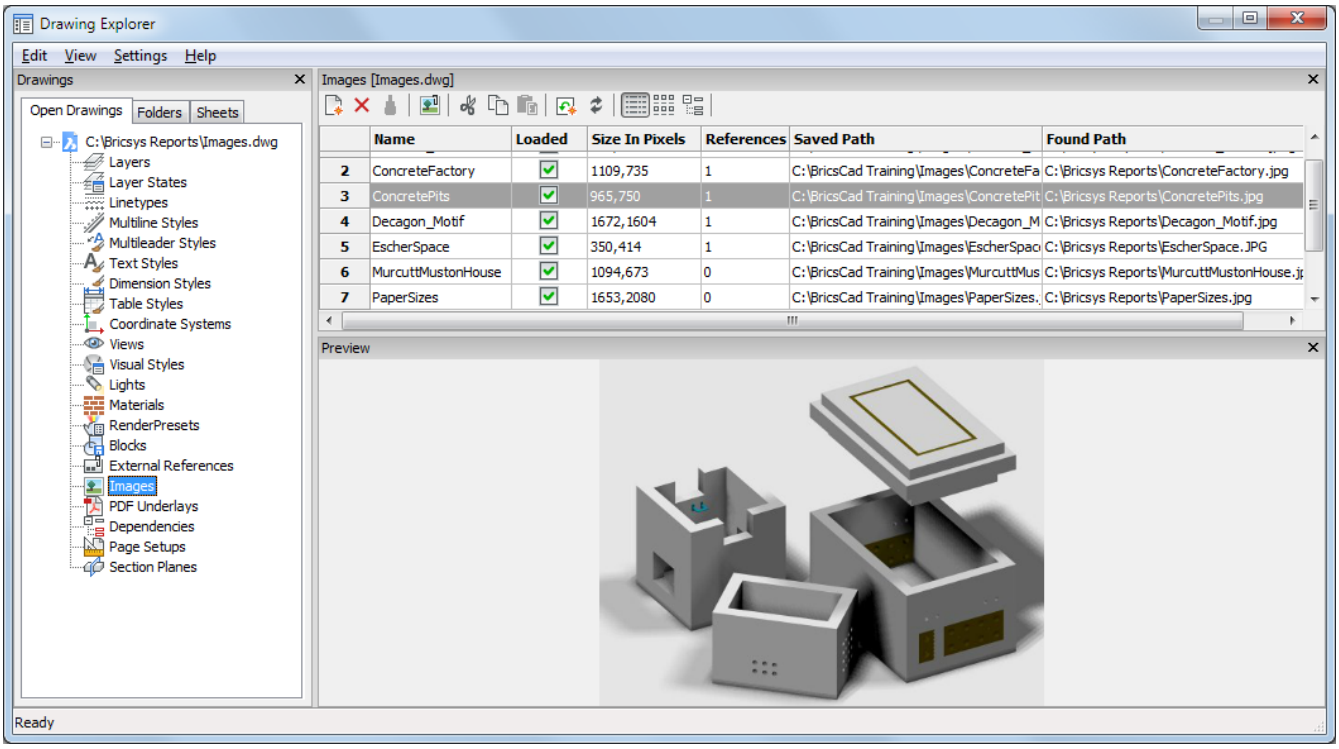


Figure 1.1126

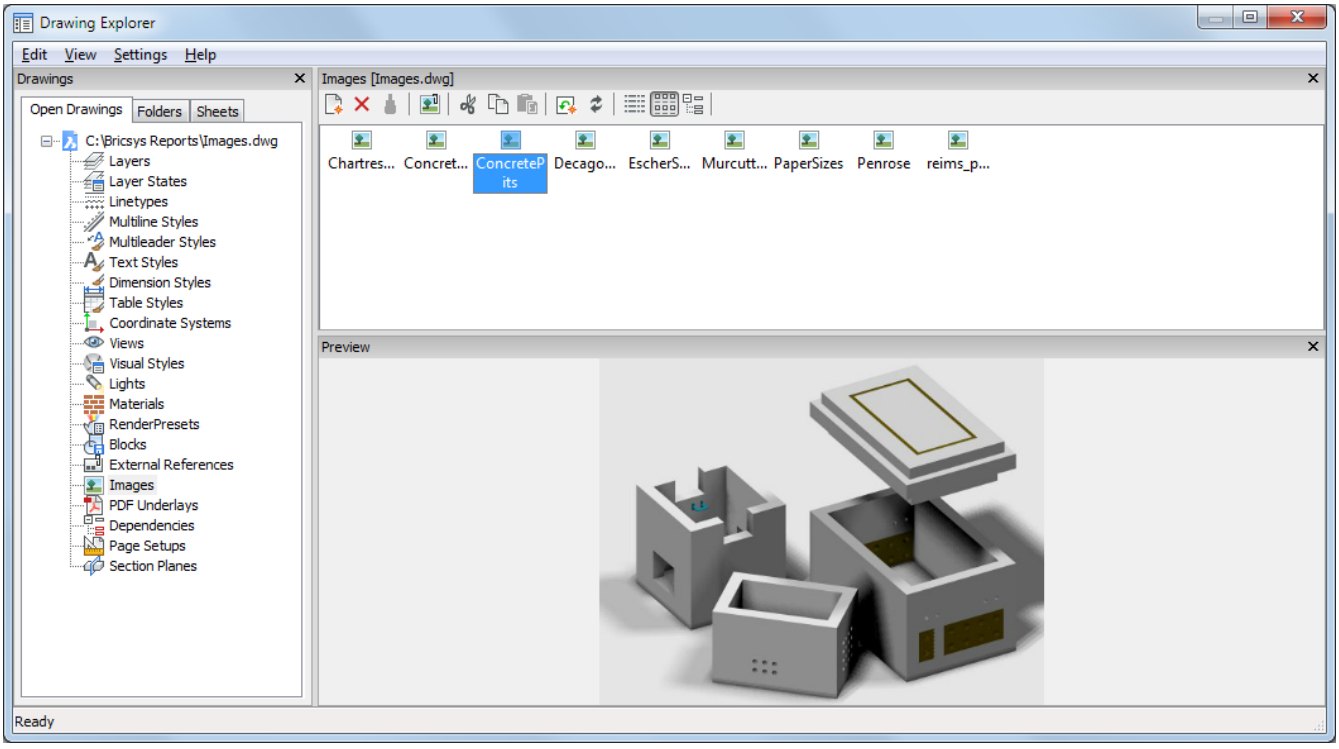
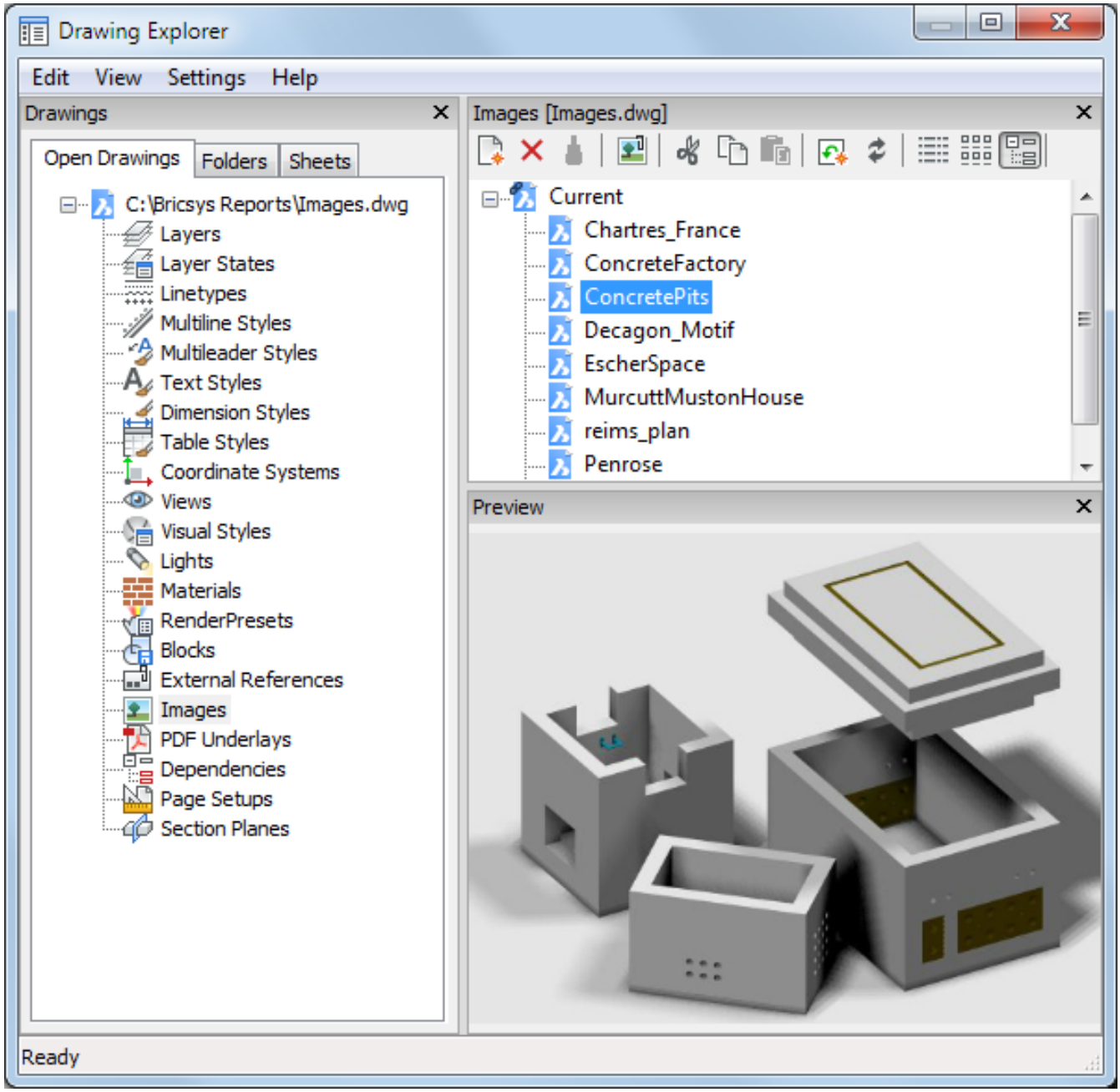


Figure 1.1127



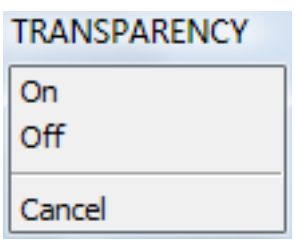
1.24.2 Image Utilities

To set the image transparency

1. Do one of the following:
   - Click the **Image Transparency** tool button ( ) on the **Images** toolbar.
   - Choose **Image Transparency** in the **Images** menu.
   - Type **transparency** in the command bar, then press Enter. **You are prompted:** Select images.
2. Select (1.23.6) the images you want to set the **Transparency** property of. **You are prompted:** Enter transparency mode [ON/OFF]<current mode>: A prompt menu displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscor
3. Do one of the following:
   - Press Enter to accept the current mode.
   - Choose **On** in the prompt menu or type **ON** in the command bar, then press Enter to switch the **Transparency** property of the selected images on.
   - Choose **Off** in the prompt menu or type **OFF** in the command bar, then press Enter to switch the **Transparency** property of the selected images off.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/13Images/ART/05Transparency.png}) Transparency OFF (left) and Transparency ON (right)

Figure 1.1128



Modify the Image Frame setting

Using the Image Frame tool:

1. Click the **Image Frame** tool () in the **Images** toolbar. **You are prompted:** Enter image frame setting <1>:
2. Do one of the following:
   - Type 0 + Enter to hide image frames.
   - Type 1 + Enter to display and plot image frames.
   - Type 2 + Enter to display but not print image frames.

Editing the Image Frame system variable:

1. Open the **Settings** dialog.
2. Go to **Drawing / Underlays**.
3. Set the **Frame** system variable = **3**: Use individual settings.
4. Select the **Image Frame** system variable. The options are:
   - 0: Hide image frames.
   - 1: Display and plot image frames
   - 2: Display but do not plot image frames.

To clip an image

1. Make sure image frames display.
2. Do one of the following:
   - Click the **Clip Image** tool button () on the **Image** toolbar.
   - Type **imageclip** in the command bar, then press Enter. **You are prompted:** Select image:
3. Click the image frame. **You are prompted:** Enter image clipping option [ON/OFF/Delete/New] <New>:
4. Press Enter to accept the **New Boundary** default option. **You are prompted:** Enter image clipping type [Polygonal/Rectangular] <Rectangular>:
5. Press Enter to accept the **Rectangular** default option. **You are prompted:** Specify first corner.
6. Click a point to define the first corner of the clipping rectangle. The clipping rectangle displays dynamically. **You are prompted:** Specify opposite corner:
7. Click a point to define the clipping rectangle. The image is clipped.

To toggle the clipping boundary of an image

1. Make sure image frames display.
2. Click the image frame. The properties of the image display in the Properties Bar (1.12.4).
3. Under **Misc / Show clipped** choose **Yes** or **No**.

Figure 1.1129



1.25 Printing and Plotting

1.25.1 General procedure for printing

Commands:PLOT (6.24.26), PRINT (6.24.46), QPRINT (6.25.3), PAGESETUP (6.24.1), PUBLISH (6.24.55)

- The **Plot** command plots drawings to printers and to files, through prompts at the command bar; this command is meant for scripts and routines.
- The **Print** command prints drawings to plotters and files through a dialog box.
- The **Qprint** command prints the drawing with the default plot configuration, without displaying the Print dialog box (short for "quick print").
- The **PageSetup** command creates and edits page setups for plotting drawings in the Drawing Explorer.

Youtube Youtube A standard print procedure consists of the following steps:

- Choose either a Layout (1.16.15) tab or the Model (1.16.13) tab.
- Open the Print dialog. (1.25.1)
- Select a Page Setup name (6.24.1). (1)

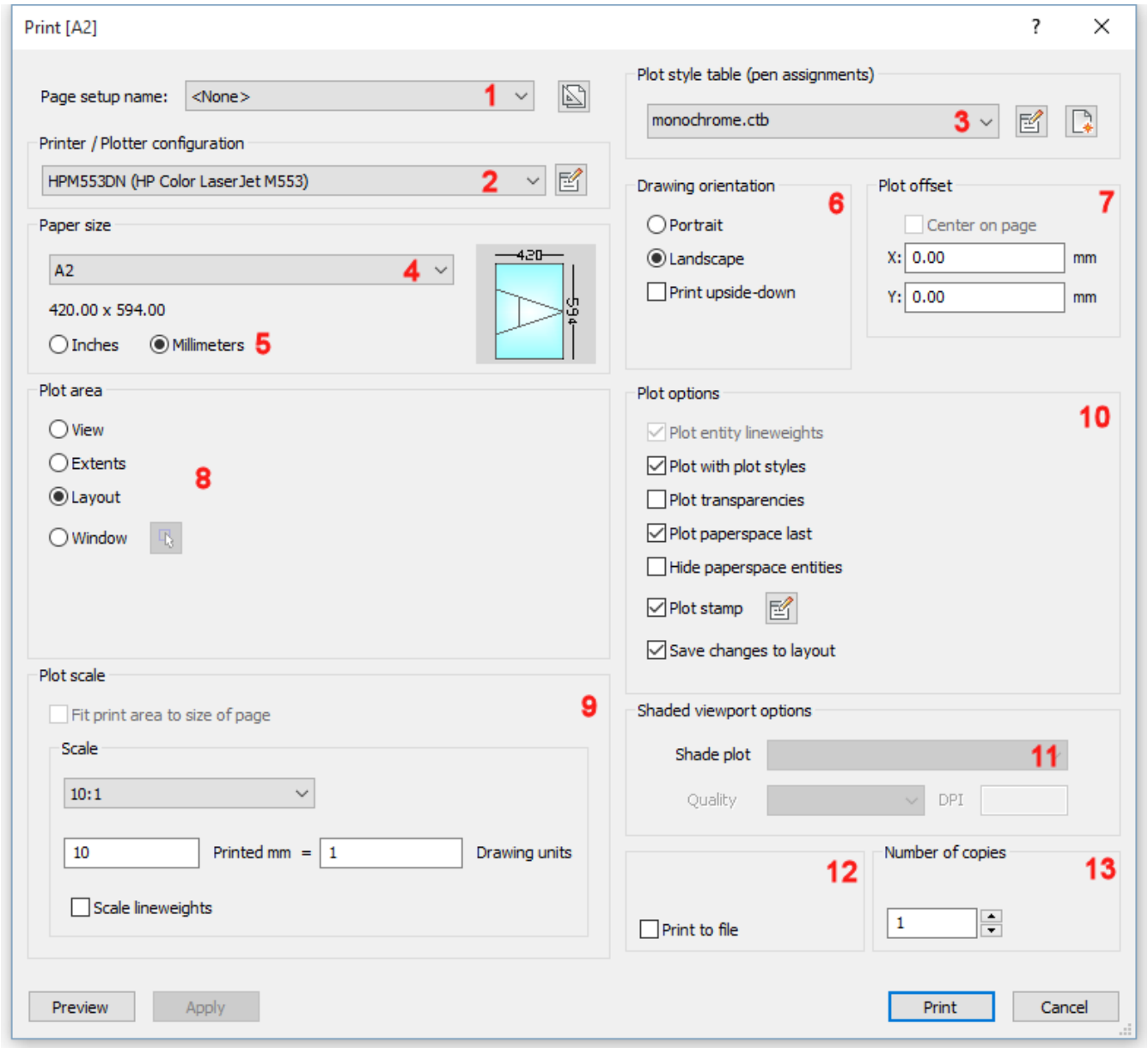
The following steps are optional and only necessary if you want to use overrides on the selected page setup definition.

- Choose a Printer or Plotter Configuration (1.25.2). (2)
- Choose a Plot style table (1.25.4). (3)
- Choose a Paper Size. (4) The image reflects the position and orientation of the drawing on the selected paper size.
- Choose the Paper Units. (5)
- Choose the Drawing Orientation.(6)
- Define the Plot offset. (7)
- Specify the Area to plot. (8)
- Define the Plot Scale. (9)
- Check the Plot Options. (10)
- When printing model space, choose a Shade Plot mode. (11) This option is disabled when printing a paper space layout.
- Check/uncheck the **Print to file** option (12)
- Specify the Number of copies. (13)
- Click the **Preview** button to see a preview of the plot output.
- When in a layout, click the **Apply** button to update the layout.
- Click the **Print** button to start printing.

NOTE	All print settings and options are saved in the **Model** and each of the **Layouts** .

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PrintingandPlotting/GeneralProcedureForPrint

Figure 1.1130



To open the Print dialog

Do one of the following:

- Choose **Print...** in the File menu.
- Type **print** in the command bar, then press Enter.
- Press **Ctrl+P**
- Select the **Model** tab or a **Layout** tab, then right click and choose **Print** in the context menu.

NOTE	If you click the **Print** ( ) button in the **Standard** toolbar, the **Qprint** command is launched, which will start printing the currently selected view tab using previously saved print settings.

To choose the paper size

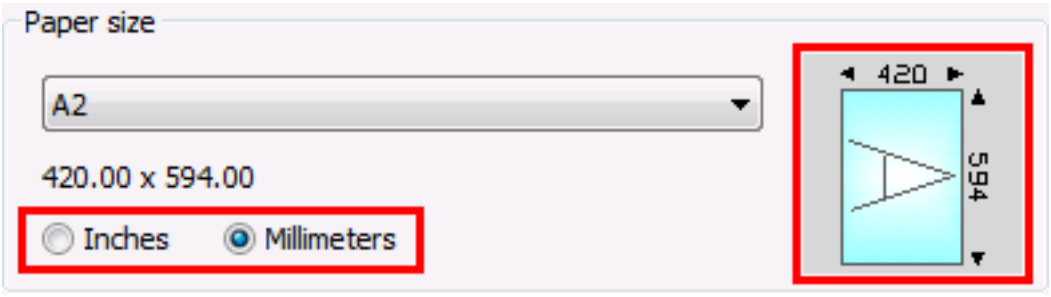
Press the down arrow, then select the paper size in the **Paper Size** list. The available sizes depend on the selected printer.

To choose the paper units

Select either **Inches** or **Millimeters** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Printingan
The paper size image indicates the size, position and orientation of the plot area on the selected paper size.

NOTE	The **Scale** and **Plot offset** setting units are changed according to the chosen **Paper units.**

Figure 1.1131



To choose the paper orientation

Select either **Portrait** or **Landscape** .

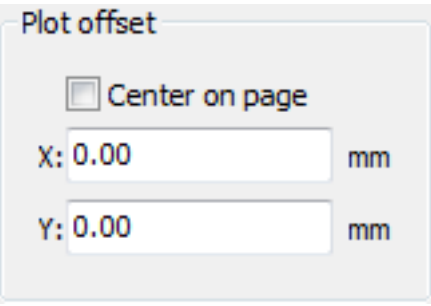
- **Portrait** : The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.
- **Landscape** : The drawing or layout x-axis is aligned with the longest edge of the selected paper size.

The paper size image changes accordingly. When in a layout, click the **Apply** button to update the display.

To define the plot offset

Select either **Center on page Or** type the plot offset in the **X:** and **Y:** fields. The offset is calculated from the bottom left corner of the paper. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PrintingandPlotting/Ge
When in a layout, click the **Apply** button to update the display.

Figure 1.1132



To specify the area to plot

Option	Result
View	Prints the portion of the drawing that displays in the current viewport or a saved view.
Extents	Prints all entities in the drawing
Limits	Prints to the limits defined in the drawing This option is only available when printing in Model space.
Layout	Prints the current layout. This option is only available when printing in Paper space.
Window	Prints the portion of the drawing contained in a user defined window, maintaining the aspect ratio of the windowed area to the drawing. Click the **Select print area** button (), then specify the two corners of the print area in the drawing. or Type the the X- and Y-coordinates of the corners of the print area in the **X:** and **Y:** fields. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Printingand

To define the plot scale

NOTE	The SCALELISTEDIT (6.27.7) command allows to edit the list of available scales in the **Print** dialog and the **Standard Scale** property of a paper space viewport.

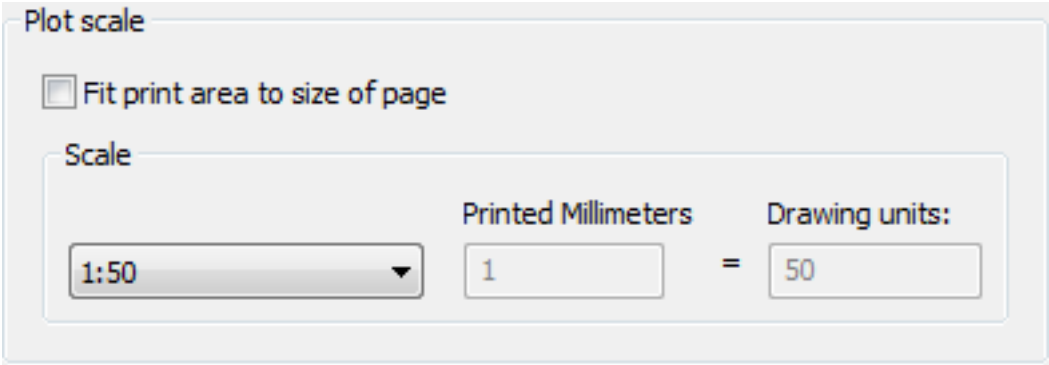
... when printing in paper space: If you print in paper space, the final scale of the plot output is the product of the viewport scale (1.16.14) and the plot scale. In order to keep control over the scale of the plot output:

- Specify the width and height of the viewports in drawing units.
- Set the scale of each viewport to the scale you want in the plot output.
- The **plot scale** reflects the relationship between paper units and drawing units. (see table below)

Paper units	Drawing units	Plot scale	Printed Millimeters or Inches	Drawing Units
mm	mm	1:1	1	1
mm	cm	10:1	10	1
mm	m	1000:1	1000	1
inches	inches	1:1	1	1
inches	feet	12:1	12	1

... when printing in model space: If you print in model space, the scale of the plot output equals the plot scale, on condition your drawing units equal the paper units. Otherwise you need to multiply the plot scale with respect to the relationship between paper units and drawing units. E.g. if you want to plot at a 1/50 scale in a drawing of which 1 DU = 1 cm, the **plot scale** = 1/5, if 1 DU = 1 m the **plot scale** = 20/1 or 1/0.05 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PrintingandPlotting/GeneralProcedureForPrintingPlotScale.png})

Figure 1.1133

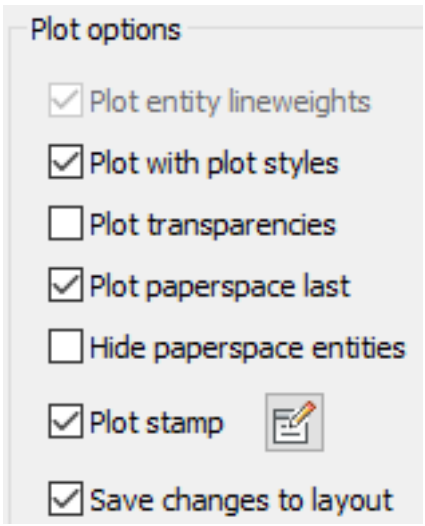


To set the plot options

Click to toggle the **plot options** On / Off. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Printing

Option	Result
Plot object lineweights	If **Plot with plot styles** is off, object lineweights are plotted.
Plot with plot styles	Settings of the assigned STB or CTB file are used.
Plot transparencies	Plots entities using the value of the Transparency (1.17.8) property.
Plot paperspace last	Entities in Model space are plotted first, paperspace entities last.
Hide paperspace entities	Removes hidden lines from 3D entities in paper space. This option is disabled when printing model space.
Plot stamp	Adds a header and / or footer. Click the **Edit plot stamp** button ( ) to define the plot stamp.
Save changes to layout	All changes made in the **Print** dialog are saved in the layout. This option toggles the **Save changes to layout** variable

Figure 1.1134

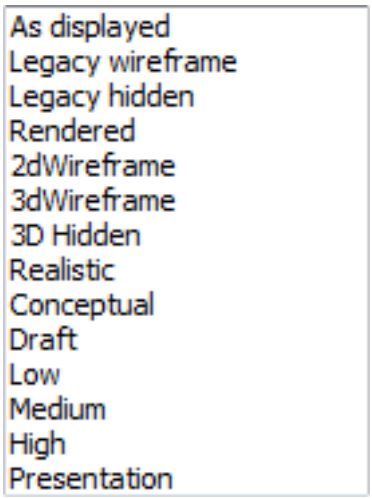


To set the shade plot mode

Under **Shade viewport options** , click the list button, then select an option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

NOTE	This option applies to model space printing only. The **Shade Plot** mode of a paper space viewport is defined in the Shade Plot (1.16.14) property of the viewport.

Figure 1.1135



To define the plot stamp

1. Click the **Edit plot stamp** button (). The **Plot stamp** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

2. In the **Header** field, do one of the following:

   - Type the header text.
   - Click the down arrow button, then choose a variable in the drop down list.
   - Type one or more variable names.
   - Leave the field empty.

3. In the **Footer** field, do one of the following:

   - Type the footer text.
   - Click the down arrow button, then choose a variable in the drop down list.
   - Type one or more variable names.
   - Leave the field empty.

4. Select a font in the **Font** list button.

5. Enter a value in the **Font Size** field

6. Set the unit type to **Millimeters** or **Inches** .

7. Click the **OK** button to confirm.

The plot stamp settings are saved to a series of user preferences in the current user profile (1.13.9), which can be edited in the Settings (1.13.11) dialog under **Program Options / Plot and Publish / Plot Stamp** : (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PrintingandPlotting/GeneralProcedureForPrintingPlotStampSettings.

Figure 1.1136

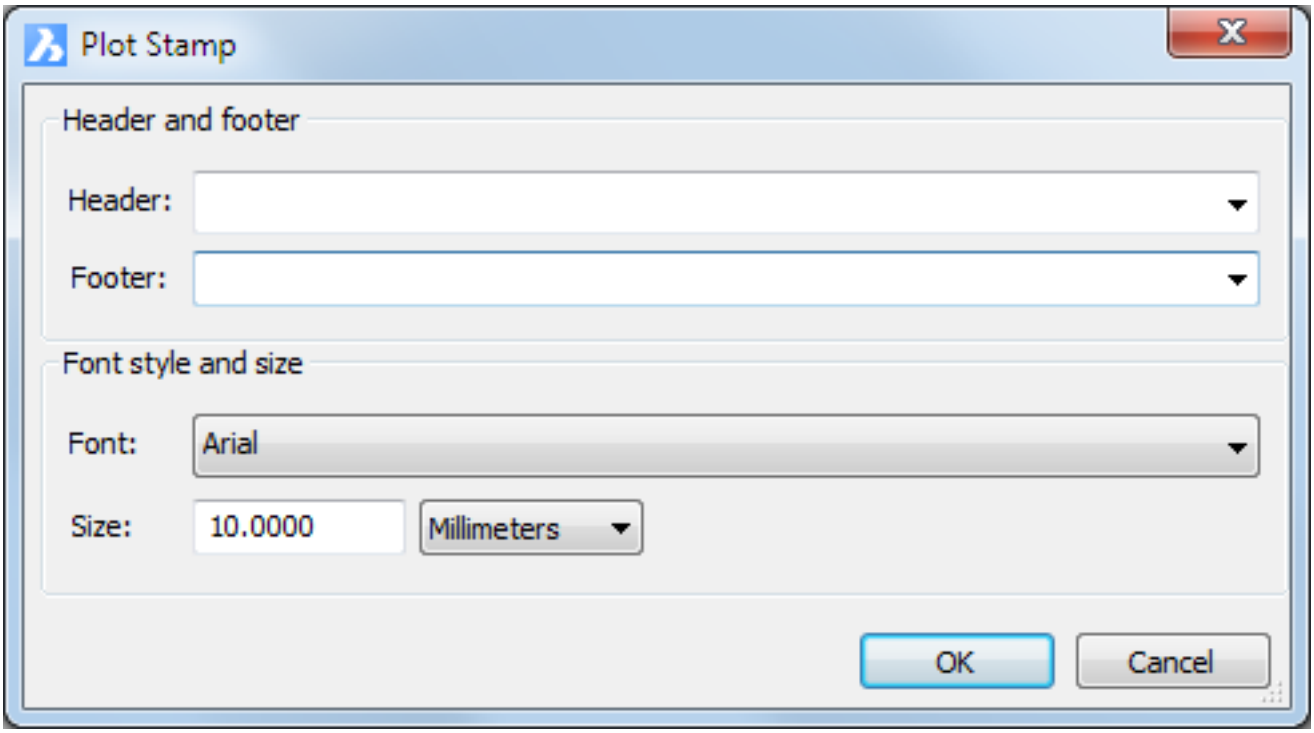


Figure 1.1137

Plot stamp	
Header	
Footer	
Font Style	Arial
Font Size	10
Units	[1] Milimeters

Name	Description
$Name	Adds the name of the current user, as definied in the **User Name** field of the License Information.
$Company	Adds company name of the current user, as definied in the **Company** field of the License Information.
$Date	Adds the current date.
$DateTime	Adds the current date and time.
$DwgName	Adds the file name and path of the drawing.

NOTE

The header and footer text consists of three parts (left, center and right), separated by commas. By default the header and footer text is centered. The header and footer text can be outlined left center and/or right by separating the text with commas. e.g. **$Name,$Company,$Date** places your name left, company name centered and the current date at the right. Type a space before one of the commas to leave one of the parts empty, thus outlining the header or footer left or right.

- **$Company, ,** ($Company,<space>,<space>) outlines the company name left.
- , ,$Company (<space>,<space>,$Company) outlines the comapny name right.

Variables

Tutorial: Creating Plot Stamps Click here (1.8.7) to watch

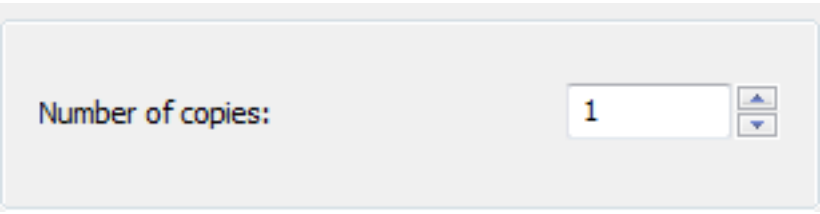
To specify the number of copies

Do one of the following:

- Type the desired number in the **Number of copies** field.
- Click the up-arrow and the down-arrow buttons to specify the number of copies.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PrintingandPlotting/GeneralProcedureForPrint

Figure 1.1138



To see a preview of the plot output

1. Do one of the following:
   - Click the **Preview** button on the **Print** or **Page Setup** dialogs.
   - Choose **Print Preview...** in the **File** menu.
   - Type **preview** in the command bar, then press Enter.
2. (option) Use the mouse scroll wheel to zoom. Double click the middle mouse button to zoom extents.
3. To close the **Print Preview** window, do one of the following:
   - Click the **Close** button to return to the drawing window.
   - Click the **Print Settings...** button to open the Print dialog.
   - Click the **Print** button to start printing.

Figure 1.1139

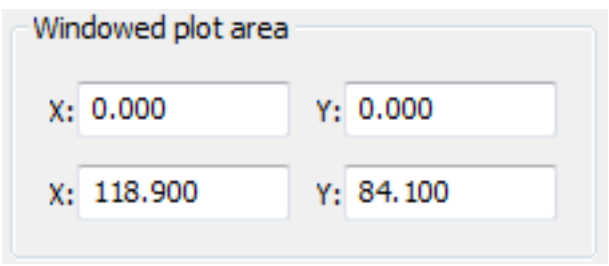
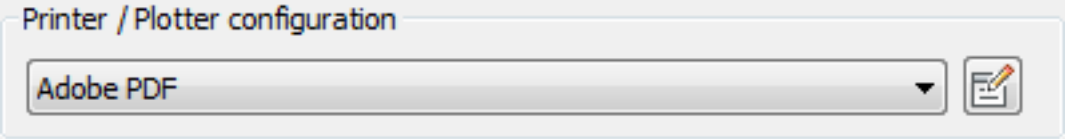


Figure 1.1140



1.25.2 Plotter Configuration

Command: PLOTTERMANAGER (6.24.30) Youtube Configurations for Windows system printers are stored in plotter configuration files (.PC3 files). Unlike in AutoCAD®, PC3 plotter configuration files in BricsCAD™ apply to Windows system printers only. In a plotter configuration file, you override one or more settings of the system printers installed on your computer. You can configure BricsCAD for many plotting/printing devices and keep multiple configurations for each single device. Each plotter configuration file contains information such as:

- The device driver and model
- The output port to which the device is connected
- Various device-specific settings (not supported on the Linux platform).

PC3 files are saved in the **Plotconfig** subfolder of your Roamable root (1.13.2) folder. After creating a PC3 file, it is available in the list of plotter configuration names in the **Printer / Plotter configuration** list on the **Print** or **Page Setup** dialog boxes. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART/0

To configure the built-in PDF printer

1. Open the Print dialog (1.25.1).
2. Click the **Printer/Plotter configuration** list button and choose **Print As PDF.pc3** in the list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PrintingandPlotting/PlotterConfigurationPrintAsPdf.png})
3. Click the **Edit Plotter Configuration** button ( ). The **Plotter Configuration Editor** dialog displays withe the **Print As PDF.pc3** configuration file loaded. The default path for plotter configuration files (*.pc3) is: **C:\Users\<User_Name>\AppData\Roaming\Bricsys\BricsCAD\Vxx\en_US\PlotConfig** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/PrintingandPlotting/PlotterConfigurationPrintAsPdf.png})
4. Click the **Settings** tab on the **Plotter Configuration Editor** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/
5. (Option) Edit the **Graphics** settings.
6. (Option) Execute the **Scale Calibration** .
7. Click the **Custom Properties** button. The **Print As PDF - Custom Properties** dialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART/PrintAsPdfCustomProperties.png})
8. (Option) Adjust the **Output quality** and **Layers suppor** t settings.
9. Click the **Manage Custom sizes ...** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG
10. To create a custom paper size do the following:

Figure 1.1141

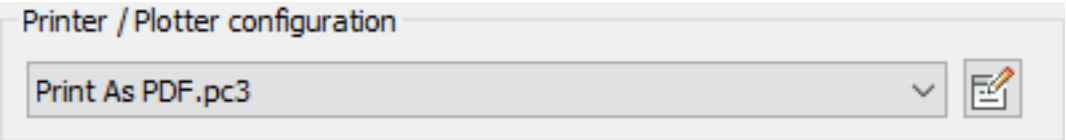


Figure 1.1142



Figure 1.1143

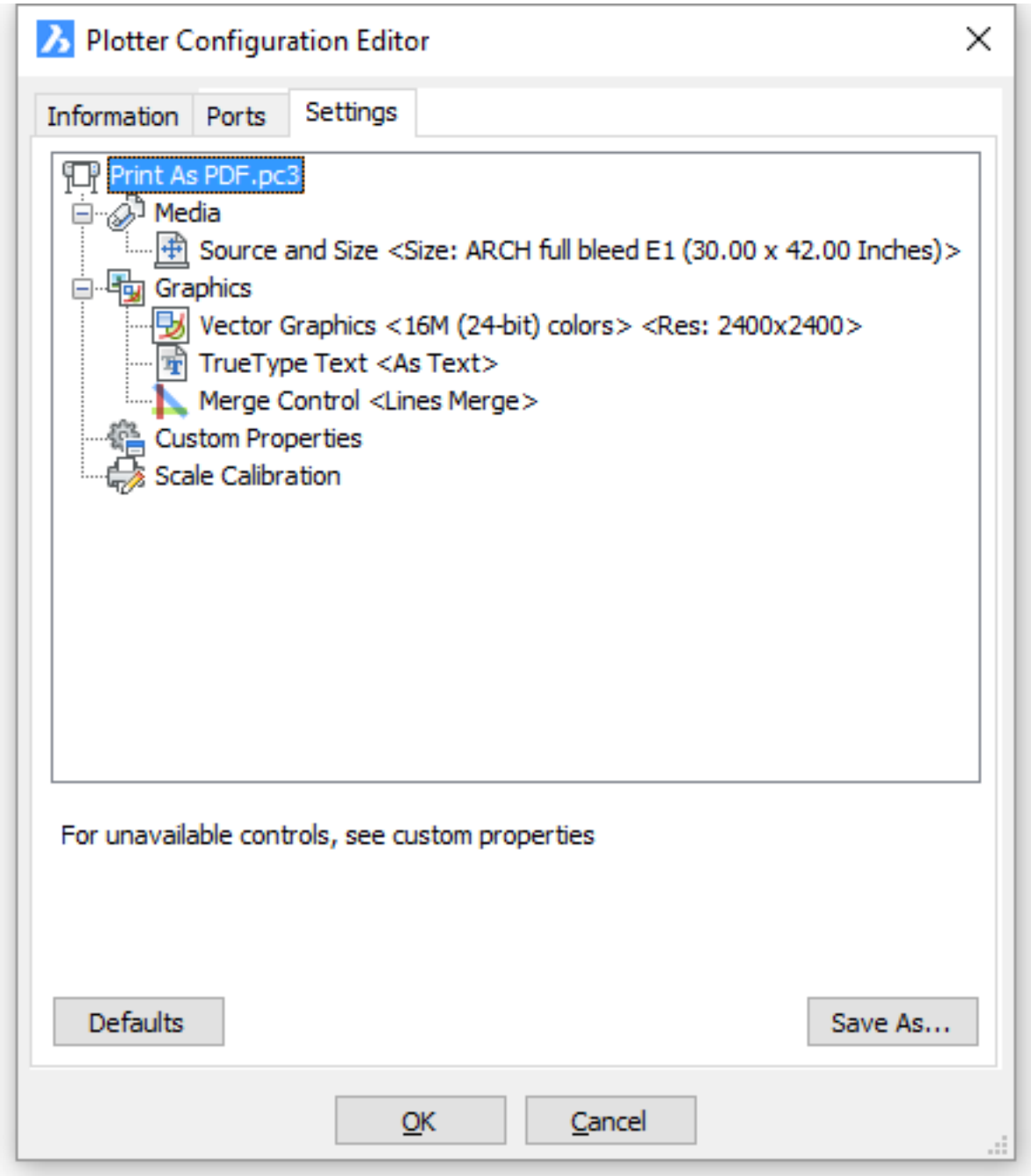


Figure 1.1144

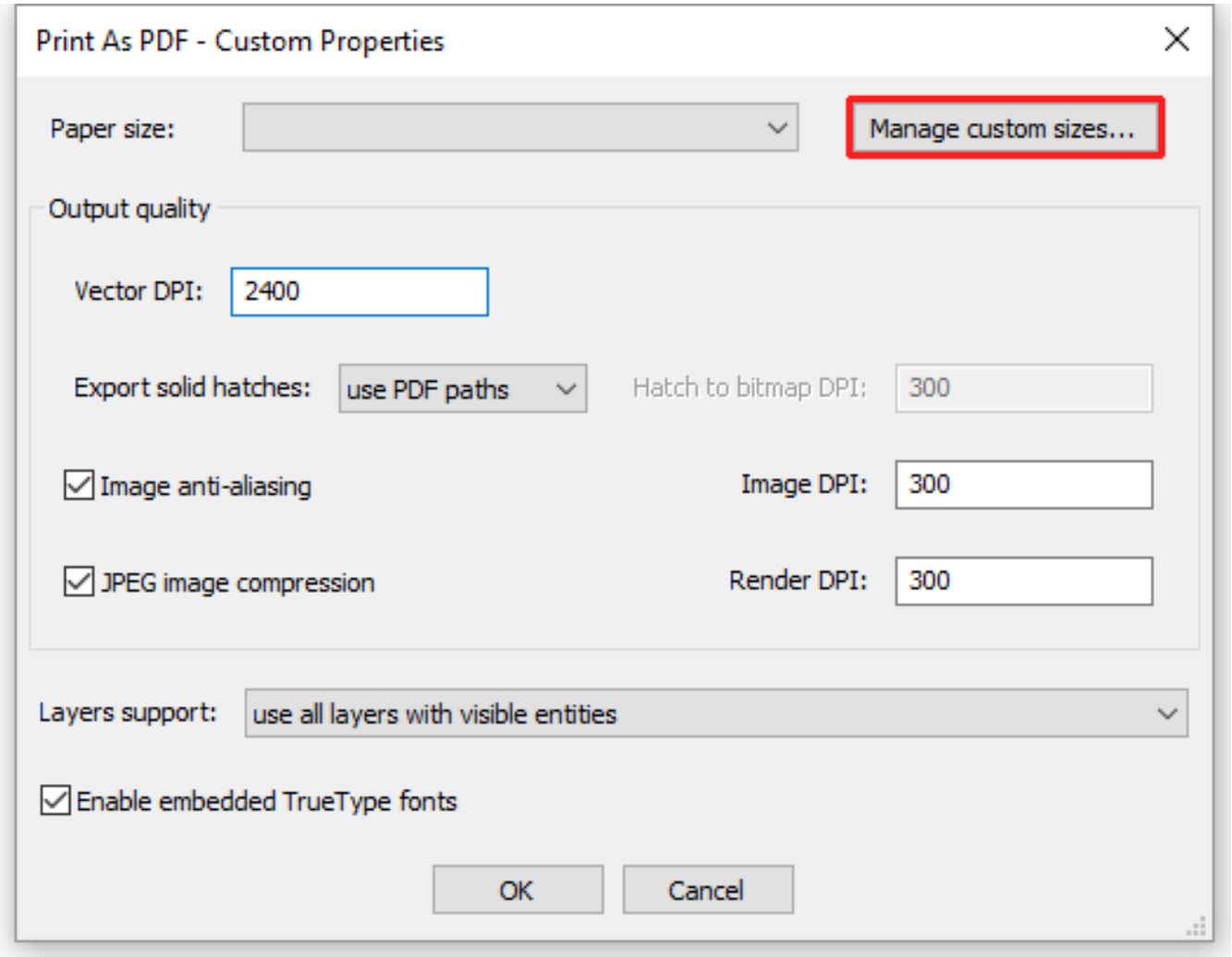


Figure 1.1145

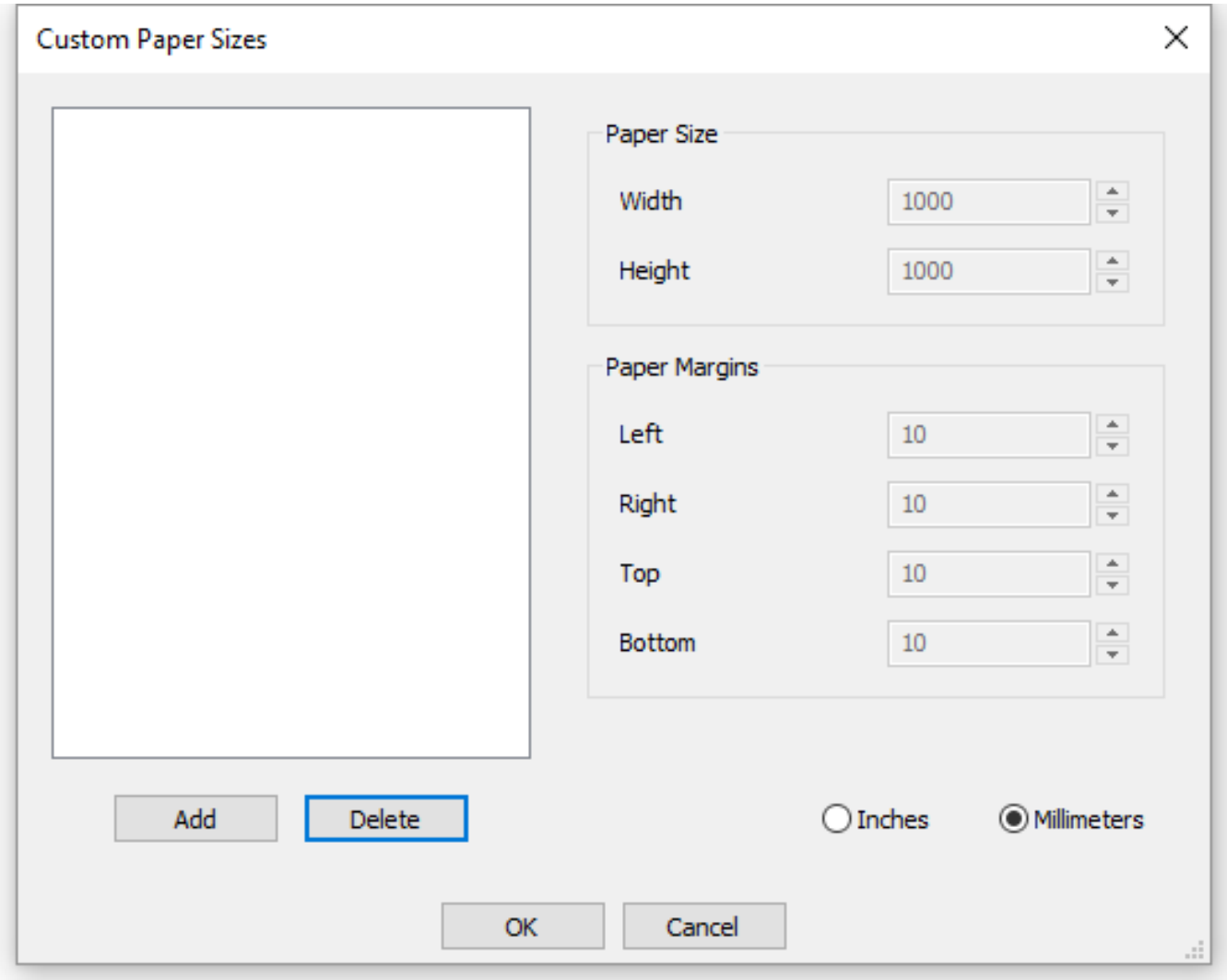


Figure 1.1146

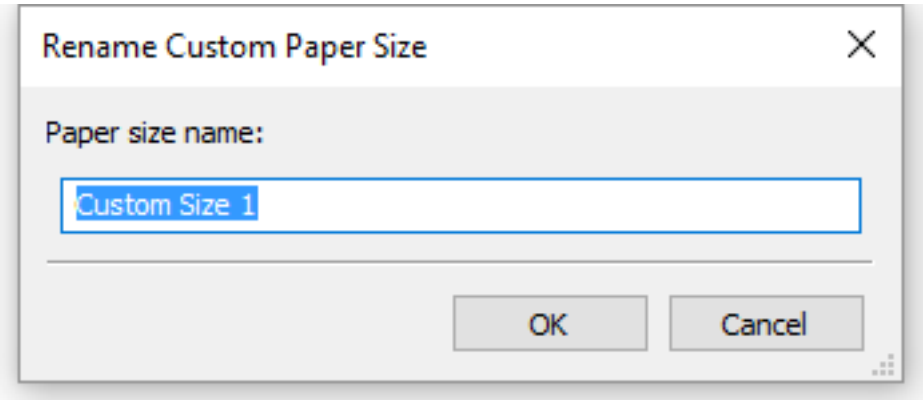
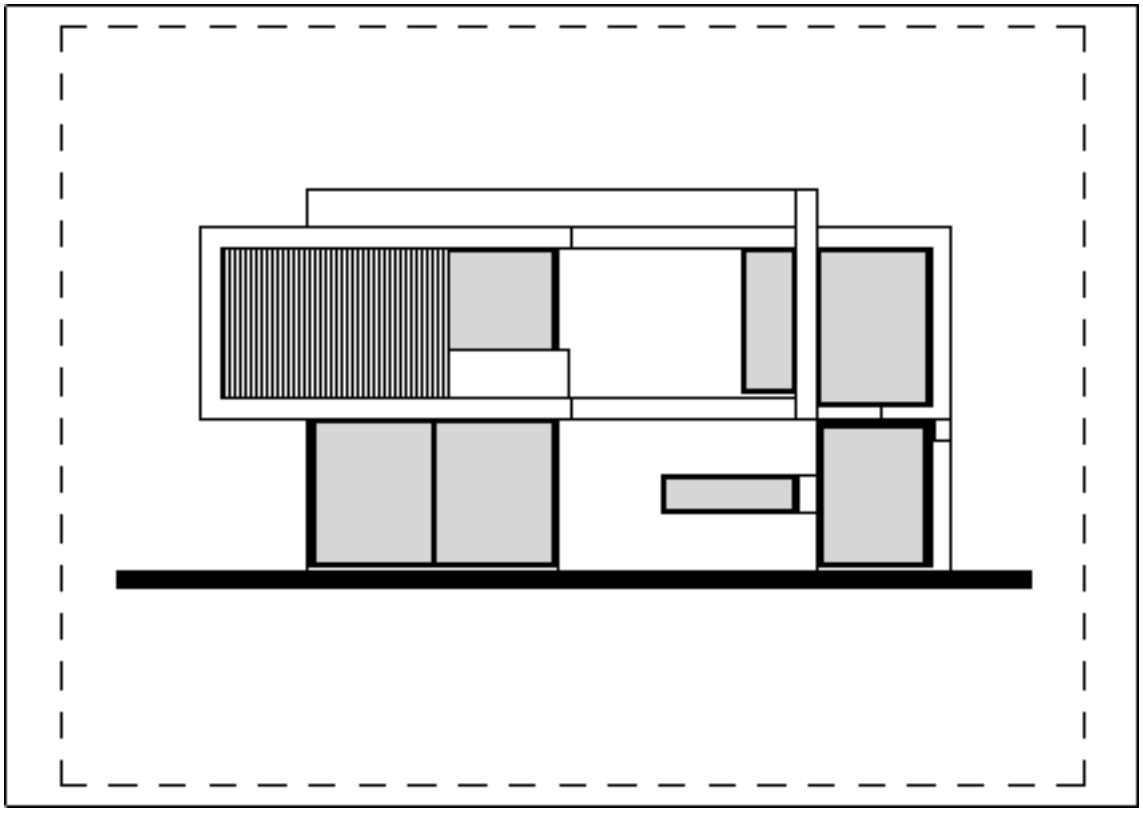


Figure 1.1147



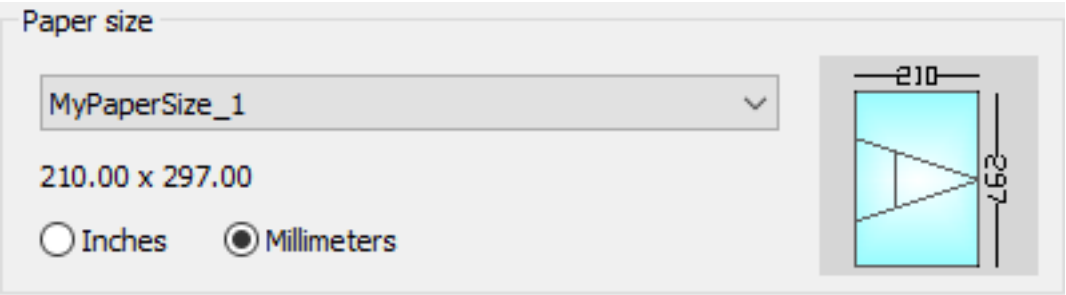
- Click the **Add** button. A new paper size is added.
- Double click the newly created papersize.
- Type a name for the new paper size, then click the **OK** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

- Define **Width** an **Height** for the new paper size.
- Define the **Paper Margins** . Paper margins define the printable area of paper sheet. If the DISPPAPER-MARGINS (1.13.11) system variable is **ON** , dashed lines indicate the paper margins on the paper sheet* in a layout. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART
  * The paper sheet background displays if the DISPPAPERBKG (1.13.11) system variable is **ON** .
- Click the **OK** button to create the custom paper size.

11. Click the **OK** button on each of the previously opened dialogs. The custom paper size(s) can now be selected in the **Paper Size** list button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15F

Tutorial: Output As Pdf Click here (1.8.1) to watch

Figure 1.1148



To create a plotter configuration file

1. Choose **Plotter Manager...** in the **File** menu, then double click **Create a Plotter Configuration** shortcut. The **Plotter Configuration Editor** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG
2. In the **Printer name** list, select the printer you want to create a plotter configuration (PC3) for.
3. Click the **Settings** tab on the **Plotter Configuration Editor** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/
4. Click the **Custom Properties...** button. The **<Selected Printer> Properties** dialog opens. The layout of this dialog is different for each printer. Custom Properties are not supported on the Linux platform.
5. On the **Properties** dialog of the selected printer, adjust the printer settings, then click the**OK** button to confirm. The **<Selected Printer> Properties** dialog closes.
6. Click the **OK** button on the **Plotter Configuration Editor** dialog. The **Save Plotter Configuration As** file dialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART/01Plo
7. (option) Adjust the PC3 file name. The default name is **<Selected Printer>.pc3** .
8. Click the **OK** button on the **Changes to a Printer Configuration File** dialog. The PC3 file is created.

NOTE	If you want to create multiple plotter configurations for the same printer, you must adjust the default name in step 7. If you accept the default name, the existing plotter configuration will be overwritten.

Figure 1.1149

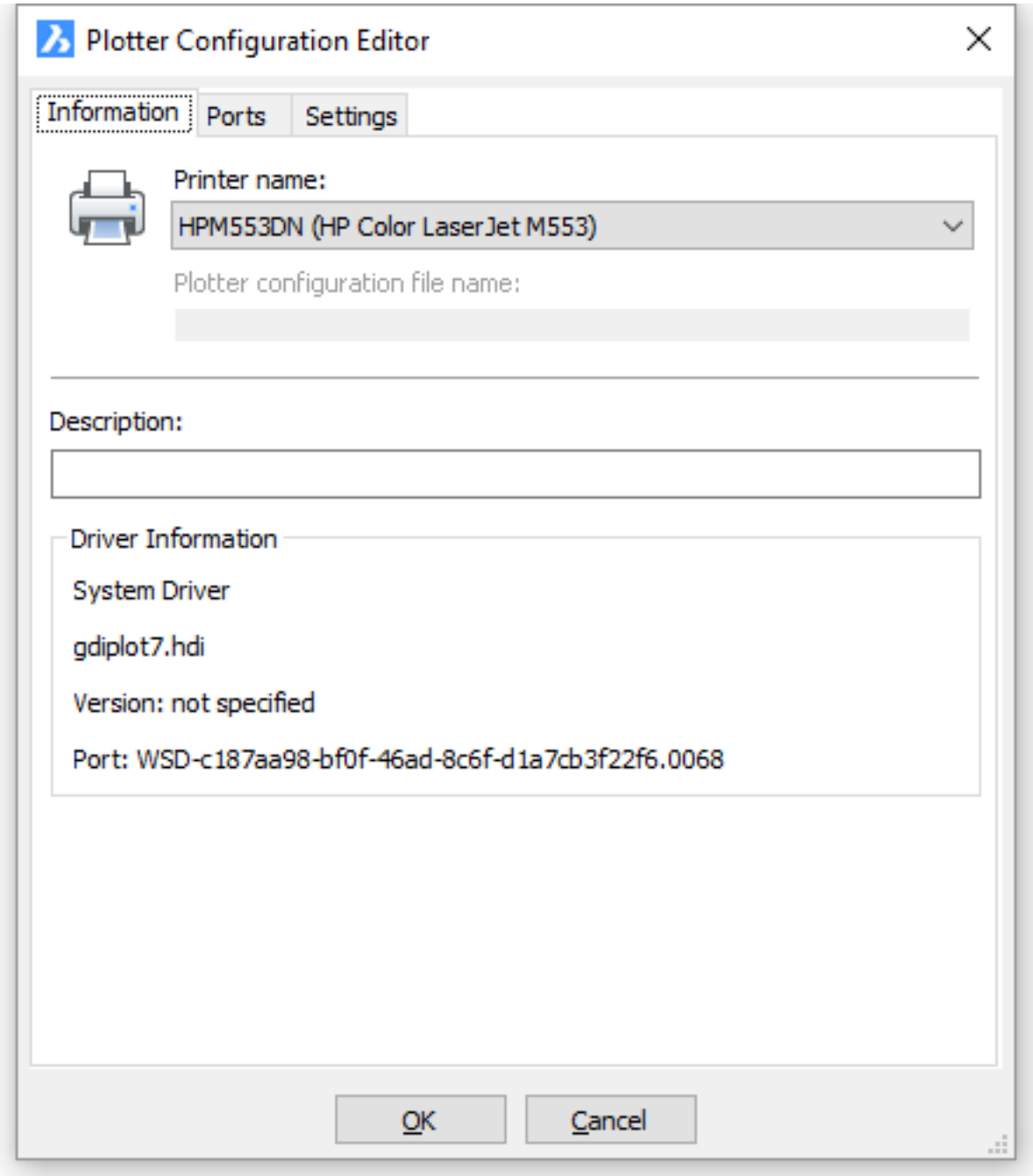


Figure 1.1150

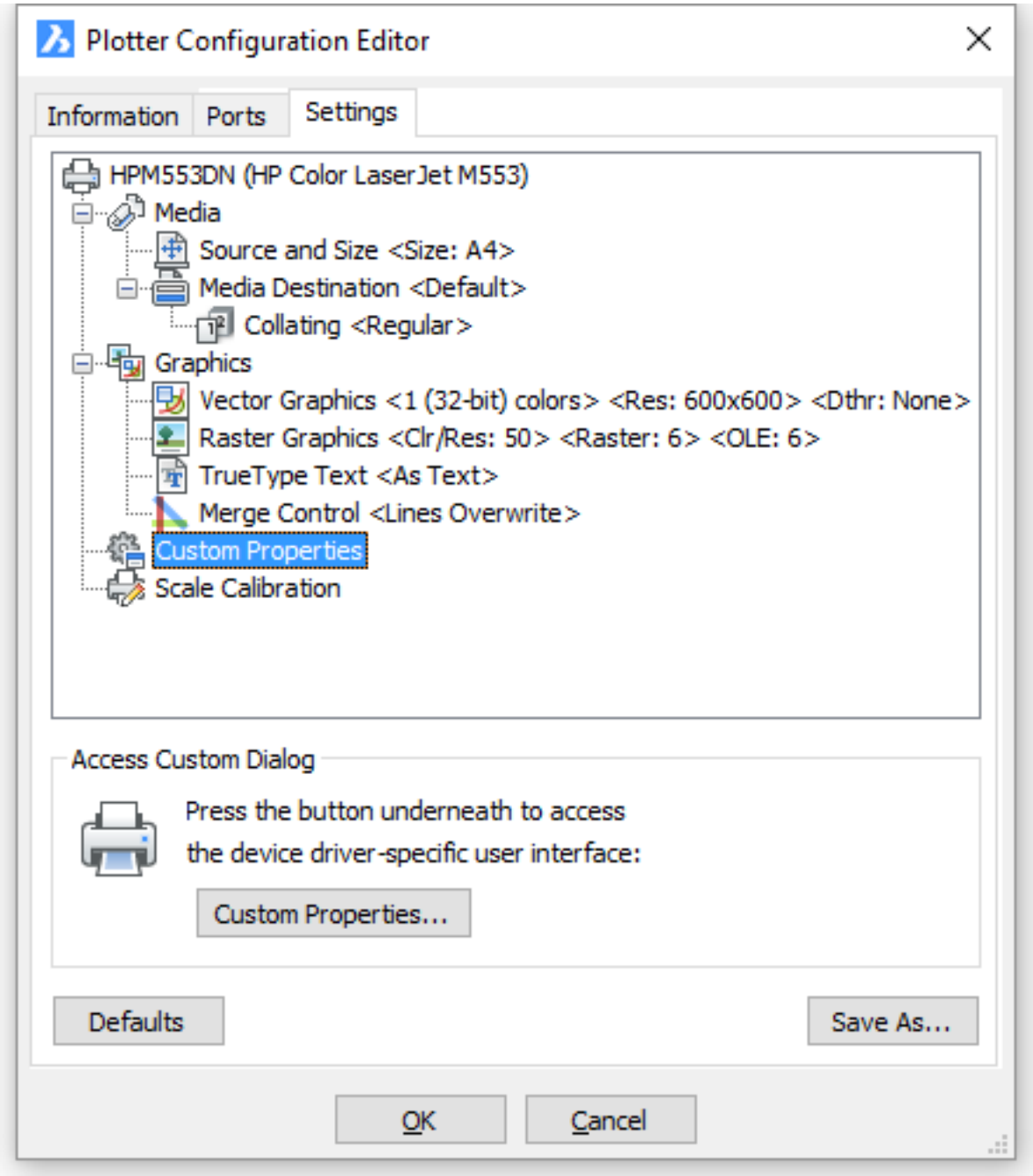
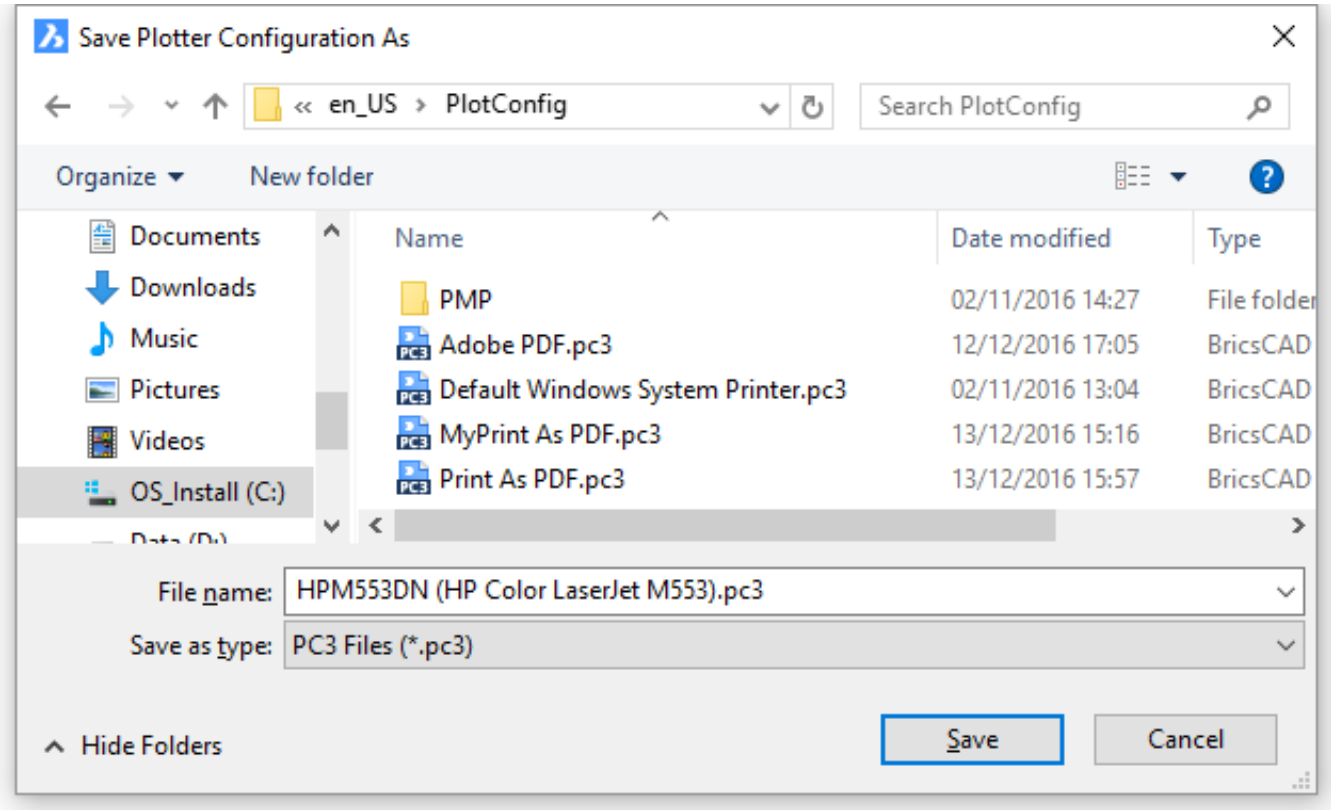


Figure 1.1151



To edit a plotter configuration

1. Do one of the following:
   - Choose **Page Setup** in the **File** menu.
   - Type **pagesetup** in the command bar, then press Enter.
2. Select a plotter configuration (PC3) in the **Printer / Plotter configuration** list.
3. Click the **Edit Plotter Configuration** button ( ) on the **Print** or **Page Setup** dialogs. The **Plotter configuration editor** dialog opens.
4. Proceed in the Create a plotter configuration (1.25.2) procedure starting from step 3.

NOTE	If you select a system printer in step 2, a new plotter configuration is created for the selected printer. When you click the **Edit Plotter Configuration** button ( ) in step 3, the **Properties** dialog of the selected printer opens first. After clicking the **OK** button on the **Properties** dialog the **Plotter configuration editor** dialog opens. You can then complete the Create a plotter configuration (1.25.2) procedure.

To assign a plotter configuration

1. Choose **Model** or the **Layout** to which you want to assign a specific plotter configuration file.
2. Do one of the following:
   - Choose **Page Setup** in the **File** menu.
   - Type **pagesetup** in the command bar, then press Enter.
3. Choose one of the following from the **Printer / Plotter configuration** list:
   - A system printer
   - A configuration file (PC3 file)
   - None (uses Default)
4. Click the **OK** button.

NOTE

- When BricsCAD™ cannot find the plotter configuration file assigned to the drawing, it changes the configuration file assigned to your drawing to **None** .
- The **None** printer device has its own set of paper sizes that can be stored in the drawing very much the same as for any 'real' printer. If a layout is opened and the stored printer is absent, the printer switches to **None** while the paper size is maintained. The missing printer and the previous paper size are indicated upon opening the **Print** dialog. The previous paper size can be saved in the drawing,
- PREVIOUS PAPER SIZE: drawings created by other parties are often set up for a printer that is not available in your office. In such case, BricsCAD resets the printer device to **None** , which results in using the default printer. Instead of resetting the paper size to a default, BricsCAD shows the Previous Paper size, in order to inform you about the intended paper size and allow to choose a similar size on any of the available printers.

Tutorial: Printing in BricsCAD: Plotter Configurations Click here (1.8.6) to watch

1.25.3 The Plot Style Table editor

https://www.youtube-nocookie.com/embed/DLVvwPWPhCg The **Plot Style Table Editor** can be opened, even if BricsCAD is not open.

To open the Plot Style Table editor

Do one of the following:

- Click the **Edit Plot Style** button () on the **Print** dialog.
- Choose **Plot Style Manager...** in the File menu, then double click a **CTB** or **STB** file.
- In a **Windows Explorer** window, double click a **CTB** or **STB** file.

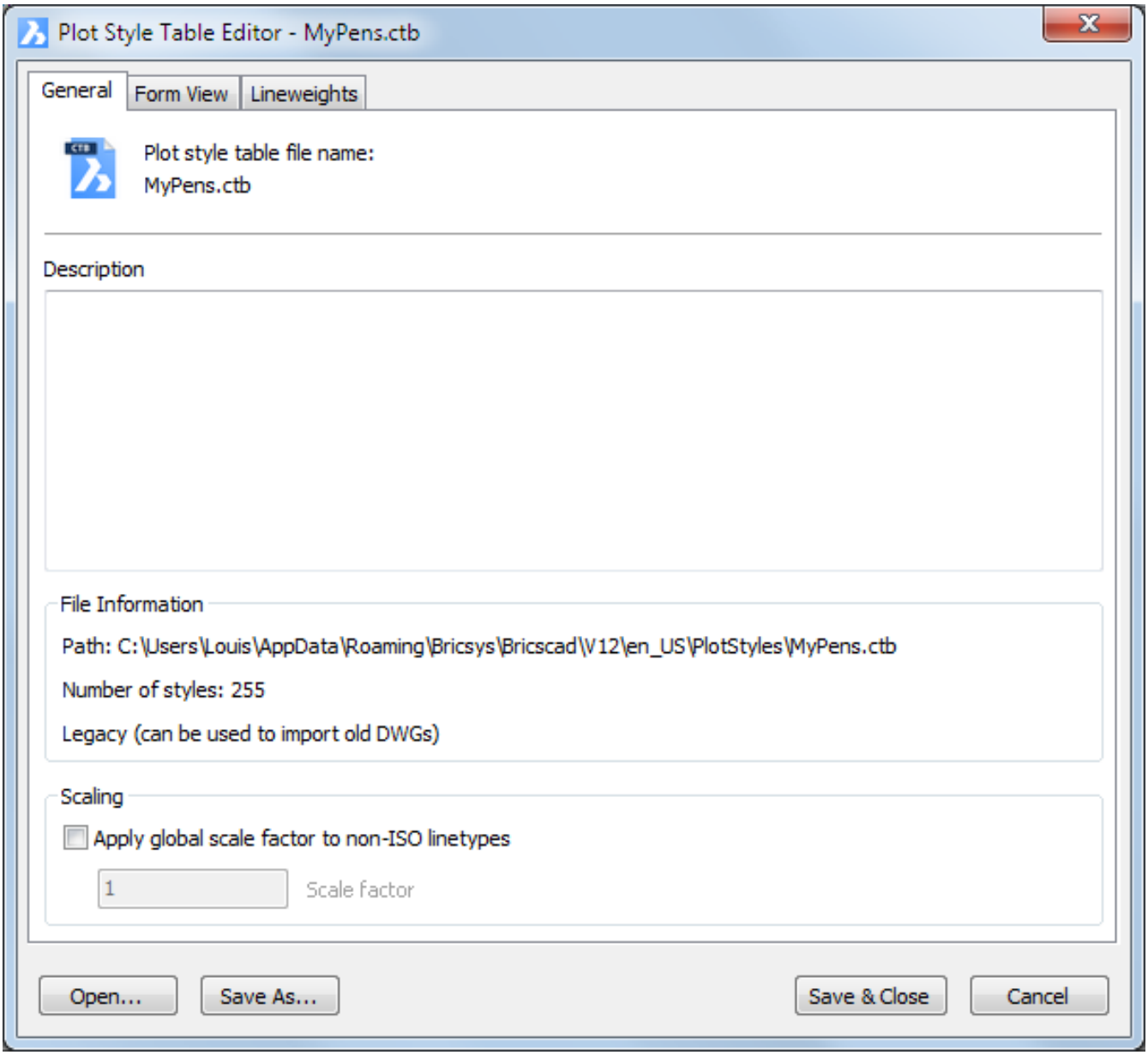
The **Plot Style Table Editor** displays the plot styles contained in the specified plot style table. The Plot Style Table Editor includes the following tabs:

- The **General** tab
- The **Form View** tab
- the **Lineweights** tab

The General tab The **General** tab lists general information about the plot style table. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART/04PlotStyleEditorGeneral.png})

Name	Description
Plot style table file name	Displays the name of the plot style table file you are editing.
Description	Displays a description you want to include about the plot style table.
File Info	Displays information about path, number of plot styles and the version number of the Plot Style Table Editor. **Note:** A color-dependent plot style table always contains 255 plot styles.
Apply global scale factor to non-ISO linetypes	Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table.
Scale factor	Specifies the factor by which to scale non-ISO linetypes and fill patterns.

Figure 1.1152



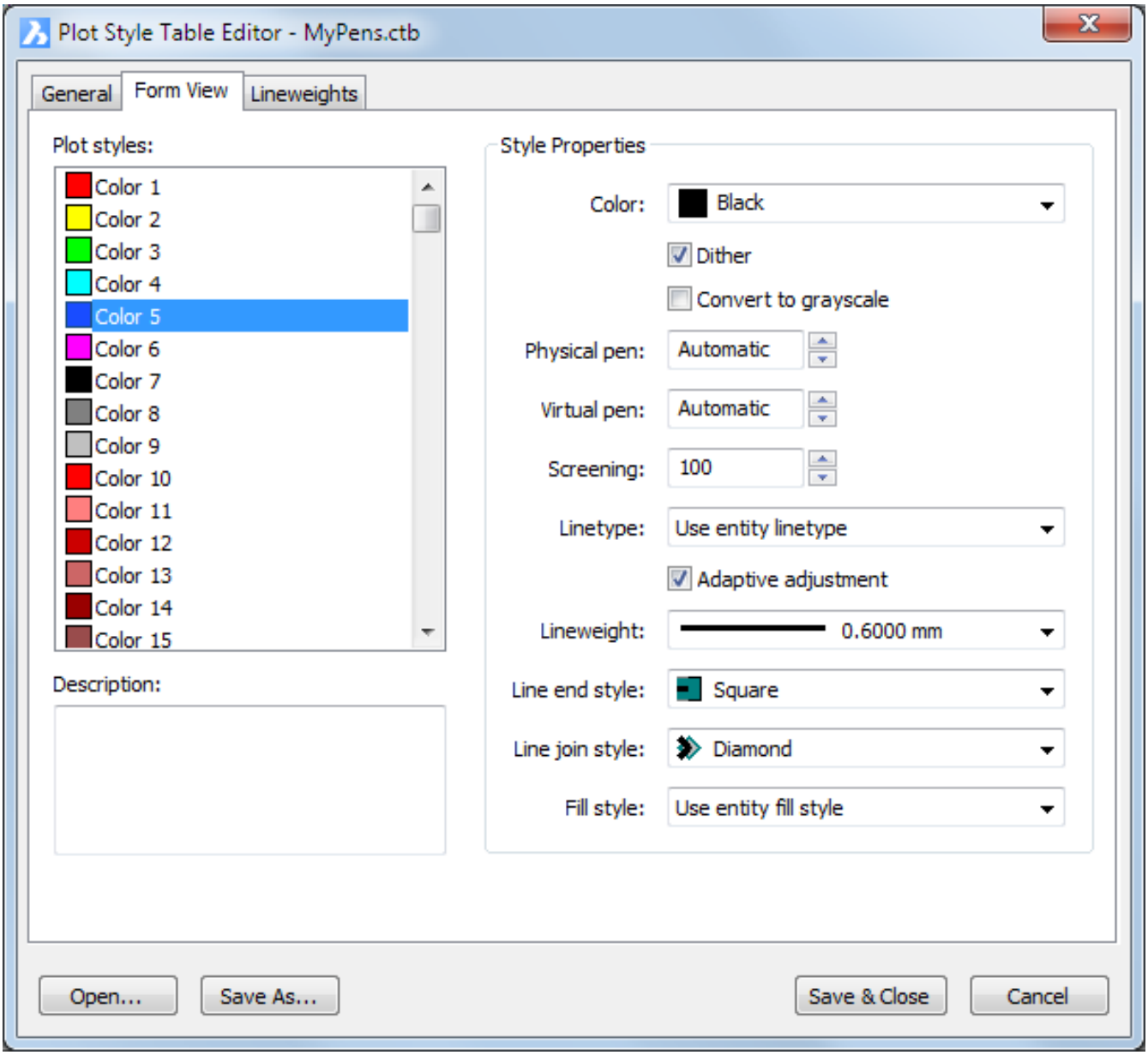
The Form View tab The **Form View** tab lists the plot styles contained in the plot style table and their settings. Plot styles are style overrides for your drawings that occur during plotting. The first plot style in a **Named Plot Style table** is **NORMAL** and represents an entity's default properties (no plot style applied). You cannot modify or delete the **NORMAL** style. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART/04Plo

Property	Description
Plot styles	Displays the names of plot styles contained in the selected plot style table. **Color-dependent plot style table:** • Always contains 255 styles (1 for each color). • They are tied to entity color. • You cannot add nor delete a plot style. • You cannot rename a plot style. **Named plot style table:** • Contains one or more plot styles. • You can add or remove plot styles, except for the **Normal** style, which cannot be deleted. • Plot styles can be renamed, except for the **Normal** style, which cannot be renamed.
Description	Displays a description for the selected plot style.
Color	Specifies the plotted color for an entity. The default setting for plot style color is **Use Object Color** . If you assign a plot style color, the color overrides the entity's color in the plot output. • Choose **Other...** to select one of 255 colors from the **Select Color** dialog box. • Choose **True color** to select a color from the **Color** dialog box. If the plot device does not support the color you have specified, the nearest available color will be used. When you have a monochrome device, **black** will be used.
Dither	Depending on the capabilities of your plotter, dithering approximates the colors with dot patterns. When this option is not active, the colors are mapped to the nearest color, resulting in a smaller range of colors when plotting. Dithering is available only whether you select the object's color or assign a plot style color. By default, this option is active. Setting it to Off has no effect in BricsCAD.
Grayscale	Depending on the capabilities of your plotter, converts the colors to grayscale in the plot output.
Pen #	Specifies a pen to use when plotting entities that use this plot style. You can select a pen number from 1 to 32.

Virtual pen #	Specifies a virtual pen number between 1 and 255. Choose **Automatic** to specify that BricsCAD should make the virtual pen assignment from the BricsCAD Color Index. The virtual pen setting in a plot style is used only by non-pen plotters and only if they are configured for virtual pens (select **255 Virtual Pens** under **Color Depth** under **Vector Graphics** on the **Settings** tab in the **Plotter Configuration Editor**).
Screening	Specifies the color intensity of the plot on the paper. The valid range is 0 through 100. If you select 100 the drawing will plotted with its full color intensity. In order for screening to work, the **Dithering** option must be active.
Linetype	Displays a list with an example and a description of each linetype. The default setting for plot style linetype is **Use Object Linetype** . The selected linetype overrides the entity's linetype in the plot output.
Adaptive	Adjusts the scale of the linetype to complete the linetype pattern. You can activate this property if a complete linetype pattern is more important than a correct linetype scaling.
Lineweight	Displays a sample of the lineweight as well as its numeric value. You can modify an existing lineweight if the one you need is not available. The default setting for plot style lineweight is **Use Object Lineweight** . The selected lineweight overrides the entity's lineweight in the plot output.
Line end style	Displays a list of several end styles for the lines. By default, the option **Use Object End Style** is selected. If you select a line end style, the object's line end style will be overridden in the plot output.
Line join style	Displays a list of several join styles for the lines. By default, the option **Use Object Join Style** is selected. If you select a line join style, the object's line join style will be overridden in the plot output.
Fill style	Displays a list of several fill styles. By default, the option Use **Object Fill Style** is selected. If you select a fill style, the object's fill style will be overridden in the plot output.

NOTE	Objects which are created in **true color** are always plotted using their own object properties (color, lineweight, ...).

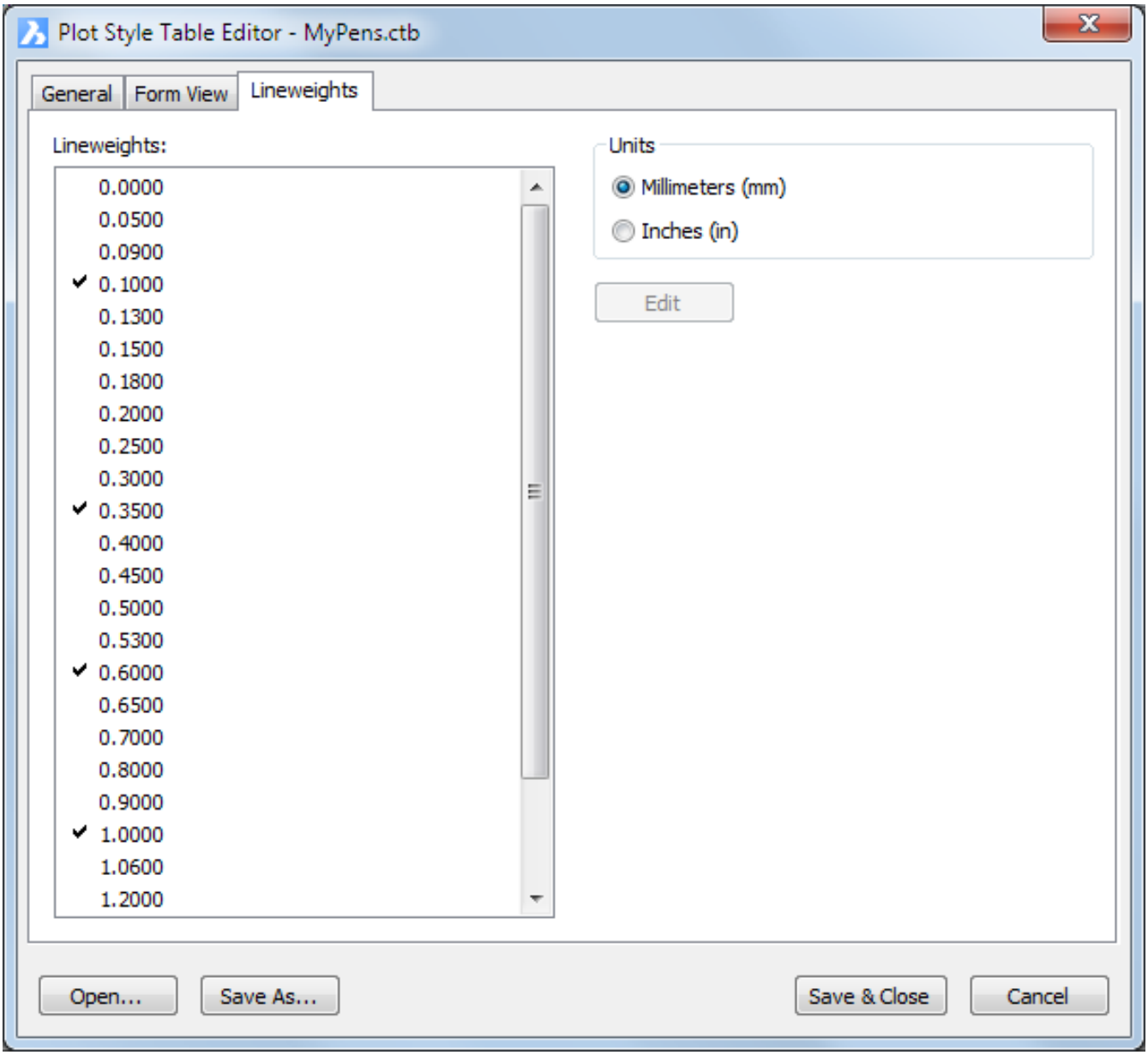
Figure 1.1153



The Lineweights tab The **Lineweights** tab lists the available lineweights. Lineweights with a tick are used in the **Lineweight** field on the **Form View** tab. To edit a lineweight, select the lineweight in the list, then click the **Edit** button.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART/04PlotStyleEditorLine

Tutorial: Printing in BricsCAD: Plot Style Tables Click here (1.8.5) to watch

Figure 1.1154



1.25.4 Assigning plot style tables

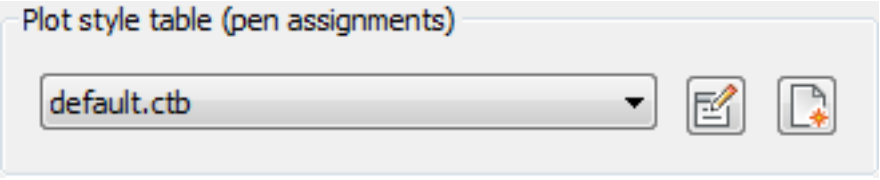
In BricsCAD you can assign plot style tables to model space and to each of the layouts in a drawing. By assigning different plot style tables to each layout in your drawing, you can control how entities in the layout are plotted. The plot style table affects both model space and paper space entities. To plot the drawing without applying plot style properties, choose 'None (use Default)' from the list of plot style tables. In STB-type drawings, each entity in the drawing either is assigned a plot style explicitly or **By Layer** .

Missing plot style table If a plot style table is not found, the text "(missing)" is added to the plot style table name. When you print the drawing, the settings of the default plot style file Default.ctb or Default.stb will be used instead.

To assign a plot style table

1. Choose **Model** or the **layout** to which you want to assign a different plot style table.
2. Do one of the following:
   - Choose **Print...** in the **File** menu.
   - Type **print** in the command bar, then press Enter.
3. Select a **Plot style table** from the list box in the **Plot Style Table (pen assignments)** section. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART/05PlotStyleTableName.png})
4. Click the **OK** button.

Figure 1.1155

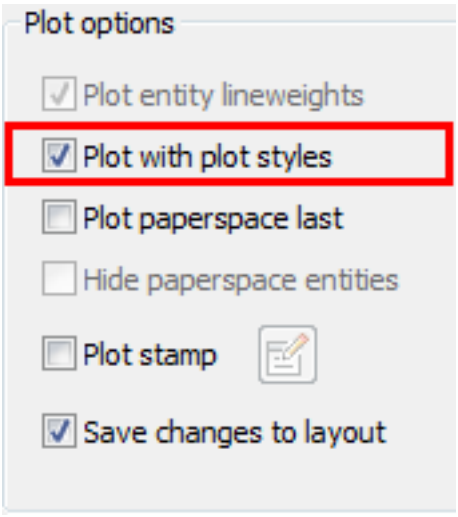


Toggling the plot style table on/off

When you turn off plot style tables, entities print according to their own properties. However, all of the plot style information is saved so you can easily turn on plot styles again. Actual plot style table files are not deleted, and for drawings that use named plot style tables, entities and layers retain their assigned plot styles.

1. Choose **Model** or the **layout** in which you want to turn the plot style table on/off.
2. Do one of the following:
   - Choose **Print...** in the **File** menu.
   - Type **print** in the command bar, then press Enter.
3. Click the **Plot with plot styles** plot option. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGu

Figure 1.1156



1.25.5 The Plot Styles Manager

Commands: STYLESMANAGER (6.27.55) The **Plot Styles Manager** is a window that lists plot style table files (CTB and STB files). The Plot Styles Manager allows you to:

- Create Color-dependent Plot Style (CTB) and Named Plot Style files (STB) using the **Create a Plot Style Table** wizard.
- Edit existing Plot Style Table files.

To open the Plot Styles Manager

Do one of the following:

- Choose **Plotstyle manager...** in the **File** menu.
- Type **stylesmanager** in the command bar then press Enter.

To create a new plot style table

1. Open the Plot Styles Manager (1.25.5).
2. Double click the **Create a Plot Style Table** shortcut. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
   The **Add Plot Style Table Editor** dialog opens. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us
3. Choose **Create new from scratch.** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sourc
4. Do one of the following:
   - Select either **CTB** or **STB** from the **Table type** selection button. **CTB** (C olor-dependent Plot Style T ab le) - bases plots on entity colors. Plotting parameters, such as pen thickness and linetype, are matched to the color of entities. This system is older and less flexible than named plot styles, but is the default in all new drawings. Creates a CTB file. **STB** (Named Plot S tyle T ab le) - bases plots on plot styles. Plotting parameters can be given to every entity and every layer. This system is new and more flexible, but may be incompatible with other software that reads DWG files. Creates an STB file.
   - Choose **Create from existing table** , then choose a file in the drop down list box or click the browse button to select an existing CTB or STB file.

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART/03PlotStyleTab
5. Type a name in the **Table name** field, then click the **OK** button.

The Plot Style Table Editor (1.25.3) dialog displays.

Figure 1.1157

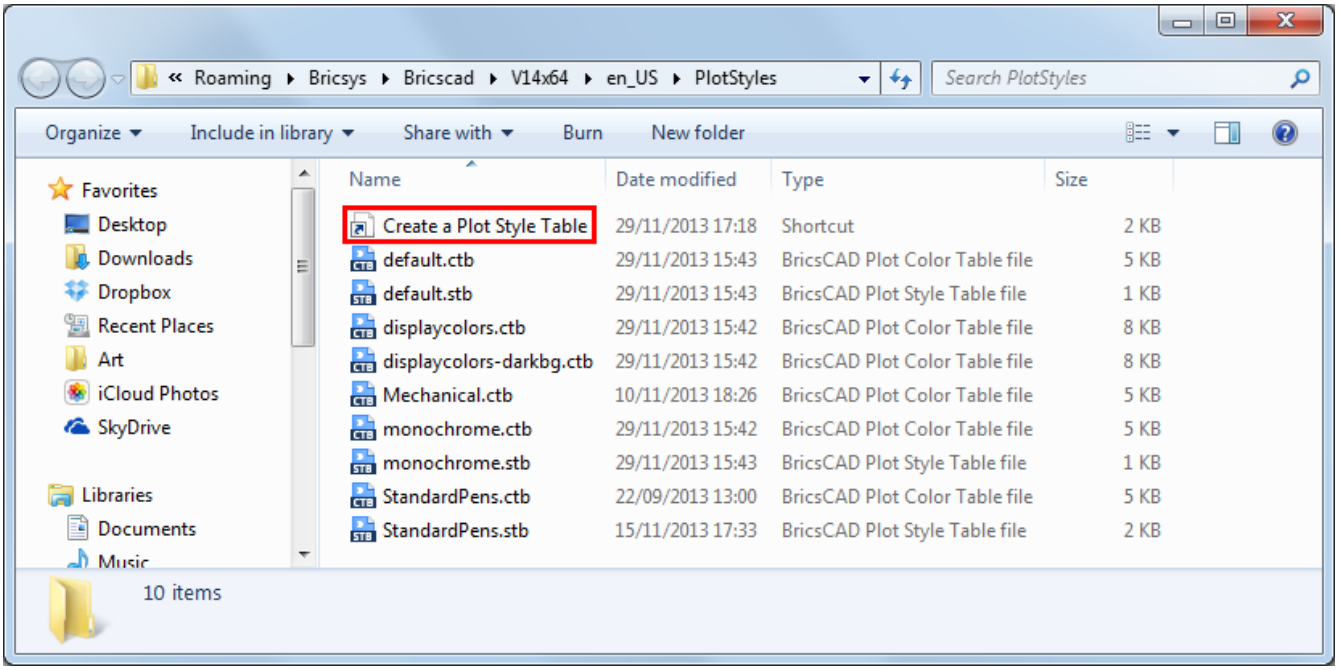


Figure 1.1158

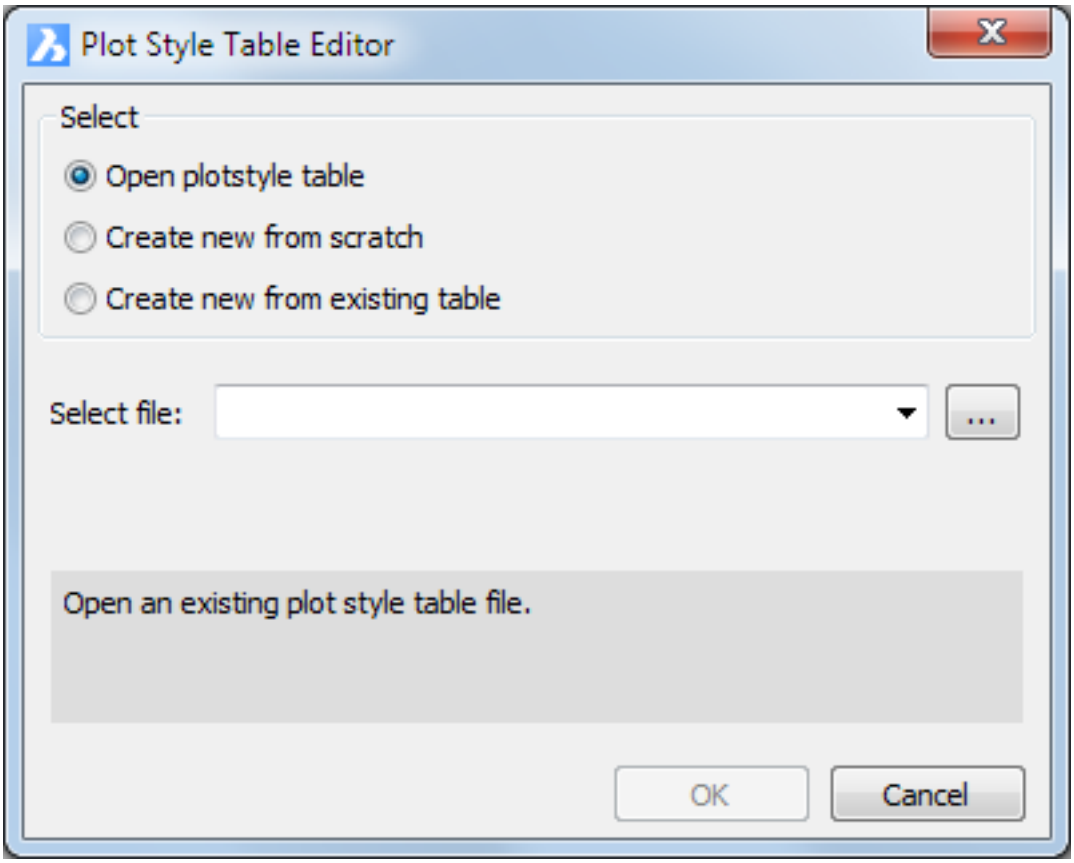


Figure 1.1159

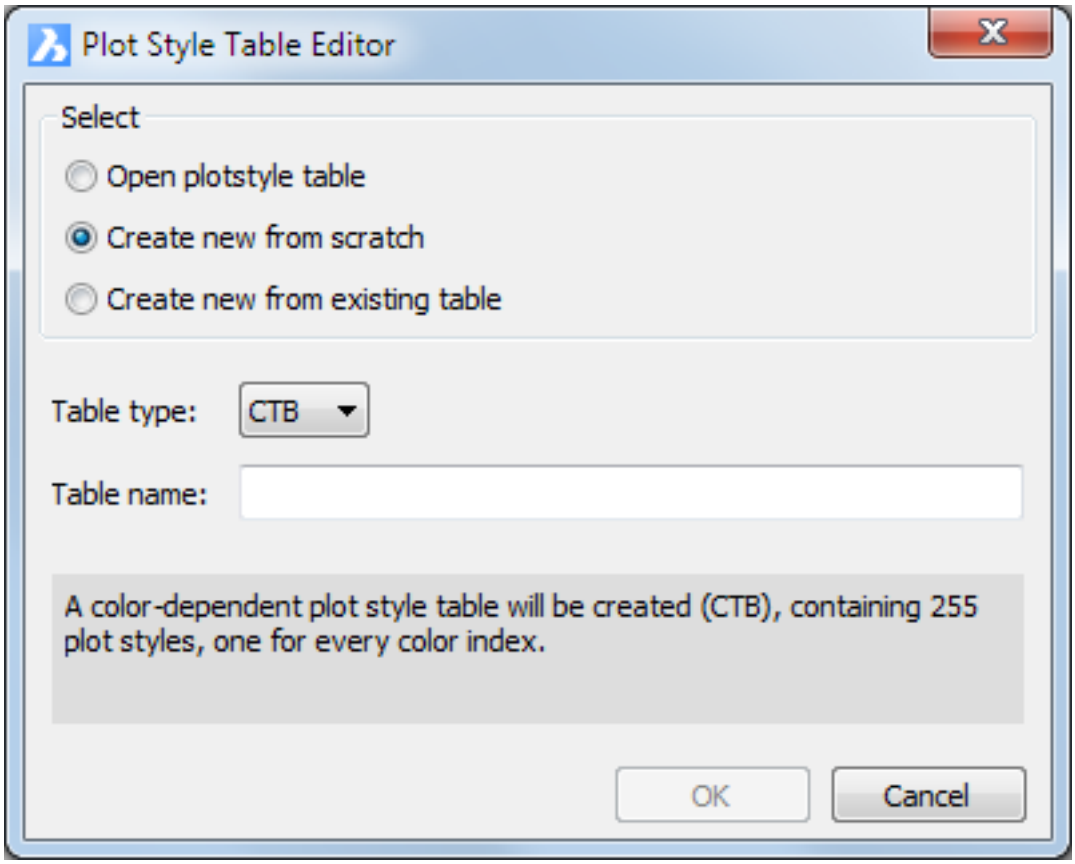
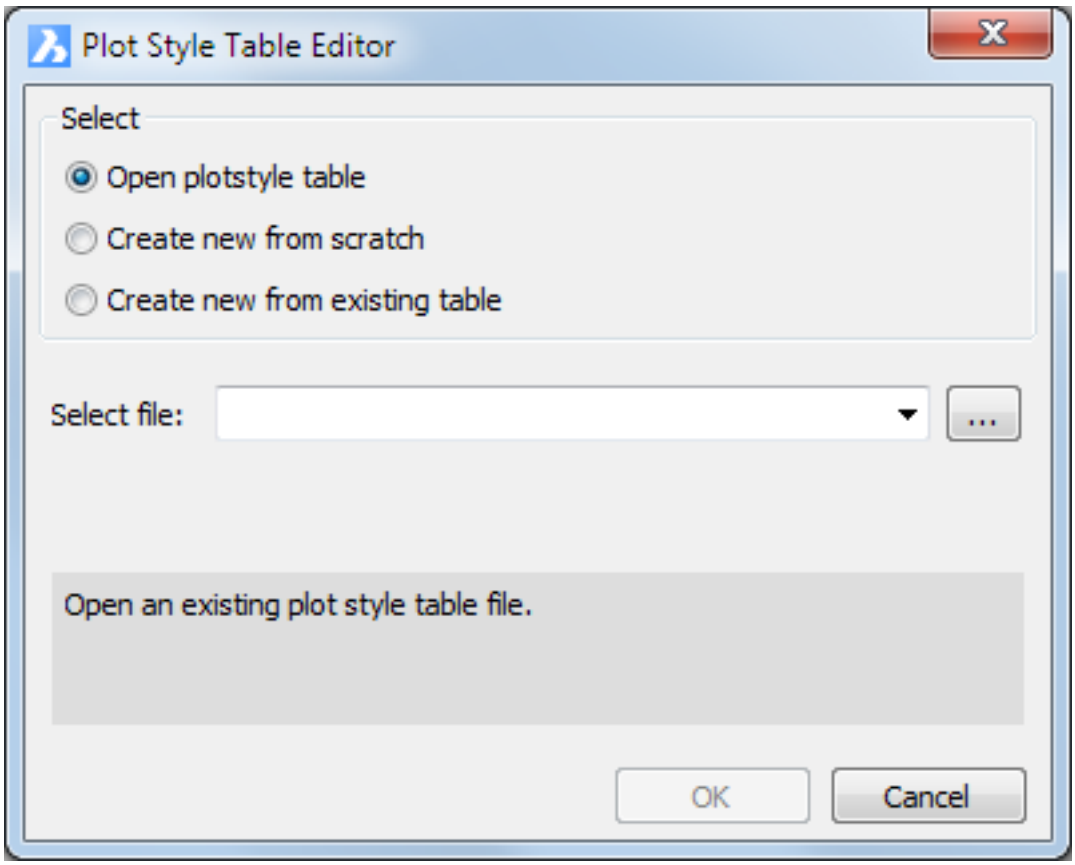


Figure 1.1160



To create a new plot style table in the Print dialog

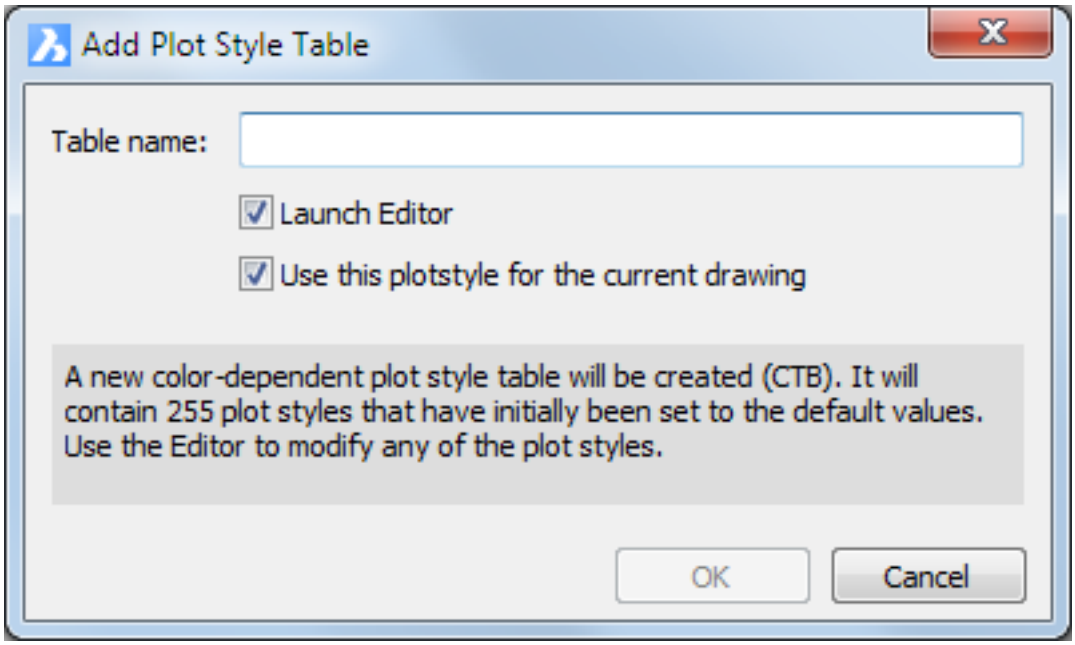
1. Open the Print dialog (1.25.1).

2. Click the **Create new plot style** button () on the **Print** dialog. The **Add Plot Style Table** dialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/15Printing/ART/03AddPlotStyle

3. Type a name in the **Table name** field, then click the **OK** button. If the Launch Editor option is chosen, the Plot Style Table Editor (1.25.3) dialog displays. If the **Use this plotstyle for the current drawing** option is chosen, the new plot style table is used in the current layout.

NOTE	The above procedure creates a CTB file in CTB-type drawing and a STB file in a STB-type drawing.

Figure 1.1161



To edit a plot style table

1. Open the Plot Styles Manager (1.25.5).
2. Double click the plot style table you want to edit.

The Plot Style Table Editor (1.25.3) dialog displays.

To edit a plot style table in the Print dialog

1. Open the Print dialog (1.25.1).
2. Select a plot style table from the **Plot Style Table** list button.
3. Click the edit **Edit plot style** button ( ).

The Plot Style Table Editor (1.25.3) dialog displays.

Tutorial: Printing in BricsCAD: Plot Style Tables Click here (1.8.5) to watch

1.25.6 Batch Plot

Commands: PUBLISH (6.24.55), -PUBLISH (6.24.56), SHEETSET (6.27.31) The **Publish** command allows to print a sheet list (= a list of model space or paper space layouts) to a printer. A sheet list can be saved to a DSD (Drawing Set Description) file. The **SheetSet** command manages sheet sets and allows to print, publish or etransmit a sheet set as a whole or a selection of sheets. A sheet set is saved to a DST file (Sheet Set Data) file.

1.26 Rendering

1.26.1 Rendering Overview

Commands:-RENDER (6.26.23),RENDER (6.26.22), SHADEMODE (6.27.29),RENDERPRESETS (6.26.24),ANIPATH (6.4.8) Rendering is the process of generating an image from a 3D model. The model contains the geometry, viewpoint, texture, lighting and shading information. The result is a digital image which can be saved as a raster image file: bmp, jpeg or png. Rendering in BricsCAD is based on theRedway3drendering engine. By default the rendering engine will try to leverage the hardware on the graphics card. If experiencing problems, please switch off theRenderUsingHardwareuser preference. It is recommended to install the most recent drivers for your graphics card. The IsRedSdkHardwareRenderingSupporteduser preference reports if Red SDK hardware rendering is fully supported and the initial value ofRenderUsingHardwareuser preference is set accordingly. TheRenderPresetscommand allows to create and edit render presets, and to set the current render preset through a dialog box.

1.26.2 Rendering

Commands: RENDER (6.26.22), SHADEMODE (6.27.29), RENDERPRESETS (6.26.24), ANIPATH (6.4.8) Rendering is the process of generating an image from a 3D model. The model contains the geometry, viewpoint, texture, lighting and shading information. The result is a digital image which can be saved as a raster image file: bmp, jpeg or png. Rendering in BricsCAD is based on the Redway3d rendering engine. By default the rendering engine will try to leverage the hardware on the graphics card. If experiencing problems, please switch off the **RenderUsingHardware** user preference. It is recommended to install the most recent drivers for your graphics card. The **IsRedSdkHardwareRenderingSupported** user preference reports if Red SDK hardware rendering is fully supported and the initial value of **RenderUsingHardware** user preference is set accordingly. The **RenderPresets** command allows to create and edit render presets, and to set the current render preset through a dialog box.

Setting the render preferences

1. Open the Settings dialog.

2. Under **Program Options** expand the **Rendering** settings category. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom Current material : Specifies the material for new entities. By default the **Global** material is used. Lighting units :

   - 0: No lighting units are used. Enables generic lighting.
   - 1: Enables American lighting units: Foot-Candles
   - 2: Enables International lighting units: Lux.

   Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 or 2. Default lighting : A distant light that follows the view direction. This setting can be different per viewport.

   - 0: Default lighting is used if no lights are switched ON.
   - 1: Default lighting is used instead of the light sources defined in the drawing.

Figure 1.1162

Rendering	
Current material	
Lighting units	[2] Enable photometric lighting using International units (lux)
Default lighting	[0] Default lighting is used only in case no other lights are switched on
Linear contrast	0
Linear brightness	0
Render using hardware	[1] Prefer hardware (faster)
Real world scale	☑ Render real-world scale materials
Sky status	[0] No sky
Texture map path	C:\ProgramData\Bricsys\RenderMaterials\textures;
Render material directory path	C:\ProgramData\Bricsys\RenderMaterials\materials\
Render material static directory path	C:\Program Files\Bricsys\BricsCAD V17 en_US\RenderMaterialStatic\
Tile mode light synch	☑ Synchronize lighting

Linear contrast : Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light. Linear brightness : Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport. Render using hardware : Choose **User software only (slower)** if hardware rendering fails. It might be necessary to restart BricsCAD after changing the setting. Real world scale : Controls the rendering of materials with units set to real-world scale Sky status :Specifies whether sky illuminiation is computed at render time. (Not implemented yet) Texture map path : Defines the search path for texture map images. In the BricsCAD program folder exist three subfolders under **Textures** , each containing a number of texture files of the same name. Images in folder 1 have a size of 256 x 256 pixels, folder 2 contains images of 512 x 512 pixels, images in folder 3 have a size of 1024 x 1024 pixels. If the **Diffuse map** setting of a material uses the image name only (not path), you can control the quality of a rendered image by setting the TextureMapPath user preference to folder 1, 2 or 3. Render material directory path : Specifies the path of the folder where the render materials for the Material Browser are installed. Render material static directory path : Specifies the path of the folder which contains read-only resources for predefined materials, such as their previews. Tile mode light synch : Controls the synchronization of lighting in all model space viewports. (Internal use only)

Rendering tools

Icon	Tool Name	Keyboard	Description
	Render	render	Photorealistic rendering using materials and lights
	Hide	hide	Hidden line view (*). If the previous shademode option was **2Dwireframe** , wireframe representation is restored if you zoom or pan. If the previous shademode option was other than**2Dwireframe,** the hidden line representation is kept until a different shade mode is chosen.
	Shade	shade	Shade s the entities between the polygon faces. Curved surfaces are segmented.
	2D Wireframe	shademode + 2	Wireframe representation
	3D Wireframe	shademode + 3	Wireframe representation
	Hidden	shademode + H	Hidden line view (*)
	Flat	shademode + F	Shade s the entities between the polygon faces. Curved surfaces are segmented
	Gouraud	shademode + G	Shade s the entities between the polygon faces. Curved surfaces are smoother and more realistic
	Flat + Edges	shademode + L	Combines flat shading and wireframe lines and curves

	Gouraud + Edges	shademode + O	Combines gouraud shading and wireframe lines and curves
	Materials	materials	Opens the Drawing Explorer - Materials
	Material Map	materialmap	Allows to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes.
	Light List	lightlist	Opens the Drawing Explorer - Lights
	Sun Properties	sunproperties	Opens the sun properties editor
	Geographic Location	geographiclocation	Opens the Geographic Location dialog.
	Animation Path	anipath	Records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.

(*) Set the **DISPSILH** system variable On to display silhouette lines instead of a polygon mesh for curved surfaces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04 Hidden line of a cone, a cylinder and a sphere with DISPSILH Off (left) and On (right).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04

Creating a rendered image of a model

The rendered image displays in the current viewport, in a separate window or can be exported to a file. The background of a rendered view can be controlled by the Background property of the current named view. The ANTIALIASRENDER system variable controls the amount of anti-aliasing (edge smoothing) applied during the rendering generation. The default value is 1, maximum value is 5.

NOTES

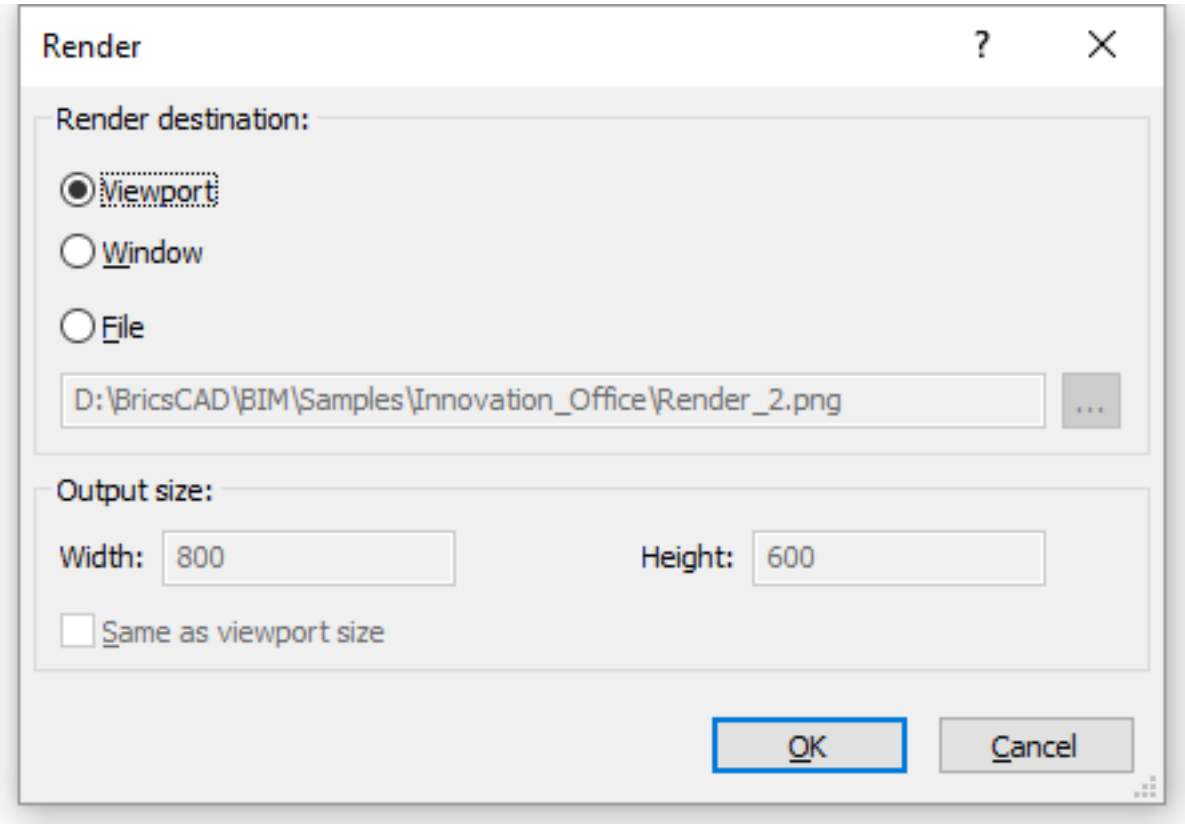
- High anti-alias values incur a high calculation cost.
- The values of the lighting system variables influence the rendering result.

To render a viewport

1. Do one of the following
   - Click the **Render** tool button ( ) on the **Rendering** toolbar.
   - Type **render** in the command bar, then press Enter.

   The **Render** dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04V
2. Click the **Viewport** option.

Figure 1.1163



3. Click the **OK** button. The view is rendered.

To render in a window

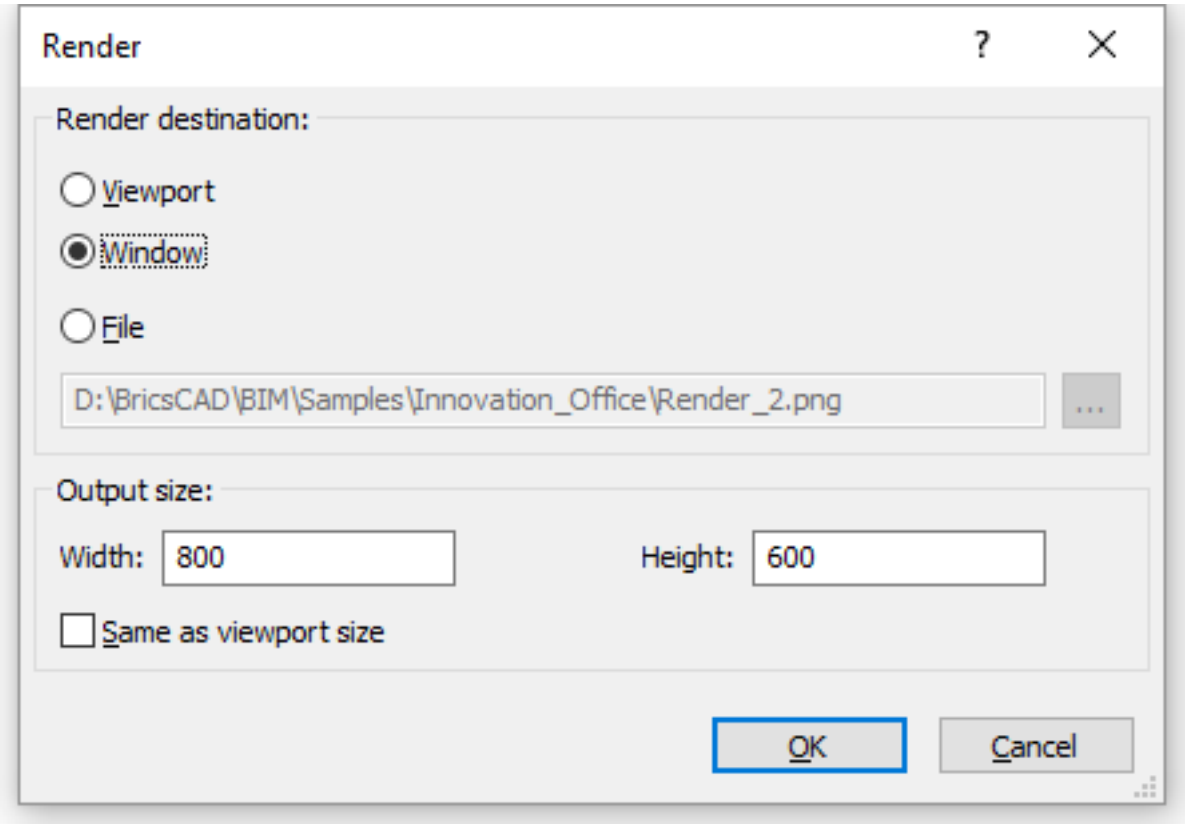
1. Do one of the following

   - Click the **Render** tool button ( ) on the **Rendering** toolbar.
   - Type **render** in the command bar, then press Enter.

   The **Render** dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04V

2. Click the **Window** option.
3. Click the **OK** button. The view is rendered in a popup window.

Figure 1.1164



To save a rendering to a file

1. Do one of the following

   - Click the **Render** tool button ( ) on the **Rendering** toolbar.
   - Type **render** in the command bar, then press Enter.

   The **Render** dialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04v

2. Click the **File** option.
3. Click the **Browse** button ( ), at the right hand side of the **File** field. The **Render output file** dialog displays.
4. Select a folder and type a name for the output file in the **File name** field.
5. Click the **Save as type** button to select the file type. The options are: **bmp**, **jpg** and **png**.
6. Click the **OK** button The **Render Output File** dialog closes.
7. Do one of the following:

   - Type the desired resolution for the output size in the **Width** and **Height** fields.
   - Check the **Same as viewport** size option to set the outsize to the size of the current viewport.
   - Click the **OK** button to create the file.

Figure 1.1165

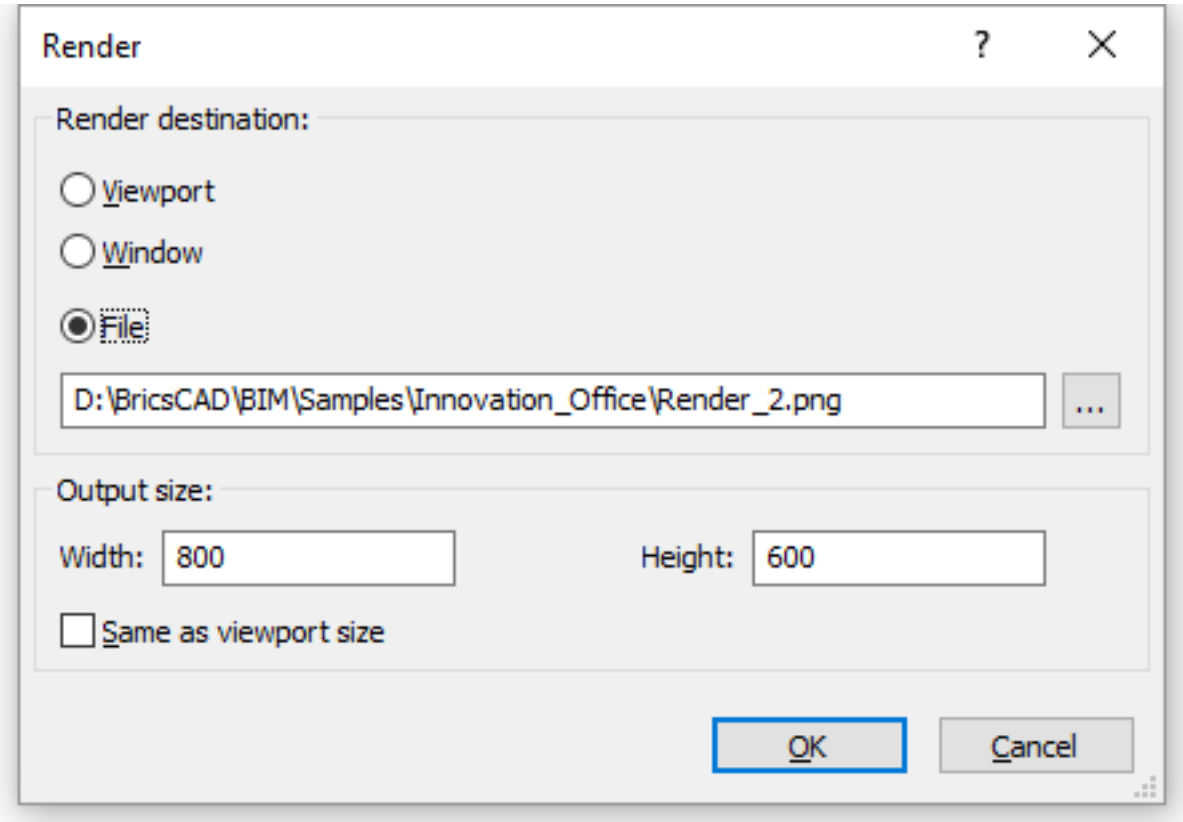


Figure 1.1166

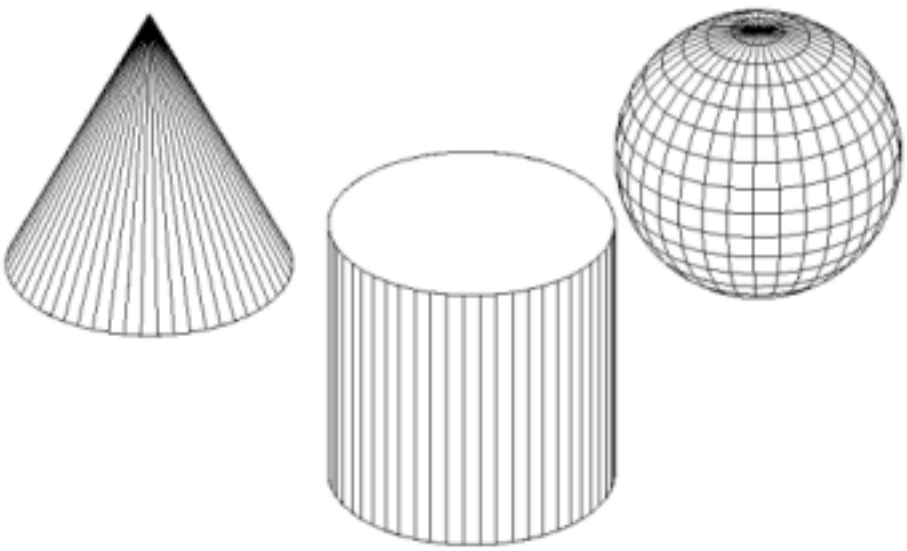
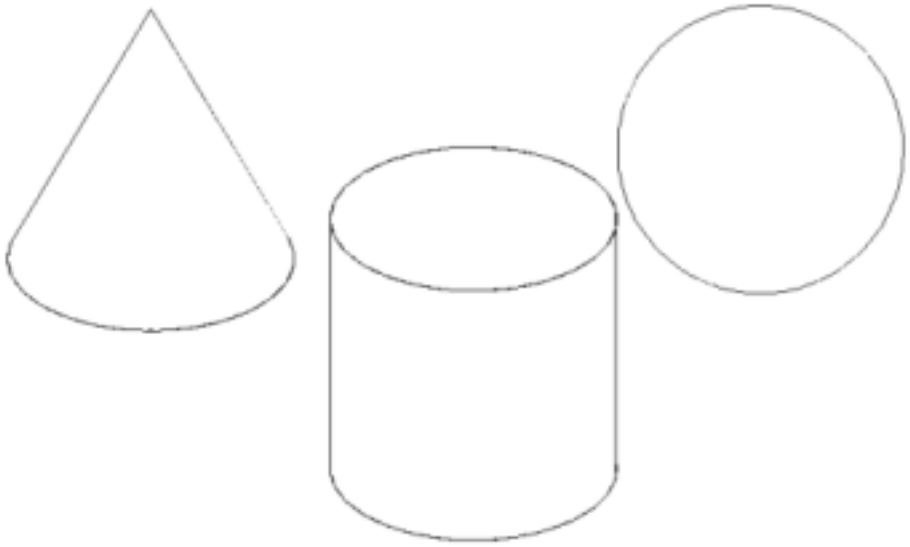


Figure 1.1167

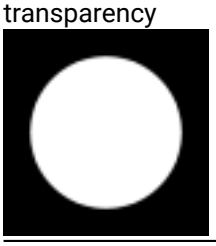
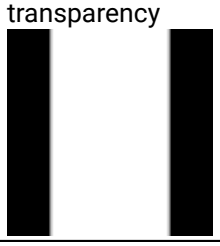


1.26.3 Rendering Materials

Command: MATERIALS (6.21.11), MATBROWSEROPEN (6.21.6), MATBROWSERCLOSE (6.21.5), MATERIALMAP (6.21.10) The **Materials** command creates materials and edits their properties through the Drawing Explorer. The **MatBrowserOpen** command opens the materials browser, which shows an overview of available high-resolution render materials. The **MatBrowserClose** command closes the materials browser. The **MaterialMap** command allows to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes. Material definitions are saved in the drawing. Each drawing contains the **Global** material. It is not possible to delete or rename the **Global** material. You can edit the properties of the **Global** material though.

Materials glossary

- Diffuse color : the color that the object reveals under pure direct, white light. It is perceived as the color of the object itself.
- Ambient color : the color of the object in the shadow, under ambient light rather then direct light. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/21Diffus
- Highlights : highly reflective highlights in the color of the material. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/20 Metallic Highlights (left) and Non-metallic Highlights (right)
- Shading : how the color and brightness of a surface varies with lighting.
- Texture-Mapping : adds detail to a surface which is not included in the 3D model itself.
- Diffuse map : adds a structure or pattern to a surface to simulate a material.
- Transparency map : sets the transparency of a texture from 0% (black) to 100% (white) and anything between (grey scales).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e transparency map:	(\vref {fig:C:/Dev/Task- transparency map:	(\vref {fig:C:/Dev/Task- transparency map:	
			

- Bump map : simulates a certain roughness or bumpiness on surfaces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help brick pattern: diffuse map only	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help brick pattern: diffuse map and bump map applied

- Reflectivity : the property of a surface to reflect light.
- Transparency : sharp transmission of light through solid objects.
- Translucency : scattered transmission of light through solid objects.
- Refraction : bending of light when passing through transparent objects.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help Refraction index = 1.000	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help Refraction index = 2.000

- Self-illumination : it appears as if a surface is emitting its own light. No light is cast on other objects.

Figure 1.1168

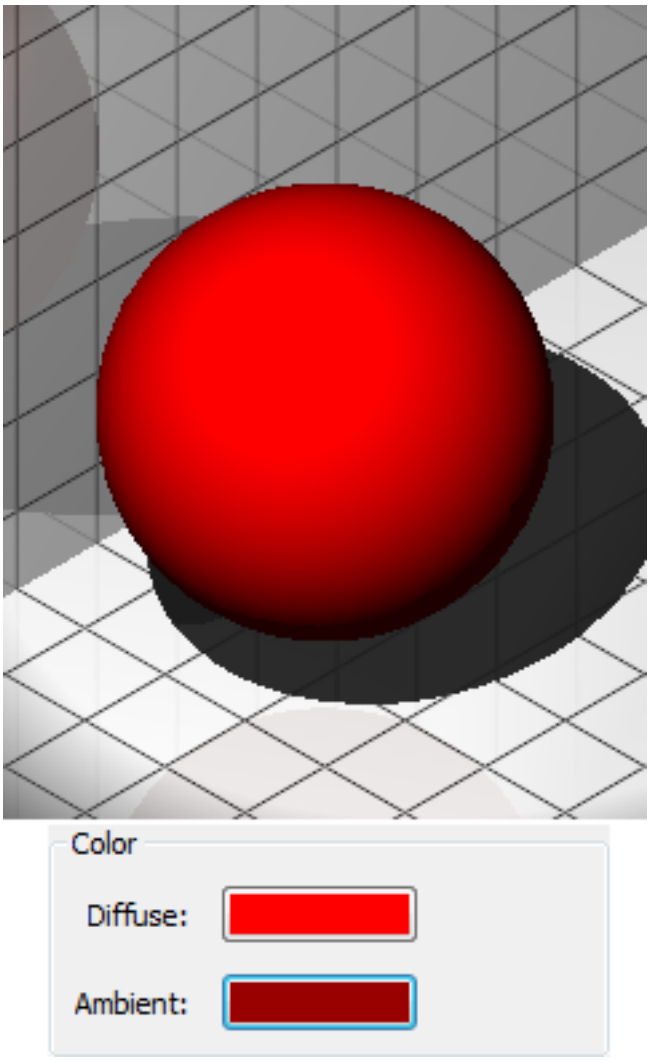


Figure 1.1169

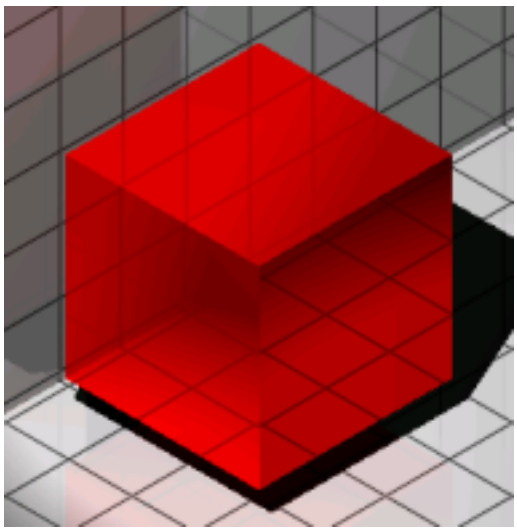
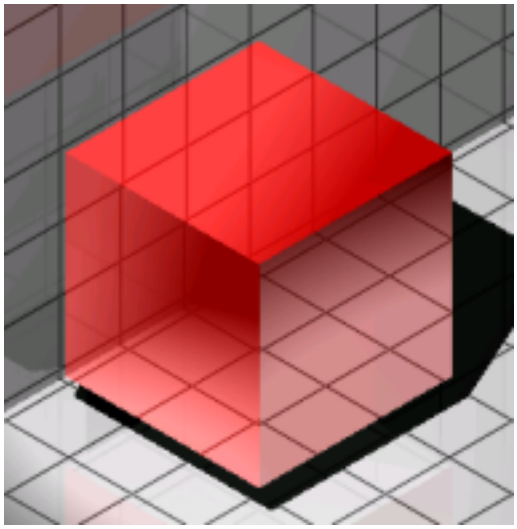


Figure 1.1170



Defining materials

See the Materials (6.21.11)command.

Assigning materials

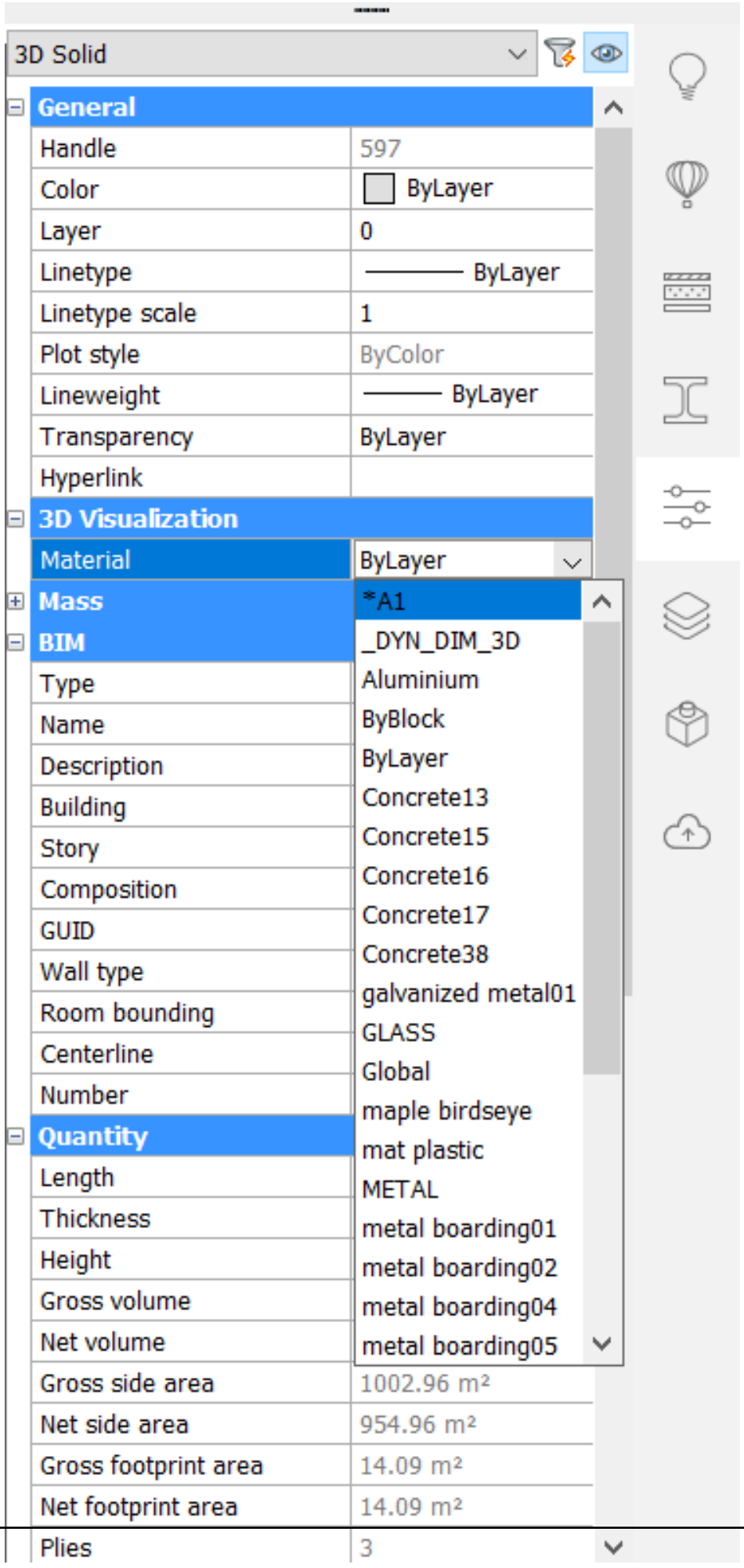
Materials can be assigned to a layer and to entities. If the **Material** property of an entity is **BYLAYER** . The material assigned to the layer of the entity will be used. By default new entities get the **BYLAYER** material setting.

To assign a material to entities

Method 1: Using the Properties Bar.

1. Select the entities or subentities (e.g. a faces a 3D solids).
2. In the Properties (1.12.4) bar, under **3D Visualization > Material** , click the down arrow and select the material in the **Material** list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyo

Figure 1.1171



Method 2: Using the Render Materials panel

To assign a material to a single entity:

1. Select the material in the Render Materials (6.21.6) panel.
2. Press and hold the left mouse button, then drag the material onto the entity.

To assign a material to multiple entities:

1. Select the entities.
2. Select the material in the Render Materials (6.21.6) panel.
3. Press and hold the left mouse button, then drag the material onto one of the selected entities.

Tutorial: Using Render Materials in BricsCAD Click here (1.6.32) to watch

To assign a material to a layer

1. Do one of the following

   - Click the **Explore Layers** tool button ( ) in the **Entity Properties** toolbar.
   - Choose **Layers > Layers...** in the **Settings** menu.
   - Type **layer** in the command bar, then press Enter.

   The **Drawing Explorer - Layers** dialog opens.

2. In the **Drawing Explorer - Layers** dialog click the **Detail view** tool button ( ).
3. Click the **Material** column of the layer you want to assign a material to.
4. Click the down arrow and select the material in the **Material** list.

Figure 1.1172

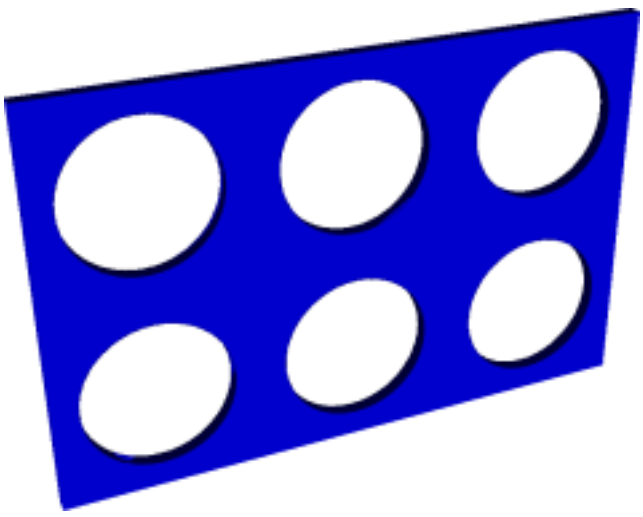


Figure 1.1173

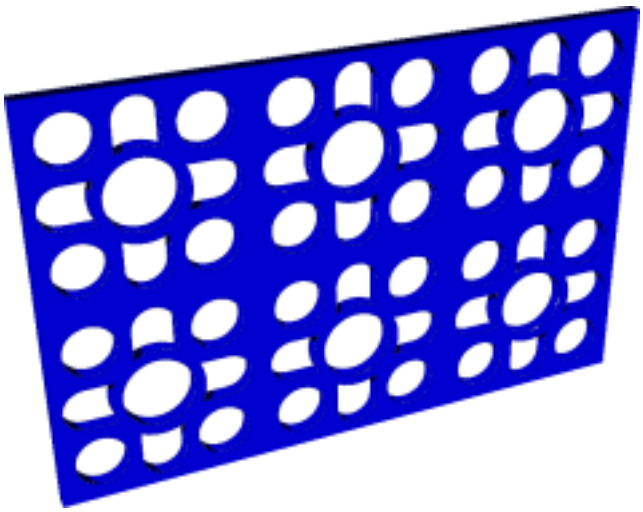


Figure 1.1174

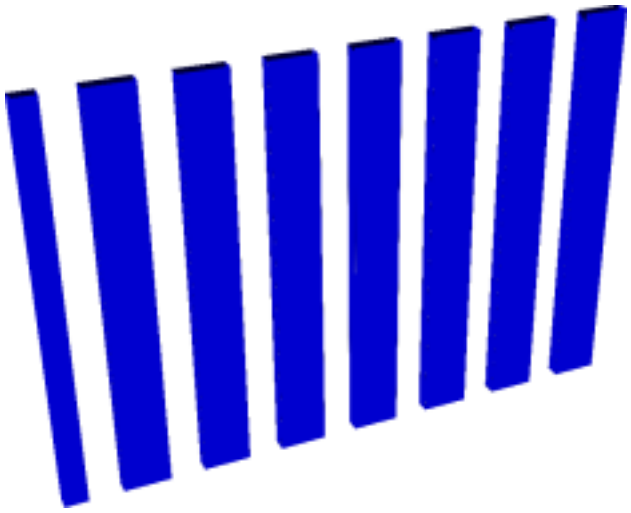


Figure 1.1175



Figure 1.1176

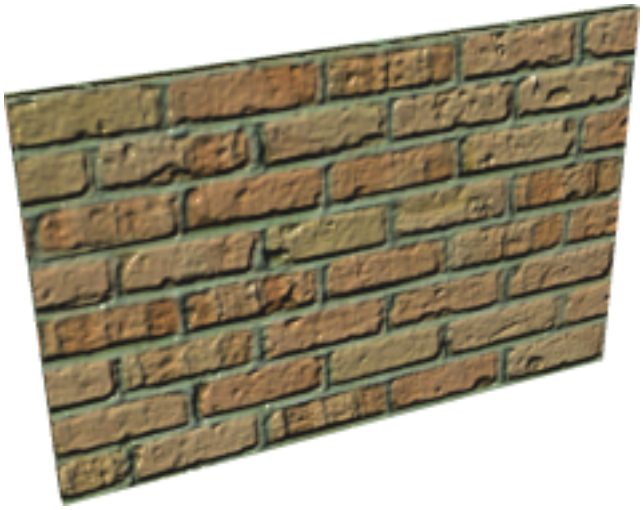


Figure 1.1177

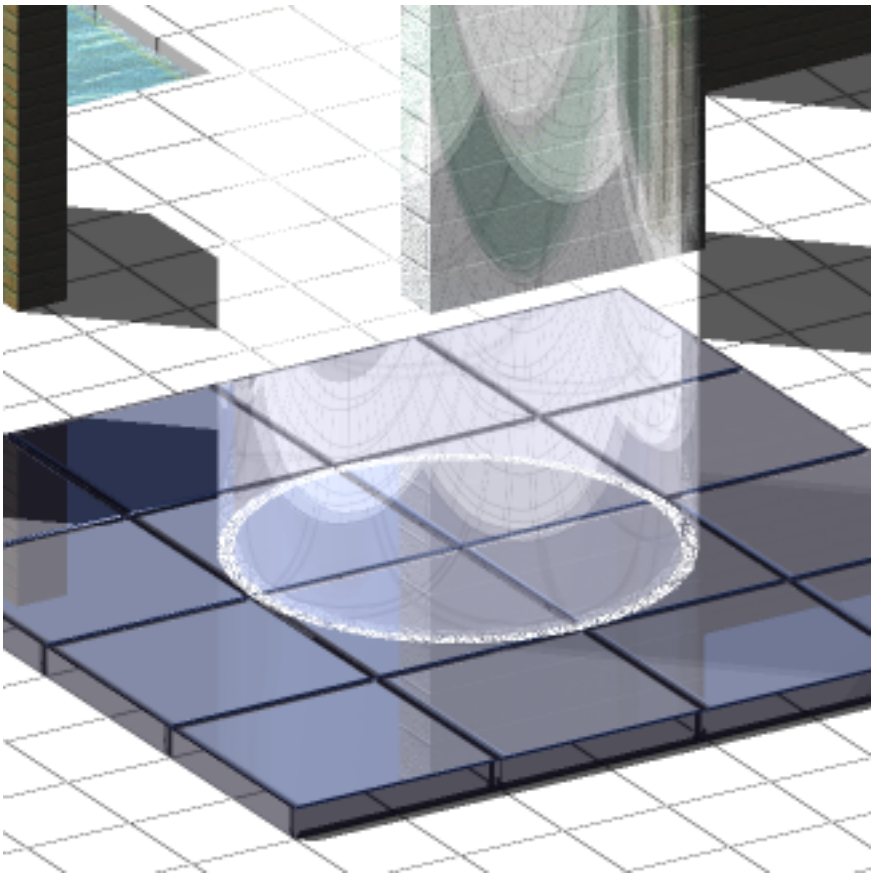
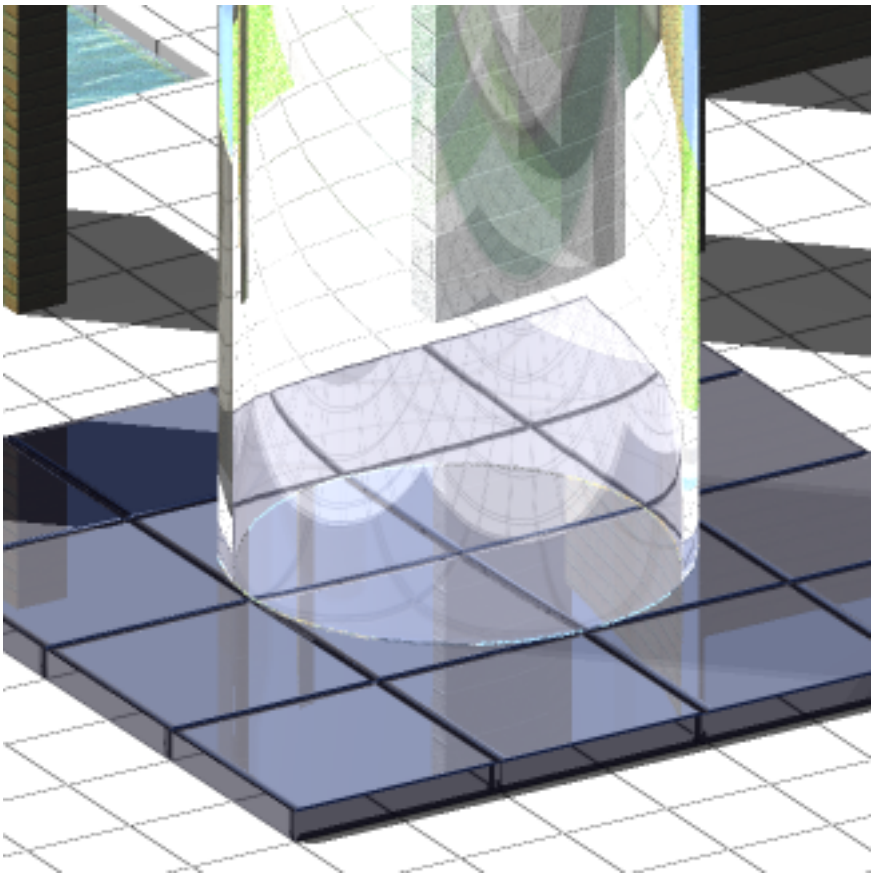


Figure 1.1178



1.26.4 Lights

Commands: DISTANTLIGHT (6.10.32), GEOGRAPHICLOCATION (6.15.15), LIGHT (6.20.28), LIGHTLIST (6.20.29), POINTLIGHT (6.24.42), SPOTLIGHT (6.27.42), SUNPROPERTIES (6.27.57), WEBLIGHT (6.33.6) Lighting is the key to make the viewer believe to look at a realistic scene.

Lighting glossary

- Ambient light: light from an unspecified source.
- Back light: light that adds depth and dimension by creating highlights.
- Distant light: light that is cast evenly throughout a scene with shadows all in the same direction.
- Fall-off: the attenuation or decrease in brightness the further from the light source.
- Fill light: the light that brightens dark areas and softens shadows from the main light.
- Point light: light that shines in all directions from a central spot.
- Spot light: focusable light that is aimed in one area.
- Photometric web: a 3D representation of the light intensity distribution of a light source.
- Web light: approximates real-world light distribution using a 3D representation of the light intensity. Web lights can be created only if the LIGHTINGUNITS (1.13.11) system variable is set to 1 (American lighting units) or 2 (International lighting units).
- Attenuation: the fall-off in brightness the further from the light source.

System variables

LIGHTINGUNITS (Lighting units):

- 0: No lighting units are used. Enables generic lighting.
- 1: Enables American lighting units: Foot-Candles
- 2: Enables International lighting units: Lux.

DEFAULTLIGHTING (Default lighting): A distant light that follows the view direction. This setting can be different per viewport.

- 0: Default lighting is used if no lights are switched ON.
- 1: Default lighting is used instead of the light sources defined in the drawing.

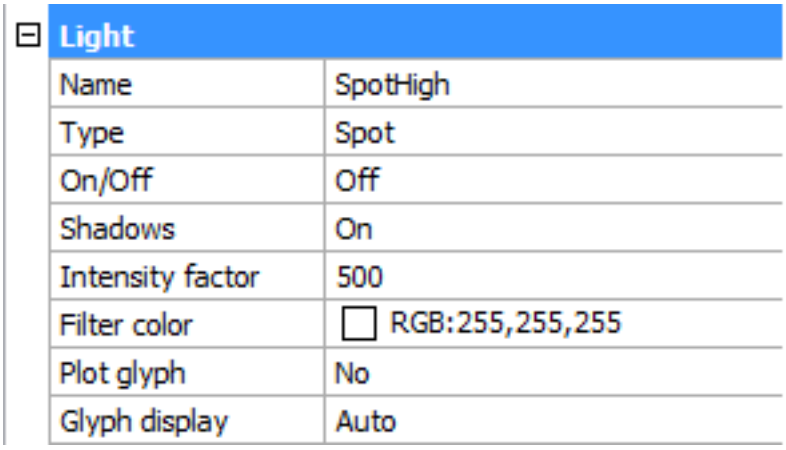
LINEARCONTRAST (Linear contrast): Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light. LINEARBRIGHTNESS (Linear brightness): Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport.

General properties

The general properties are common to all lights: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGu

- Name: a user defined name for the light.
- Type: defines the type of light: spot, point, web or distant.
- On/Off: specifies whether the light is turned on or off.
- Shadows: specifies whether the light casts shadows or not. Turning shadows off to increases performance.
- Intensity factor: multiplies the **Lamp Intensity** property, the result is the **Resulting Intensity**.
- Filter color: defines the color of the light.
- Plot glyph: if on, the light glyph is plotted.
- Glyph display: controls the display of the light glyph.

Figure 1.1179



Photometric light properties

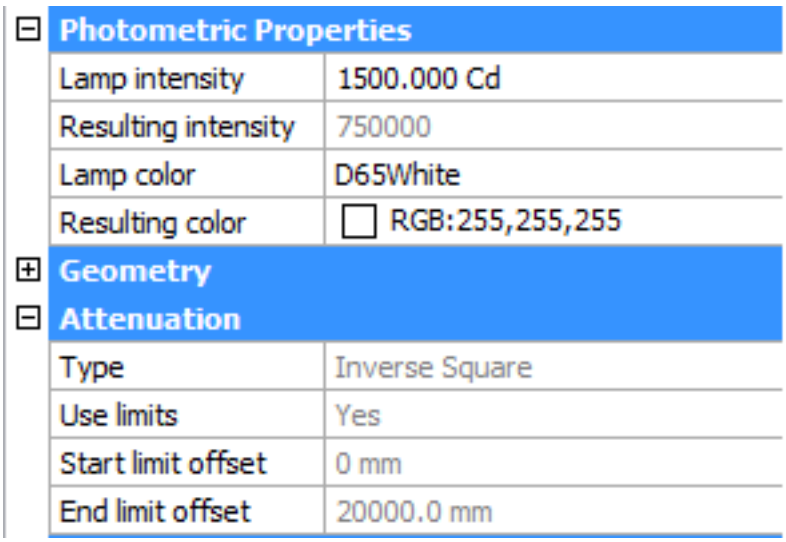
LIGHTINGUNITS = 1 (American units) or 2 (International units). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en

- Lamp intensity: Defines the brightness of the light. The lamp intensity is expressed in candela (cd) which is the SI base unit of luminous intensity: the power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function)
- Resulting intensity: the product of the lamp intensity and and intensity factor.
- Lamp color: defines the inherent color of the light or color temperature in Kelvin or standards.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en color temperatures	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en lamp color list

- Resulting color: this is the final color of the light: a combination of the lamp color and the filter color.
- Photometric web: If the light type is **Web**, **Photometric Web** and **Web offsets** are available (not implemented yet).

Figure 1.1180

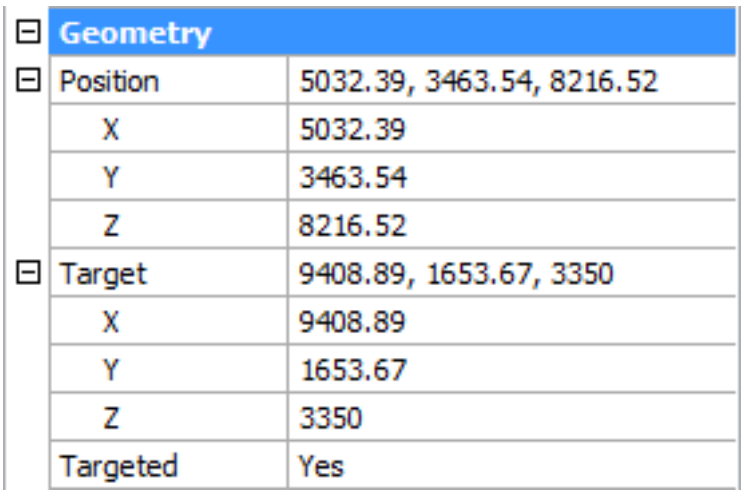


Geometry properties

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/22LightP

- Position: controls the location of the light.
- Target: defines the target point for point lights, spotlights and weblights.
- Targeted: switches the **target** property on/off.

Figure 1.1181



Attenuation properties

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/22LightP
In the real world, an object appears darker if it is farther away from the light source. The attenuation properties define how the light diminishes over distance. Attenuation does not apply to photometric lights. An alternative method to control the distance a light shines is the use of limits, which define the points from where a light starts to shine and where it stops. Using limits decreases the time needed to calculate the illumination of a scene. Attenuation properties apply to spot lights and point lights.

- Type: the options are **none**, **inverse linear** and **inverse square**
- None: no attenuation. The distance to the light source has no influence.
- Inverse Linear : the attenuation is the inverse of the linear distance from the light: at a distance of 2 units from the light source, light is half as strong; at a distance of 4 units, light is one quarter as strong.
- Inverse Square: the attenuation is the inverse of the square of the distance from the light: at a distance of 2 units, light is one quarter as strong; at a distance of 4 units, light is one sixteenth as strong.
- Use Limits: if this property is set to Yes, the **Start limit offset** and **End limit offset** fields become active.
- Start limit offset: defines the point where the light starts to shine, measured from the center of the light.
- End limit offset: defines the point where the light stops to shine, measured from the center of the light.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	(\vref {fig:C:/Dev/Task-	(\vref {fig:C:/Dev/Task-
No attenuation	Inverse linear attenuation	Inverse square attenuation

The **Attenuation** properties apply to generic lights only (LIGHTINGUNITS = 0). Inverse square attenuation is applied automatically for photometric lights (LIGHTINGUNITS = 1 or 2).

Figure 1.1182

Attenuation	
Type	Inverse Linear
Use limits	Yes
Start limit offset	0 mm
End limit offset	20000.0 mm

Rendered shadow details

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/22LightP

- Type: Sets the shadow type.
- Sharp (raytraced): generates sharp edged shadows.
- Soft (shadow map): generates realistic shadows with soft edges.
- Soft (sampled): Not supported yet
- Map size : specifies the amount of memory for shadow calculation. Click the field, then select a value in the drop-down list (64/128/256/512/1024/2048/4096).
- Softness : sets the softness of the shadow map. Enter a value between 1 and 10.

Figure 1.1183

Rendered Shadow Details	
Type	Soft (shadow map)
Map size	256
Softness	1

Spotlight hotspot and falloff properties

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/22LightP
When a surface is illuminated by a spotlight, there is an area of maximum illumination (hotspot = 1) that is surrounded by an area of lesser intensity (falloff = 2). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG

- Hotspot angle: angle of the central light cone (defines the hotspot)
- Falloff angle: angle of the full light cone

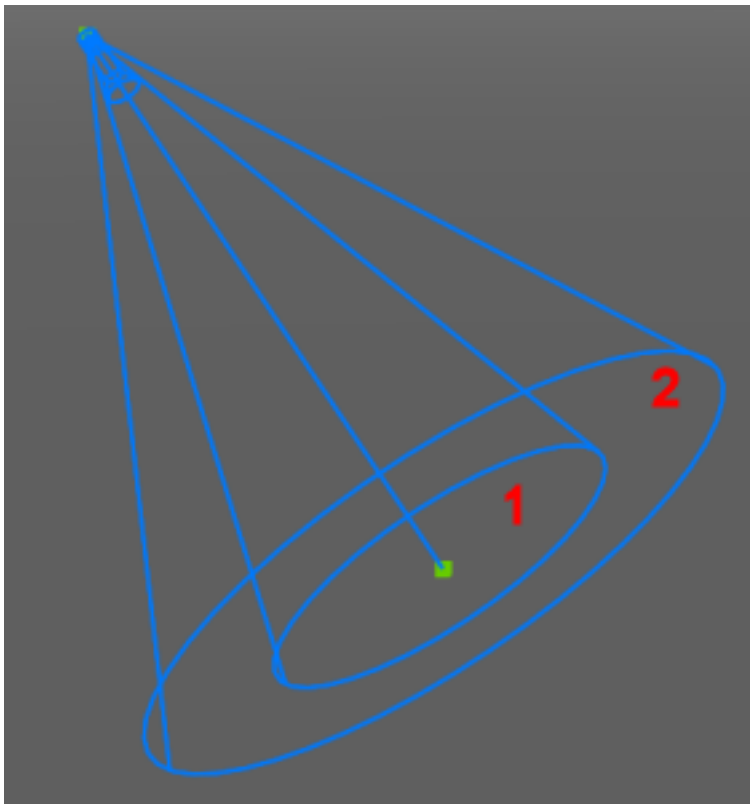
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/22SpotS Hotspot angle = 55, Falloff angle = 60	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/22SpotS Hotspot angle = 30, Falloff angle = 60

The difference between the hotspot angle and the falloff angle defines the area of lesser light intensity. If the falloff angle and the hotspot angle are almost equal the edge of light cone is rather sharp (left). The greater the difference between both angles, the softer the edge of the light cone (right).

Figure 1.1184

Hotspot angle	45
Falloff angle	50

Figure 1.1185



To open the light list in the Drawing Explorer

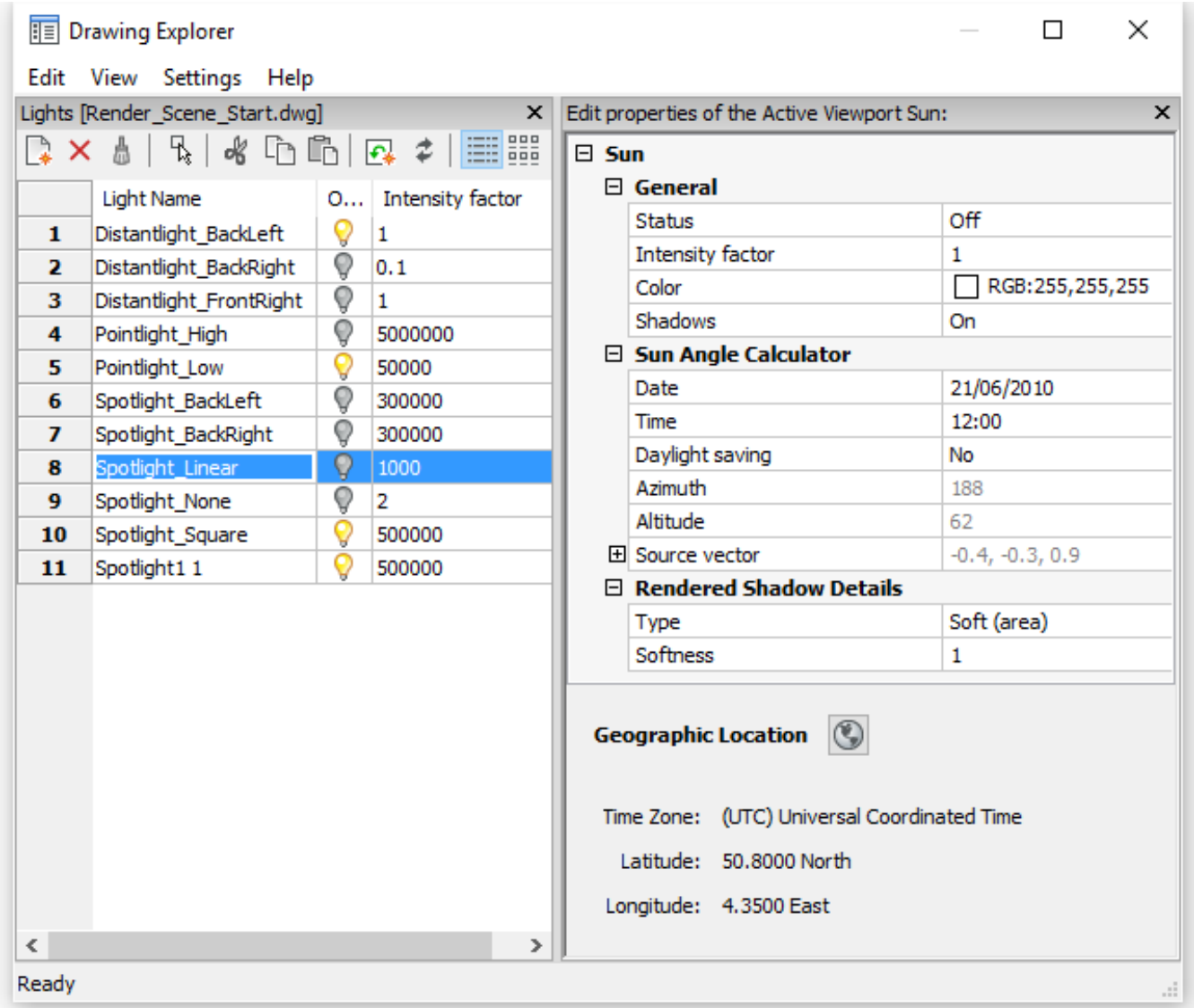
Do one of the following:

- Click the **Light List...** button ( ) on the **Lights** toolbar.
- Choose **Drawing Explorer > Lights** in the **Tools** menu.
- Choose **Light List..** in the **View | Rendering | Lights** menu.
- Type **lightlist** or **LL** in the command bar, then press Enter.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04Viewingyourdrawing/ART/20LightL

Icon	Tool name	Description
	New light	Closes the **Drawing Explorer - Lights** dialog, then guides you to the creation of a new light in the drawing.
	Delete light	Deletes the selected light.
	Select in drawing	Closes the **Drawing Explorer - Lights** dialog and selects the light's glyph in the drawing. The properties of the light display in the **Properties Bar**.
	Light ON	Click to switch light off.
	Light OFF	Click to switch light on.

Figure 1.1186



To define a light

1. Do one of the following:

   - Type **light** in the command bar, then press Enter. **You are prompted:** Enter light type [Point/Spot/Web/Distant] <Point>: A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/s
     Choose a light type in the prompt menu.
   - Click the tool button of the light type you want to create on the **Lights** toolbar:
     - New spot light
     - New point light
     - New distant light
     - New web light

2. Do one of the following:

   - Specify the source position in the drawing.
   - Type the coordinates of the source position in the command bar and press Enter.
   - Press Enter to accept the default coordinates.

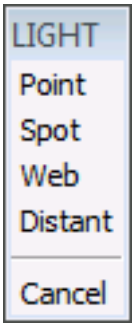
   You are prompted: Specify target position <0,0,-10>:

3. Do one of the following:

   - Specify the target position in the drawing.
   - Type the coordinates of the target position in the command bar and press Enter.
   - Press Enter to accept the default coordinates.

   You are prompted: Enter an option to change: Name/Intensity factor/Status/Photometry/Hotspot/Falloff/Shadow/Atten
   A prompt menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/04View

Figure 1.1187



4. Do one of the following:

   - Select **Name** in the prompt box.
   - Type **N** in the command bar and press Enter.

   You are prompted: Enter light name <Spotlight1>:

5. Do one of the following:

   - Type a name in the command bar and press Enter.
   - Press Enter to accept the default name.

   You are prompted: Enter an option to change: Name/Intensity factor/Status/Photometry/Hotspot/Falloff/Shadow/Atten

6. Do one of the following:

   - Click a light property in the prompt menu.
   - Type the capitalized letter of a light property in the command bar and press Enter.

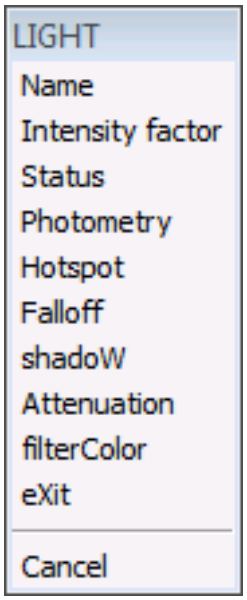
   You are prompted to define the selected property in the command bar.

7. Do one of the following to conclude the creation of the light:

   - Choose Exit in the prompt menu.
   - Press Enter.

NOTE	It is not possible to define a web light if the **LIGHTINGUNITS** system variable is zero (No lighting units).

Figure 1.1188



To edit a light

1. Select the light in the drawing. If the Properties bar is not open yet, double click the light. The properties of the selected light display in the Properties Bar:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sc	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/sc
Generic light properties	Photometric light properties

2. To edit a property:
   - Expand the settings group if needed.
   - Select the property
   - Edit the selected property.

To define the sun properties

1. Do one of the following:
   - Click the **Sun Properties...** tool button ( ) on the **Render** toolbar.
   - Type **sunproperties** or **sun** in the command bar, then press Enter.

   The **Drawing Explorer - Lights** dialog displays, showing the sun properties in the **Editor** pane.
2. To edit a property:
   - Expand the settings group if needed.
   - Select the property
   - Edit the selected property.

NOTE	When **LIGHTINGUNITS** = 1 or 2, the **Color** property of the sun cannot be edited.

To define the geographic location

1. Do one of the following:
   - Click the **Geographic location...** tool button ( ) on the **Render** toolbar.
   - Click the **Geographic location...** tool button ( ) on **Editor** pane of the **Drawing Explorer - Lights** dialog.
   - Type **geographiclocation** or **geo** in the command bar, then press Enter.

   The **Geographic Location** dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGu

Figure 1.1189

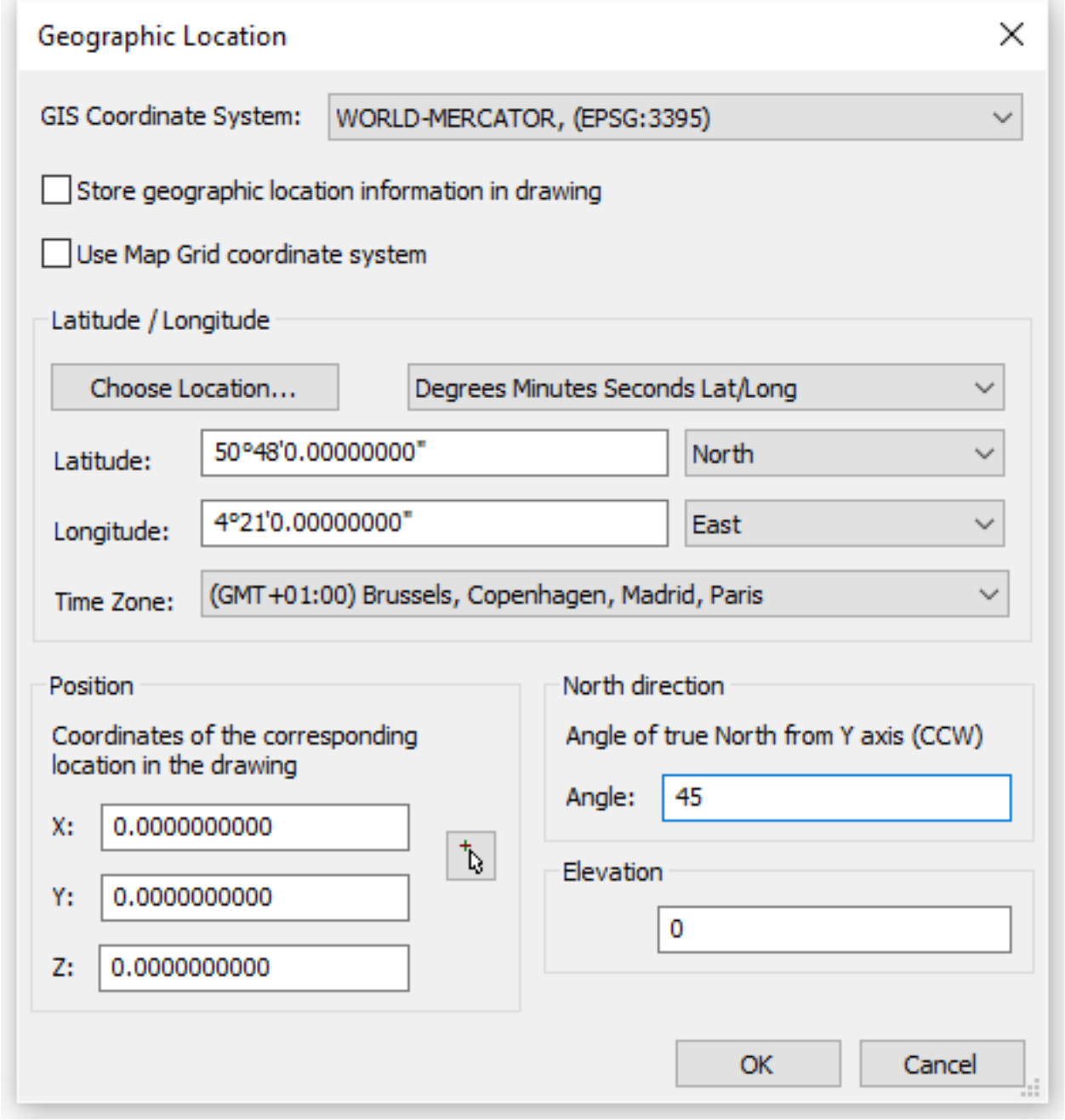


Figure 1.1190

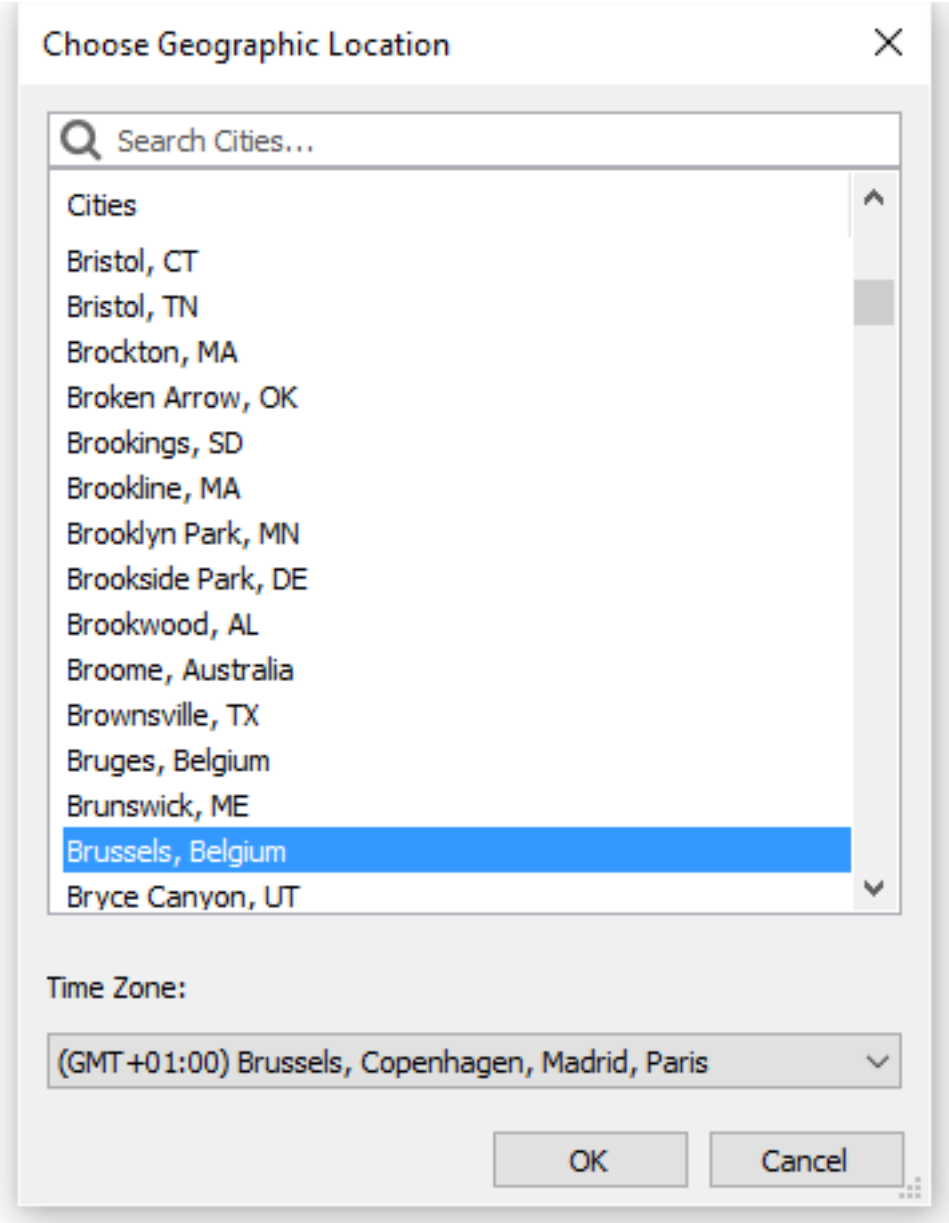


Figure 1.1191

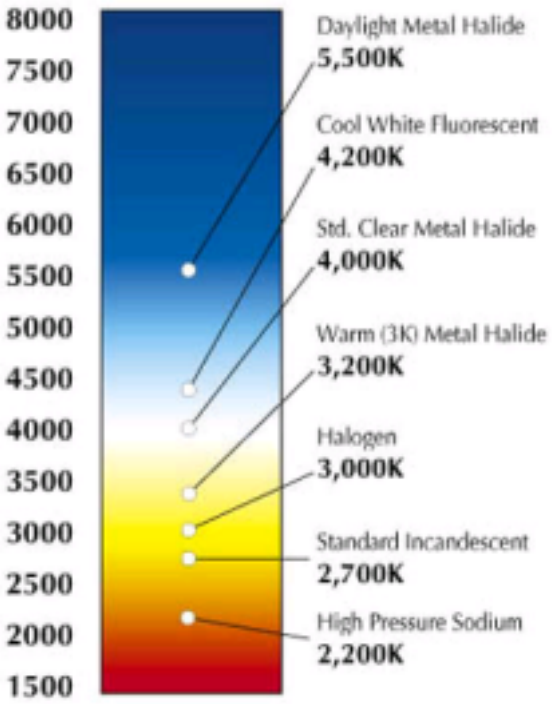


Figure 1.1192

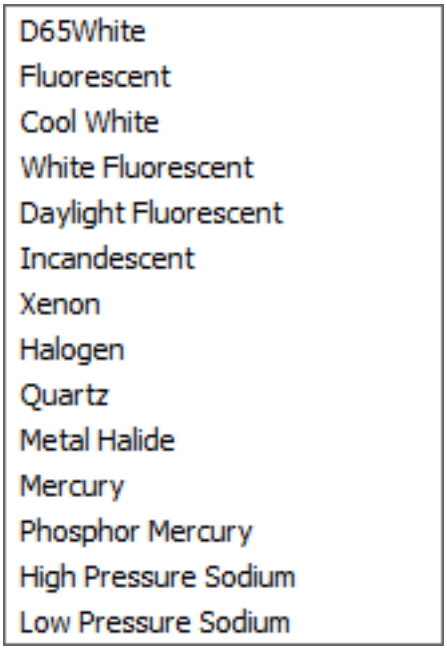


Figure 1.1193

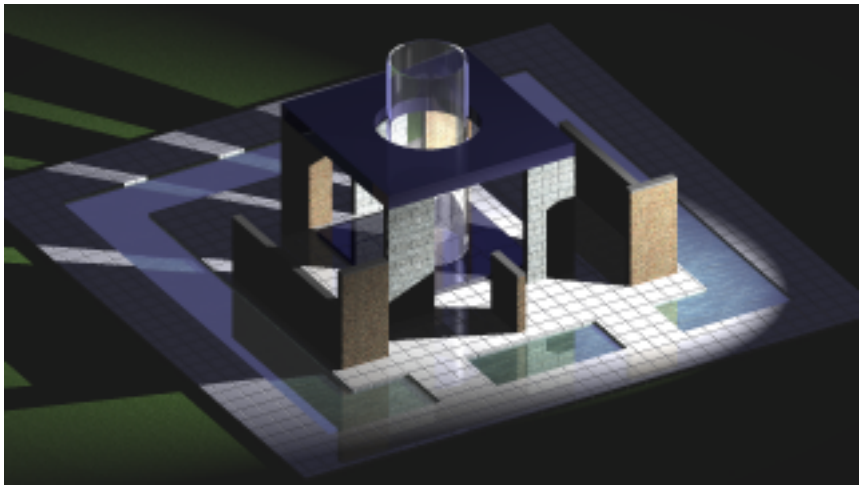


Figure 1.1194

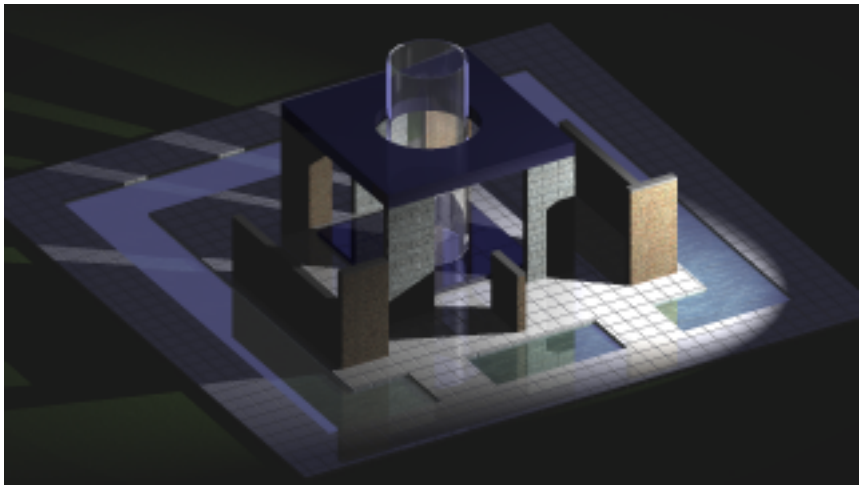


Figure 1.1195

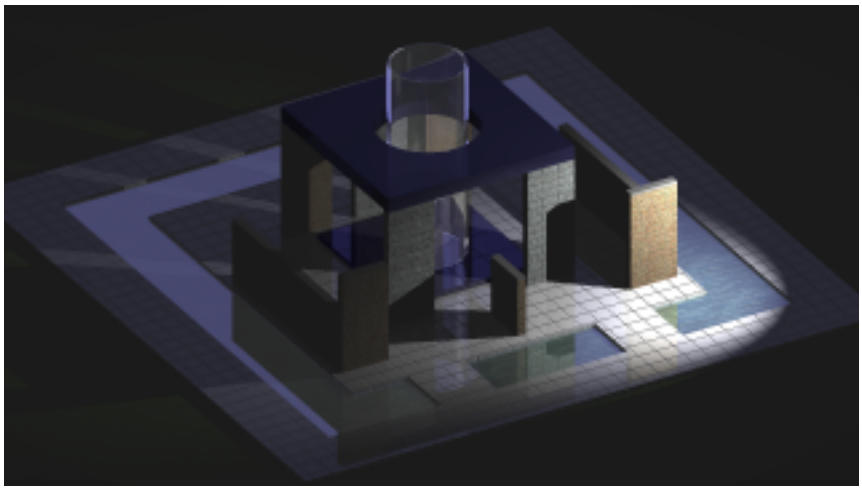


Figure 1.1196

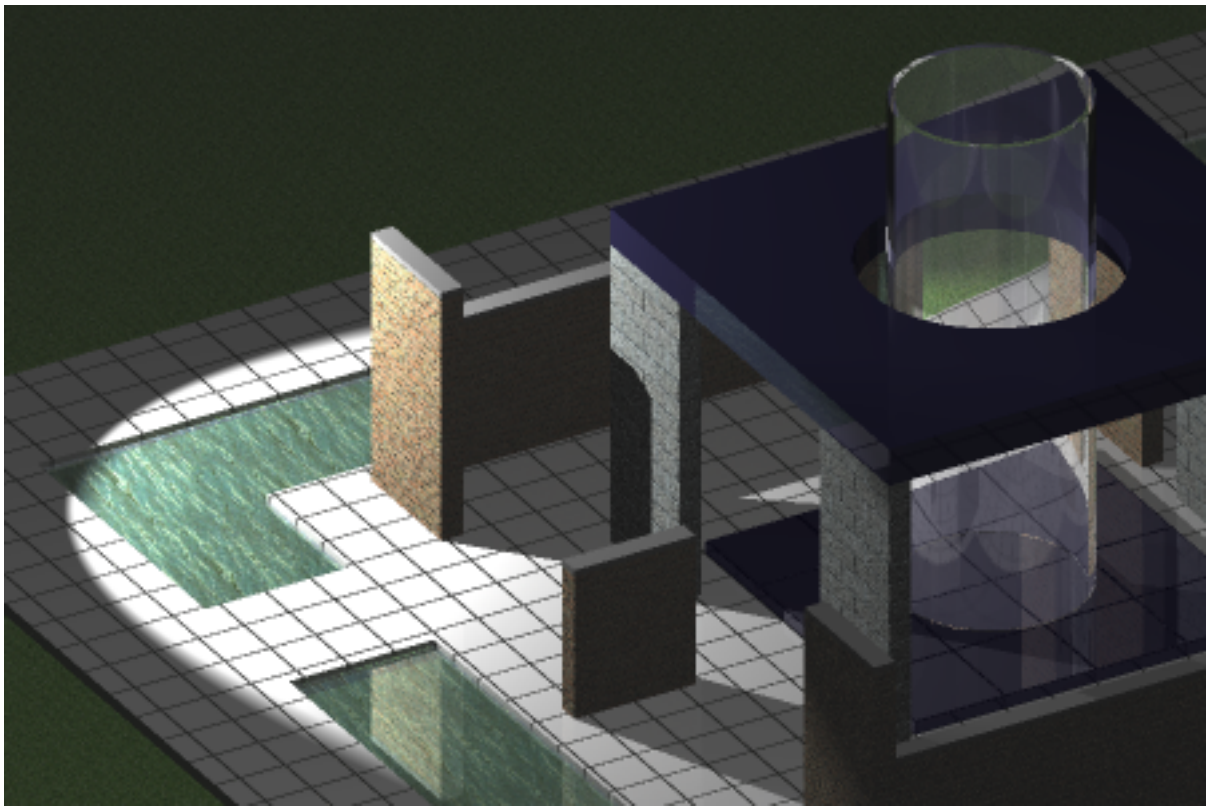


Figure 1.1197

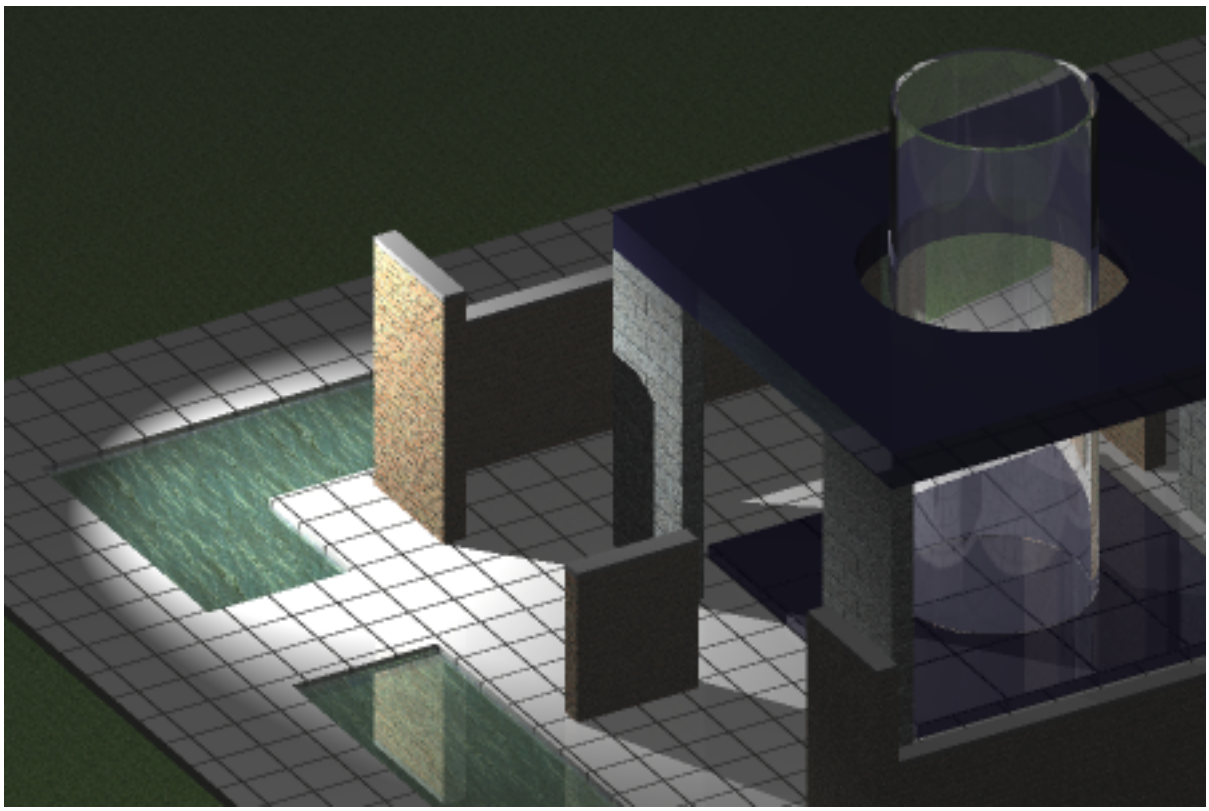


Figure 1.1198

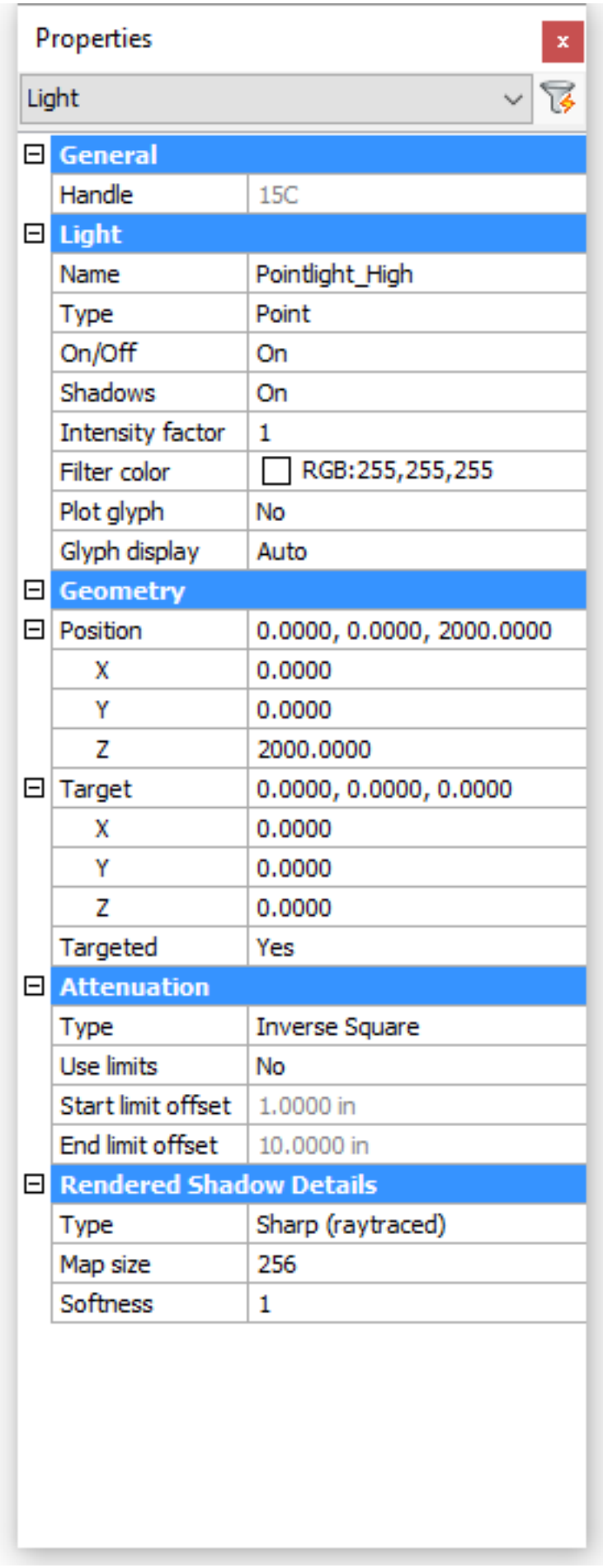


Figure 1.1199

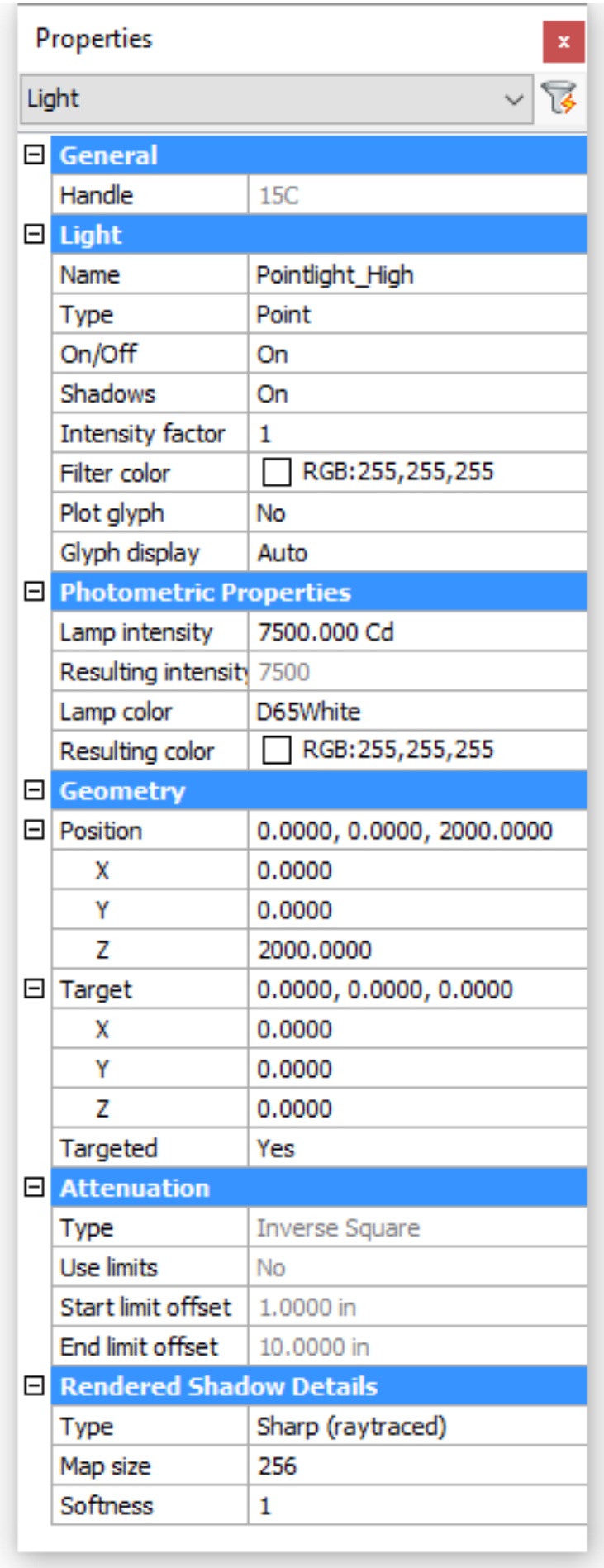


Figure 1.1200

Rendering	
Current material	
Lighting units	[2] Enable photometric lighting using International units (lux)
Default lighting	[0] Default lighting is used only in case no other lights are switched on
Linear contrast	0
Linear brightness	0
Render using hardware	[1] Prefer hardware (faster)
Real world scale	☑ Render real-world scale materials
Sky status	[0] No sky
Texture map path	C:\ProgramData\Bricsys\RenderMaterials\textures;
Render material directory path	C:\ProgramData\Bricsys\RenderMaterials\materials\
Render material static directory path	C:\Program Files\Bricsys\BricsCAD V17 en_US\RenderMaterialStatic\
Tile mode light synch	☑ Synchronize lighting

1.26.5 Setting the Render Preferences

Setting the render preferences

1. Open the**Settings (1.13.11)**dialog.
2. Under**Program Options**expand **theRendering**settings category. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/
Current material: Specifies the material for new entities. By default theGlobalmaterial is used. **Lighting units**:

   - 0: No lighting units are used. Enables generic lighting.
   - 1: Enables American lighting units: Foot-Candles.
   - 2: Enables International lighting units: Lux.

   Web lights can be created only if theLIGHTINGUNITS (1.13.11)system variable is set to 1 or 2. Default lighting: A distant light that follows the view direction. This setting can be different per viewport.

   - 0: Default lighting is used if no lights are switched ON.
   - 1: Default lighting is used instead of the light sources defined in the drawing.

   Linear contrast: Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light. **Linear brightness**: Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport. **Render using hardware**: Choose User software only (slower)if hardware rendering fails. It might be necessary to restart BricsCAD after changing the setting. **Real world scale**: Controls the rendering of materials with units set to real-world scale **Sky status**: Specifies whether sky illuminiation is computed at render time. (Not implemented yet) **Texture map path:** Defines the search path for texture map images. In the BricsCAD program folder exist three subfolders underTextures, each containing a number of texture files of the same name. Images in folder 1 have a size of 256 x 256 pixels, folder 2 contains images of 512 x 512 pixels, images in folder 3 have a size of 1024 x 1024 pixels. If the Diffuse mapsetting of a material uses the image name only (not path), you can control the quality of a rendered image by setting theTextureMapPathuser preference to folder 1, 2 or 3. **Render material directory path**: Specifies the path of the folder where the render materials for theMaterial Browser (6.21.6)are installed. **Render material static directory path**: Specifies the path of the folder which contains read-only resources for predefined materials, such as their previews. **Tile mode light synch**: Controls the synchronization of lighting in all model space viewports. (Internal use only)

1.26.6 The Rendering Tools

Icon	Tool Name	Keyboard	Description
	Render (6.26.22)	render	Photorealistic rendering using materials and lights
	Hide (6.16.10)	hide	Hidden line view (*). If the previous shademode option was2Dwireframe, wireframe representation is restored if you zoom or pan. If the previous shademode option was other than2Dwireframe,the hidden line representation is kept until a different shade mode is chosen.
	Shade (6.27.27)	shade	Shades the entities between the polygon faces. Curved surfaces are segmented.
	2D Wireframe (6.27.29)	shademode + 2	Wireframe representation
	3D Wireframe (6.27.29)	shademode + 3	Wireframe representation
	Hidden (6.27.29)	shademode + H	Hidden line view (*)
	Flat (6.27.29)	shademode + F	Shades the entities between the polygon faces. Curved surfaces are segmented
	Gouraud (6.27.29)	shademode + G	Shades the entities between the polygon faces. Curved surfaces are smoother and more realistic
	Flat + Edges (6.27.29)	shademode + L	Combines flat shading and wireframe lines and curves
===ERROR WITH DOWNLOADED IMAGE===	Gouraud + Edges (6.27.29)	shademode + O	Combines gouraud shading and wireframe lines and curves
	Materials (6.21.11)	materials	Opens the Drawing Explorer - Materials
	Material Map (6.21.10)	materialmap	Allows to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes.
	Light List (6.20.29)	lightlist	Opens the Drawing Explorer - Lights
	Sun Properties (6.27.57)	sunproperties	Opens the sun properties editor
	Geographic Location (6.15.15)	geographiclocation	Opens the Geographic Location dialog.
	Animation Path (6.4.8)	anipath	Records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.

Figure 1.1201

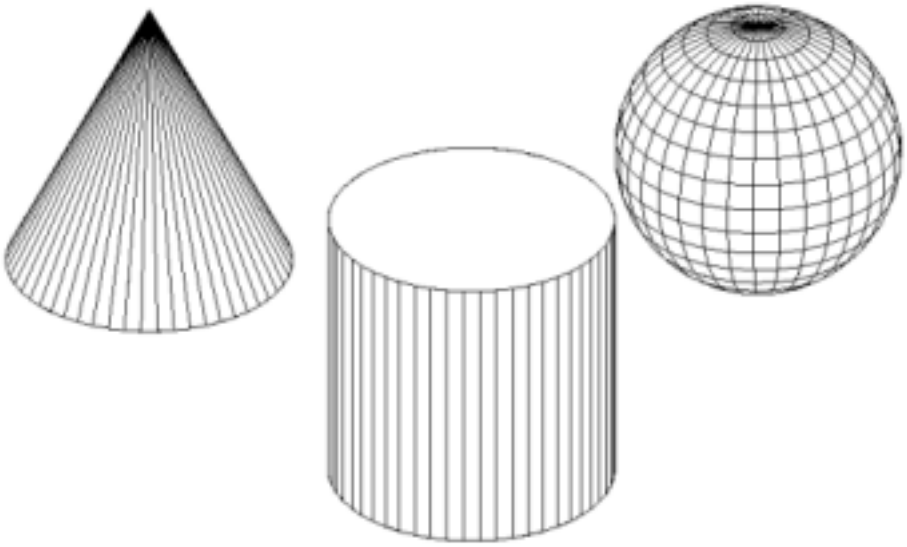
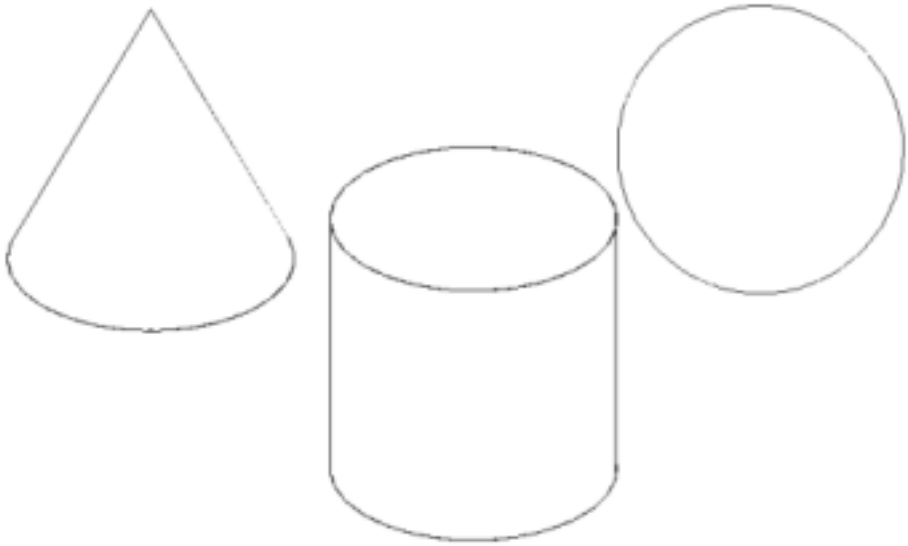


Figure 1.1202



(*) Set the DISPSILH (1.13.11)system variable On to display silhouette lines instead of a polygon mesh for curved surfaces.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enINTL/V18/UI/Gui/source/04/Viewing/Hidden line of a cone, a cylinder and a sphere with DISPSILH Off (left) and On (right).

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enINTL/V18/UI/Gui/source/04/Viewing/Hidden line of a cone, a cylinder and a sphere with DISPSILH Off (left) and On (right).

1.26.7 Creating a Rendered Image of a Model

The rendered image displays in the current viewport, in a separate window or can be exported to a file. The background of a rendered view can be controlled by theBackground (6.6.1)property of the currentnamed view (1.16.10). TheANTIALIASRENDER (1.13.11)system variable controls the amount of anti-aliasing (edge smoothing) applied during the rendering generation. The default value is 1, maximum value is 5.

NOTES

- The values of thelighting system variables (1.13.11)influence the rendering result.
- High anti-alias values incur a high calculation cost.

To render a viewport

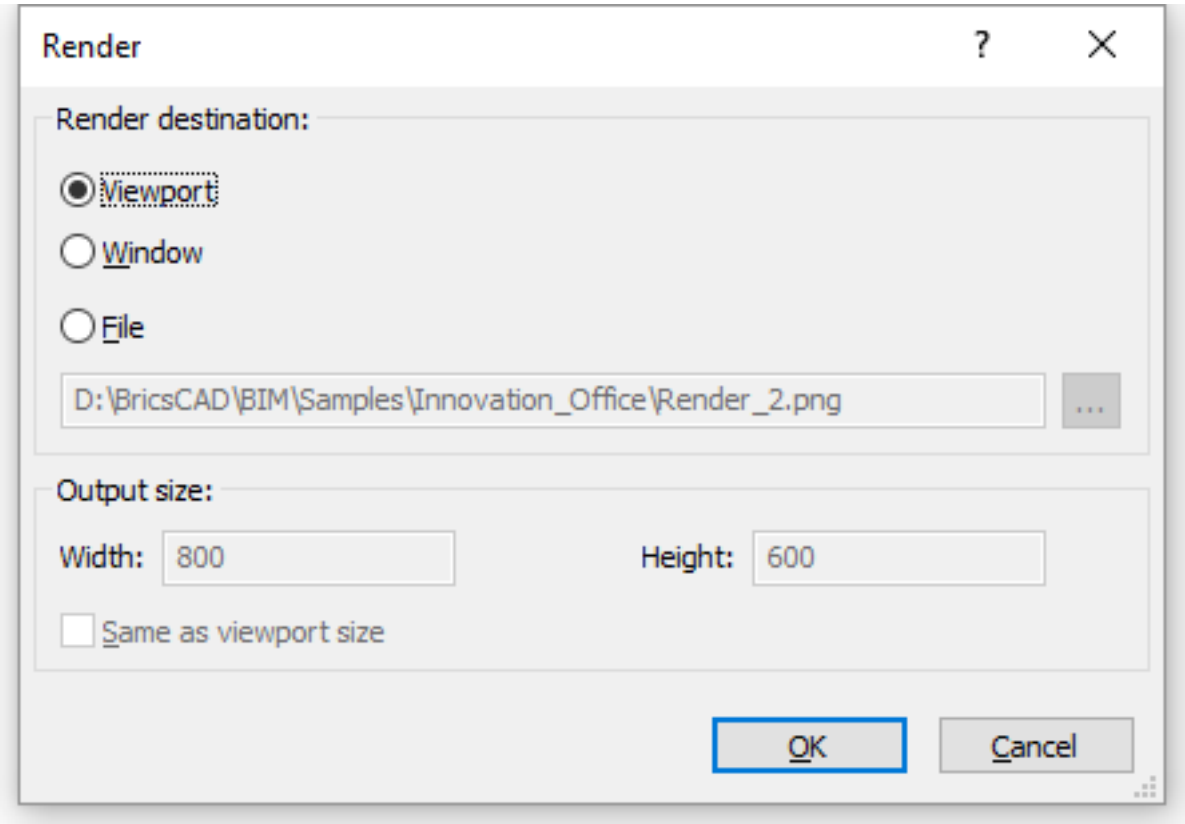
1. Do one of the following

   - Click the **Rendertool button** () on the Renderingtoolbar.
   - Type **Render**in the command bar, then press Enter.

   TheRender dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Rendering/Cr

2. Click the**Viewportoption**.
3. Click the**OK**button. The view is rendered.

Figure 1.1203



To render in a window

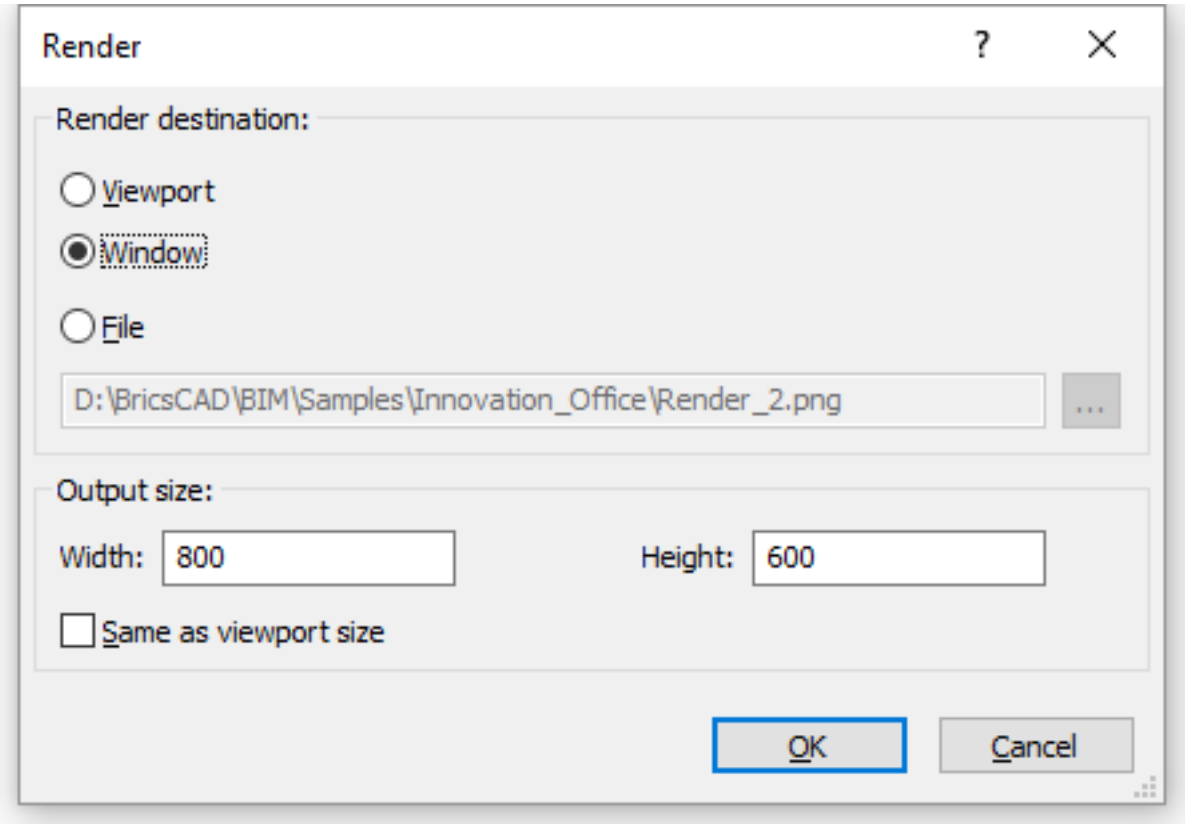
1. Do one of the following

   - Click the**Rendertool button** ( ) on theRenderingtoolbar.
   - Type**Render**in the command bar, then press Enter.

   TheRenderdialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Rendering/Cr

2. Click the**Windowoption**.
3. Click the**OK** button. The view is rendered in a popup window.

Figure 1.1204



To save a rendering to a file

1. Do one of the following

   - Click the**Rendertool button** ( ) on theRendering toolbar.
   - Type**Render**in the command bar, then press Enter.

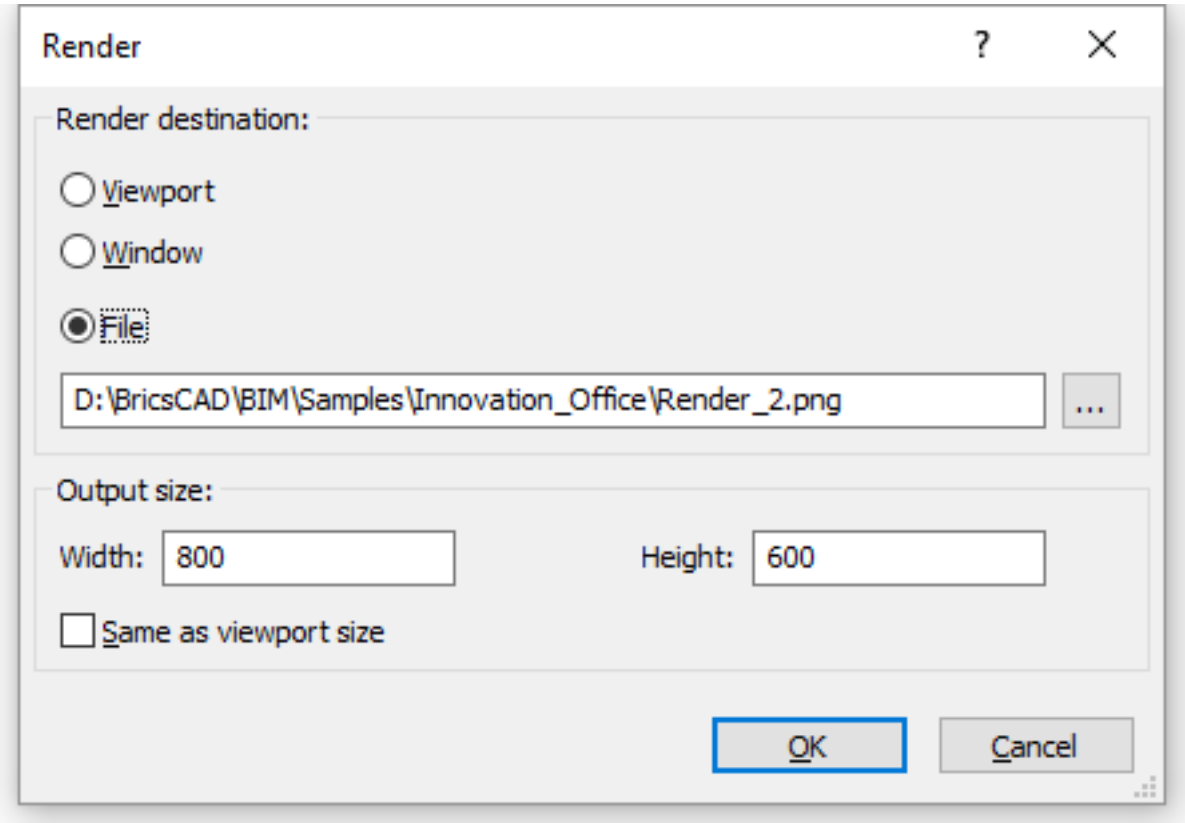
   TheRenderdialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Rendering/Cr

2. Click the**Fileoption**.
3. Click the**Browsebutton** ( ), at the right hand side of theFilefield. TheRender output filedialog displays.
4. Select a folder and type a name for the output file in theFile Namefield.
5. Click theSave as typebutton to select the file type. The options are:bmp,jpgandpng.
6. Do one of the following:

   - Type the desired resolution for the output size in theWidthandHeightfields.
   - Check theSame as viewportsize option to set the outsize to the size of the current viewport.

7. Click the**OK**button to create the file.

Figure 1.1205



1.27 Programming

1.27.1 Developing in BricsCAD

With BricsCAD, you can run many programs originally created for use with AutoCAD without modification on BricsCAD. BricsCAD supports the AutoCAD dialog control language (DCL). You can use all DCL files unmodified within BricsCAD. To write applications in C or C++, BricsCAD offers BRX, a C/C++ language interface, code-compatible with the AutoCAD Runtime eXtension (ARX) interface in AutoCAD. Available topics:

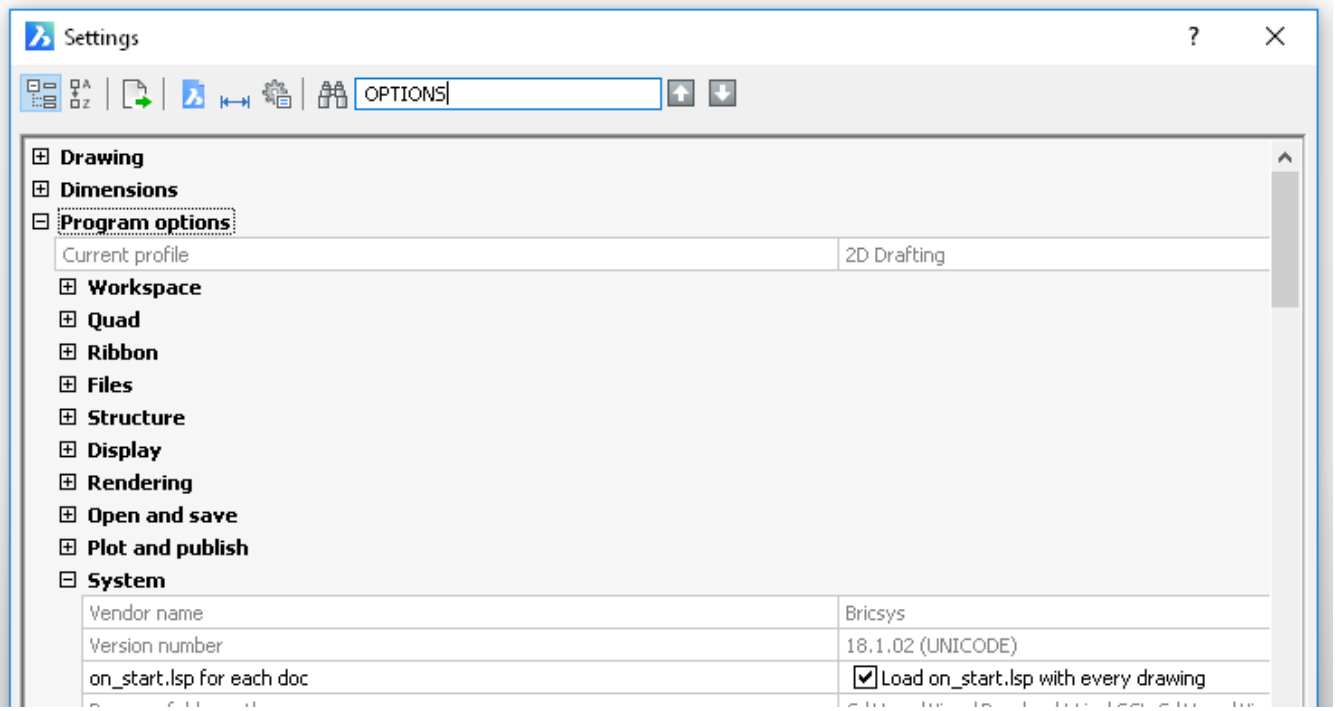
- BricsCAD Solution Build (9.1.1)
- Developer Reference Guide (9.2.1)
  - LISP (LISt Processing)
  - DCL (Dialog Control Language)
  - DIESEL (Direct Interactively Evaluated String Expression Language)
  - COM (Component Object Model)
  - VBA (Visual Basic for Applications)
  - BRX (BricsCAD Runtime eXtension)
  - TX (Teigha eXtension)
  - .NET
- Application Porting Process
  - What are Bricsys LGS components (9.3.2)
  - How to port and add my application? (9.3.1)
  - Supported APIs (9.3.3)
- How to add your application to our catalog (9.4.1)

1.27.2 BricsCAD Lisp – Migrating from AutoCAD

Almost no differences

When you're ready to start migrating, you'll find that there are virtually no differences in BricsCAD LISP versus OtherLISP. Your codeloads and runs, and the functionality is identical. The primary differences when "porting" your apps will be a few minor setup steps, command line structure, and possibly file locations. To start, let review the setup and launch of your existing programs in BricsCAD. If you are auto-launching your programs using **Acad.lsp** or **Acaddoc.lsp**, you will simply need to rename and or combine those files into a single file named "**on_start.lsp**". To enable this feature, simply toggle the option in Settings / Program options / System: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Programming/BricscadLispMigrationSettingsDialog01.png})

Figure 1.1206



Enjoy the BricsCAD Settings Panel

I would also recommend that you prepare to spend a little time familiarizing yourself with the Options/Setting menu in BricsCAD. As you dive in, you will find a ton of settings and options you probably wish you had in your "other" CAD environment. Welcome to BricsCAD! The next step in the settings dialog would be to add your support folders to the Files search path, which is found within Settings / Program Options / Files: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Programming/BricscadLispMigrationSettingsDialog02.png}) As a rule, I never hard code any paths unless absolutely necessary. The support paths I use will typically be added as in the manner above and/or located one level below the current project folder. Typically, I will use a "findfile" function to validate the file exists before proceeding with my commands, as in the method below:

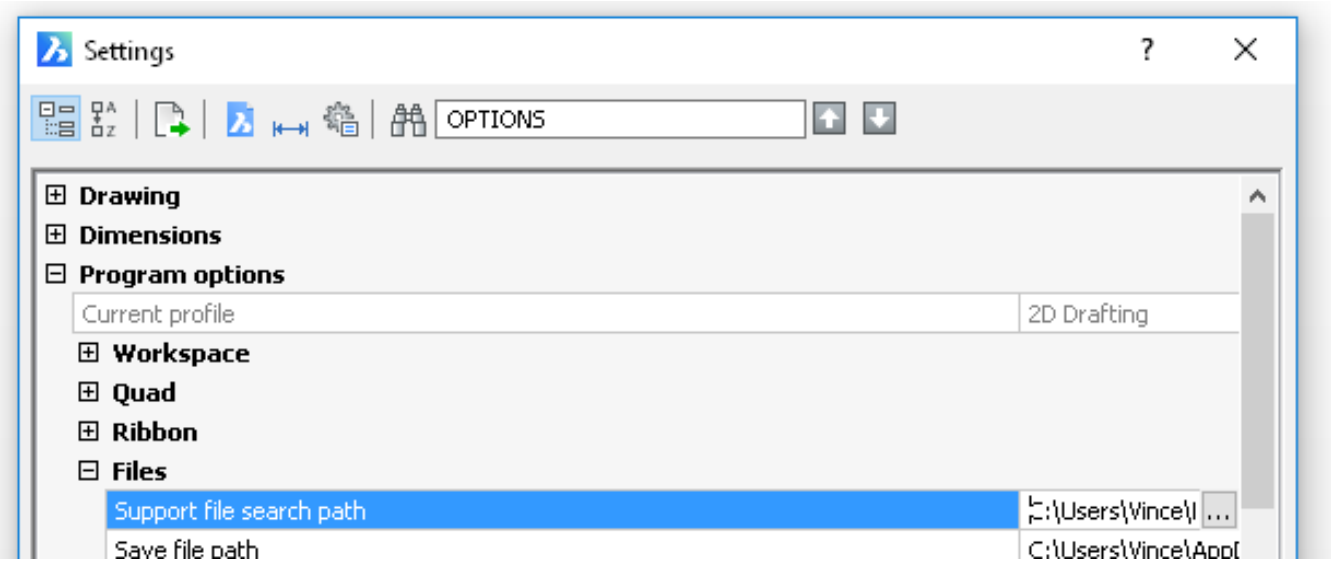
```
(if (findfile "custom-code.lsp") (load "custom-code"))
```

In fact, this is one of the methods I've used to demand-load my functions within the **on_start.lsp** file:

```
(defun c:my-function ()                          ; this defines the command(if (findfile "custom-code.lsp"
```

The command defun (DEfine FUNction) is the same name as the function inside ofyour **on_start.lsp**file and that function redefines the calling function when it's launched. This is another cool reason for using this method, as it demonstrates how dynamic LISP can be!

Figure 1.1207



Take advantage of your move to BricsCAD

When your engineering workflow is running, you usually build along list of maintenance items that you keep putting off. Migrating to a new CAD environment offersan opportunity to make some of those updates andupdateyour documentation standards. Almost every client that I've helped to migrateneed to address old-style hard-coded paths, 8.3 syntax filenames, old layer naming conventions and other no-nos within their standards. As we move forward in this series, I will share some of the methods I use to store user settings and defaultsexternal to yourLISP code, so that they can be updated more easily. Until next time, happy coding!

1.27.3 Drawing Setup Lisp Routines for BricsCAD

This simple drawing setup routine might work for you, with just minimal changes. Here's the code with comments and detailed notes in the comments below:

```
(defun c:mysetup ()                    ;define the command line function "mysetup" (command "audit" "y")
```

How does this drawing setup routine work?

First, you'll see that each individual line in this drawing setup routine could have been typed in at the command line in BricsCAD – and you probably do that today. By wrapping them into a Lisp routine, the **defun** (**define function**) makes the drawing setup routine "callable" from a menu or by typing its name on the BricsCAD command line.

- If we don't include the **defun**, the program will run once, when you load it by its file name. Using the **defun**, and loading this routine within your **lsp** will make the function available at any time by typing its function name: **mysetup**.

Next, look at the **command** strings themselves. The commands with the preceding dashes (e.g. **-layer** and **-purge**) are used to prevent the command's dialog boxes from launching. As a first step when opening files from others, I usually execute an **audit**followed by a **purge** to remove any unused items. In the case of the **purge** command, I often run it multiple times to remove nested items. Here's a bit of added Lisp logic that can make that three-peat happen:

```
(repeat 3 (command "-purge" "all" "*" "n"))
```

Notice that **purge** is done before adding the new, empty layers… as they would be removed if we did it after! The **command** calls for setting layers could be combined into a single command statement, but I think that it's easier to read when each line stands alone. **Filedia** is a global setting (a.k.a. "system variable"), so it is not drawing dependent. I add these types of settings to routines like this to ensure that I get the behavior I expect from the new drawing.

Max what? I included the **maxsort** setting for two reasons. First, to show that some settings require a numeric value instead of a text string (notice the missing quotes in **maxsort**'s command line). **Maxsort** controls the maximum number of dialog box entries that will be sorted. The second reason is that the "factory default" for **maxsort** is 1,000. With today's computers and often found larger project files, that number can be too low. If you open drawings where the layer names won't sort into alphabetical order, the default **maxsort** value could be the reason! Additional **setvar** "calls" include style settings for dimensions and text, unit settings, etc…

A more versatile Layer Maker for your drawing setup routine I'll leave you for now with the following example, which could be added to your version of mysetup.lsp. By creating a simple text file with an editor like Notepad, we can add a list of layer names to be created when the routine is run. This is an alternative to the hard-coded process shown above. By using the **findfile** function, BricsCAD will first search the current directory, then sequentially through your defined support folders to find the layers.txt file. This being known, you can place your default layer setup text file in the support path and place alternative versions of the in individual project folders. This way, you can support multiple layer layouts easily.

```
  (if (not (findfile "Layers.txt"))    ;"not" = looking for no result from findfile (alert "Can't find La
```

1.28 FAQ

1.28.1 License Pooling

Editing the options

The options can be edited in 2 ways:

- in the bricsys.opt file, located in the Network License Manager installation folder.
- on the webinterface of the Network License Manager.
  - Browse to http://localhost:5054 on the license server.
  - Click ”Status” in the menu on the left side.
  - Click ”bricsys” in the OPTIONS column for the bricsys ISV row.

Once edited, Reread/Restart the license server.

Commonly used options

Groups can be used to define a list of usernames, hostnames or IP-addresses. The name can later be used to apply rules for that group. **GROUP** Defines a group of usernames. Syntax: GROUP name list-of-usernames Example: GROUP engineers userA userB userC **HOST_GROUP** Defines a group of hostnames. Syntax: HOST_GROUP name list-of-hostnames Example: HOST_GROUP firstFloor machineA machineB machineC **INTERNET_GROUP** Defines a group of IP-addresses. Syntax: INTERNET_GROUP name list-of-ip-addresses Example: INTERNET_GROUP networkGhent 192.168.1.* 192.168.2.* The wildcard character (*) can be used in IP addresses. **RESERVE** Reserve a number of licenses to a specific (group of) user(s). Syntax: RESERVE num product user|host|group|host_group|internet|internet who [id=nnn] Example: RESERVE 20 bricscad GROUP engineers **MAX** Limit the number of licenses available to a specific (group of) user(s). Syntax: MAX num product user|host|group|host_group|internet|internet_group|project who [id=nnn] Example: MAX 5 bricscad HOST_GROUP firstFloor **INCLUDE** Only allow licenses for a product to a specific (group of) user(s). Anyone not specified by the INCLUDE line is not allowed to use the product. Syntax: INCLUDE product userhost|group|host_group|internet|internet_group|project who [id=nnn] Example: INCLUDE bricscad GROUP bricscadusers **EXCLUDE** Disallow licenses for a product to a specific (group of) user(s). Syntax: EXCLUDE product user|host|group|host_group|internet|internet_group|project who [id=nnn] Example: EXCLUDE bricscad INTERNET_GROUP 1.1.*.* More options can be found in the RLM enduser manual under the paragraph ”The ISV Options File”

Identifying a license file

Since V15, a new feature in the Reprise license manager allows you to add a ’_id’ field in your license file. With this ’_id’ you can identify each license file as a separate license pool. When specifying an [id=nnn] parameter for an added OPTION, that OPTION will only have effect on the corresponding license file. Usage Example: You have 2 licenses, one with 10 seats for BricsCAD Pro, and one with 10 seats for BricsCAD Platinum, and want to specify which users can use which level. **THE LICENSE FILES** Pro License HOST host_name ethernet_mac 5053 ISV bricsys LICENSE bricsys bricscad 15 permanent 10 share=uh contract=1 options=”LA:xx_XX TY:FCO LE:PRO” _ck=6a06665467 sig=”” HOST host_name ethernet_mac 5053 ISV bricsys LICENSE bricsys bricscad 15 permanent 10 share=uh contract=1 options=”LA:xx_XX TY:FCO LE:PRO” _id=1 _ck=6a06665467 sig=”” Platinum License HOST host_name ethernet_mac 5053 ISV bricsys LICENSE bricsys bricscad 15 permanent 10 share=uh contract=1 options=”LA:xx_XX TY:FCO LE:PLA” _ck=6a06665467 sig=”” HOST host_name ethernet_mac 5053 ISV bricsys LICENSE bricsys bricscad 15 permanent 10 share=uh contract=1 options=”LA:xx_XX TY:FCO LE:PLA” _id=2 _ck=6a06665467 sig=”” Now the Pro license is defined by id=1, and the Platinum license by id=2. **THE OPTIONS** If now, for example, you would like to only allow engineers in your firm to use the Platinum version, this can be done by adding the following to the options. GROUP engineers userA userB userC INCLUDE bricscad GROUP engineers id=2 Symptoms I want to adjust the way my licenses are pooled. More information http://www.reprisesoftware.com/RLM_Enduser.h

1.28.2 BricsCAD vs. AutoCAD features comparison

BricsCAD represents the future of design. It is the all-in-one solution for 2D, 3D, Mechanical Design and Building Information Modeling. Based on industry-standard DWG, BricsCAD brings a modern yet familiar approach to design on Windows, macOS and Linux-based PCs. It's available in three editions - Classic, Pro and Platinum - so that you can buy exactly what you need. **BricsCAD (Windows) AutoCAD** *= New feature Platinum Pro Classic Full LT Mac Files Native .dwg 2018 file format dwg for AutoCAD version 9 to 2019 × × × DWGCOMPARE command * * * KEEPME command * * * × × × Password protection Lock files Xrefs in-place editing PDF underlays Export to PDF using style tables Export to DWF 2D & 3D Export to SVG × × × DAE import/export (XML 3D data) × × × eTransmit Sheet set manager Bricsys 24/7® (upload, share, view, annotate documents in the cloud) × × × Communicator (extended import/export) (9) × × × × × Interface Get Started dialog Documents tabs × CUI menus and toolbars Ribbon × Lockable interface (1) * * * Clean screen mode (1) * * * Workspaces × Visual menu customization Extended Quad cursor 2D/3D × × × LookFrom widget (view orientation) Tool palettes Industry standard commands Command Line with auto completion × Extended Drawing Explorer Searchable Settings Manager × × × Selection, Snap & Track Selection preview Quick Select tool ADDSELECTED command SELECTSIMILAR command Cycling through entities or subentities during selection Hot grips for multiple grip points modification SnapTrack™ & Polar tracking ZINTERSECTION 3D snap × × × 3D Full ACIS modeling & editing × (3) (3) Direct 3D modeling × × Polysolids (walls, slabs, ...) × Hardware library (more than 30.000 standard parts) × × × Geometric 3D constraints creation × × × × Geometric 3D constraints solving × × × × Design intent × × × × Visual styles (8) × × Section planes × × Generated views × × × Surface entity creation × × Deformable Modeling × × × 3D Comparison tool × × × × Lofting × × Geometry Healing × × × × Support for Photometric Lighting × × Powerful rendering engine × × × 600 high-definition render materials × × MATERIALMAP command × × Walkthrough navigation × × × ANIPATH command × × × Mechanical Mechanical browser (3) × × × Assemblies import (10) × × × × Assembly modeling (hierarchical component organization) × × × × Assembly viewing × × × × Assembly constraints × × × × Kinematic analysis × × × × Bill of materials × × × × Sheet Metal (9) Sheet metal design × × × × Form features × × × × SMEXTRUDE command * × × × × SMRIBCREATE command * × × × × 3D solids conversion to unfoldable sheet metal parts × × × × Corner relief for adjacent edge flanges × × × × Lofted Bends × × × × Coloring × × × × Diagnostic for invalid sheet metal features × × × × Batch Processing × × × × Building Information Modeling (BIM) (9) BIM features × × × × Structural modeling toolset * × × × × Project database × × × × Library database × × × × Site, Building and Stories × × × × Windows and doors × × × × Fully parametric window creation * × × × × Rooms × × × × 3D Rooms * × × × × IFC import × × × × SketchUp import (version 15 and earlier) × × × × IFC export × × × × Dimensions Dynamic dimensions Association of an existing dimension to an entity Dimensional 2D constraints (7) Dimensional 2D constraints using expressions (7) Dimension Style families Dimension Style comparison Dimension sub units AIDIMFLIPARROW command Text In-place WYSIWYG MTEXT editor Multi-column text Stacked text MLEADER Annotative Styles Spelling Fields Hatch HATCHEDIT command Hatch grip edit Raster images Raster image support (clipping, transparency, ...) Support for ECW and JPEG2000 Support for MrSID Other Support for 3Dconnexion mouse Multithreaded display regeneration Preview filter Layer preview × × × Transparency Layer States Manager Layer filters Multilines and multiline styles (7) (7) Dynamic Arrays Associative arrays Data extraction wizard Block attribute definition manager Block attribute redefinition Dynamic blocks support (4) Dynamic blocks grip editing RESETBLOCK command Dynamic input Dynamic UCS (temporarily switch to surface of 3D solid) × Real time boundary detection × × × Tables & table styles Formulas in table cells Design tables * × × × × XEDGES command × × SOLPROF command × × FLATSHOT command × × Geometric 2D constraints (7) (7) Web map service × × AEC & ADT object enablers × × Civil object enablers * * * × × Printing CTB & STB table files PC3 printer configuration files Import & editing of page setups PUBLISH command Shade plot for paperspace and modelspaceviewports Layout printing with combinations of viewports and visual styles Layout export Publish to PDF Export to STL (3D printing) × Printer calibration (scale only) Programming ActiveX, including in-place editing × × Entity data editor × × × Script recorder × × × Full LISP support (vl, vlr, vla and vlax) × × LISP encryption × × Solutions Development System (SDS/ADS) × × DCL engine × × COM API × × Visual Basic for Applications (VBA) (2) (2) × × × VBA DVB projects support (2) (2) × × × .NET × × × × BRX (BricsCAD Runtime eXtension) × ARX × × TX (Teigha eXtensions) × × × CUI menu files support, with Diesel expressions Legacy menu MNU and MNS file support API for toolbar and menu customization Licensing Multi-platform licenses × × × Single All-In Network × × Volume × × × Academic (1) On Mac and Linux, available from version 18.2. (2) Version 7.1 - For Windows 64 bits and 32 bits. (3) Partial. (4) Defining dynamic blocks is not possible yet. (5) View only. Edit under development. (6) Without MFC classes. (7) View/edit only. Not create. (8) Limited (no rendering). (9) Optional module. (10) Requires BricsCAD Communicator. In BricsCAD Pro, only for geometry (no structure). (11) Partial. Supports fewer formats on Mac and Linux, see Communicatorsupported file formats.

1.28.3 Failure to add license key on Windows Vista or higher

Log in with administrator rights and try again to enter the license key. Note that on Windows Vista, Windows 7 and Windows 8 it is not sufficient to log in as administrator or main user. You also have to start BricsCAD by right-clicking the shortcut and choosing "Run as administrator". This will launch BricsCAD with full elevated rights and privileges. During such an elevated session, you will be able to enter the license key.

Symptoms

Though the user is logged in as administrator, the license manager still doesn't accept a valid license key, with one of the messages below:

- "Failed to add license key (error 25)" (or error 27).
- "Error creating registry key (need administrator rights for this operation)."
- "Error opening registry key."
- "Error setting registry key value (need administrator rights for this operation)."

Cause

On Windows Vista or higher, access rights are more restricted. Even if you are logged in as administrator, it appears that the application has not enough rights to write in the registry under HKEY_CLASSES_ROOT.

1.28.4 How to download an old BricsCAD release?

The three latest previous major releases are available for each platform and language (if existing). Go to the Download page, then click the ”More download options” link and the ”Show old releases” link. Select the platform, language and version you want to download.

1.28.5 How to migrate my license to another computer?

I have a Single or a Volume license and I want to migrate my BricsCAD license to another computer.

- First deactivate your license on the old computer.
- Once the license is deactivated, you can activate it again on another computer.

All your valid licenses can be found on the Bricsys website in My Account > Licenses and support contracts (login required).

1.28.6 How to port and add my application?

With the BricsCAD development platform you can port your AutoCAD ® based applications using unmodified source code. And the BricsCAD team will always be there for you, not only to support the porting but also to assist with the marketing of the solution to the worldwide community of BricsCAD users.

- Reach new cost conscious customers worldwide.
- Straightforward port with unmodified source code.
- Active sales & marketing campaigns, free of charge.
- Application Catalog and Online Store.
- Customer Support Infrastructure.

On the Bricsys website, register as developer and contact us to get started!

1.28.7 How to run BricsCAD in a lower features level (Pro or Classic)?

The RunAsLevel preference setting allows to run BricsCAD in a lower license level, e.g. a Platinum version can be run as a Pro or a Classic, a Pro can be run as a Classic. This facilitates the evaluation of the features available for each license level. In BricsCAD, in the Settings dialog box, go to the RunAsLevel preference or type "RunAsLevel" on the command line. Enter the value for the level you want to try: 0 = Classic, 1 = Pro, 2 = Platinum (default). Restart BricsCAD to apply the changes.

Symptoms

I cannot try the Pro or Classic features.

Cause

BricsCAD is installed as a full Platinum version (or in trial mode). **More information** If the licensed level is lower than RunAsLevel, RunAsLevel will be ignored. Look at the tutorial movie on Bricsys TV. To try BricsCAD in a higher features level (Pro or Platinum), see How to try a higher features level in the BricsCAD knowledge base.

1.28.8 Norton Antivirus gives false positive alerts about BricsCAD

This is a recurrent problem with the Norton heuristics module. When the problem turns up we report it to Symantec and they provide an updated database with a fix. Because there is no definitive solution we recommend using another AV. But it is always possible to fix things locally and keep using Norton with BricsCAD. For this one has to add an exception to Norton locally for BricsCAD, so that Norton ignores it.

Symptoms

Norton Antivirus considers BricsCAD installsets a threat of WS.Reputation.1 and will delete the installer. If Norton is disabled during download of the package and during installation, then Norton will again complain about the installed program. Problem files are dcl.dll , acmodeldocobj_3.06_10.tx , pstyle_lib.dll and licensemanager.dll .

Cause

In the cases we are aware of this is an erroneous detection (false positive) from Norton Antivirus.

More information

There is a simple check to see if a virus has tampered with the installer: All BricsCAD installsets (MSI, EXE files) are signed with the ”Bricsys NV” certificate. If someone alters the file (viruses, ...) after publication, windows will no longer consider the certificate to be valid. So this acts as a check.Symantec’s WS.Reputation.1 System is contested, so you can bypass Symantec’s WS.Reputation.1 System:

- http://www.symantec.com/business/support/index?page=content&id=TECH105602&actp=search&viewlocale=en_US&searchid=1309521900035
- http://www.symantec.com/security_response/writeup.jsp?docid=2010-051308-1854-99
- http://www.symantec.com/security_response/writeup.jsp?docid=2010-051308-1854-99&tabid=2
- http://www.symantec.com/security_response/writeup.jsp?docid=2010-051308-1854-99&tabid=3

1.28.9 Print has become lighter and thinner

The change may require small modifications in the CTB's on the client side. Our apologies for the inconvenience. From V10.3.12 on we deliver two new CTB's that can serve as a reference for conversion. The two new CTBs allow to print the colors as they show onscreen on a white background, and on a black background, so they are close to the 'default.ctb' file but slightly darker.

Symptoms

After upgrading to V10.3 printed text has become less fat and printed colors have become lighter.

Cause

We've recalibrated our colormapping for print. Historically it was set too dark. Print results are now more compatible with 'other programs'. Actual print with default.ctb (all colors 'by entity')is now a bit lighter than the view on display. It used to be a bit darker than displayed. Every CTB, including default.ctb, now yields lighter prints than before. Two new print color table files, displaycolors.ctb and displaycolors-darkbg.ctb, were added to allow printing of colors as they display onscreen, both for a white background and for a dark background. In addition, and independent from the above changes, printing of TTF text has been changed and the print result will now look slightly thinner, though better readable. The excessive boldness tended to turn small characters into blobs.

More information

Other causes of thinner/fatter results

- Verify LWDEFAULT. This is stored at application level.

1.28.10 Supported file formats in BricsCAD

VECTOR FILES

1. DWG format import/export
   - open DWG/DXF/DWT
   - saveas DWG/DXF/DWT
   - export DWF
2. Non-DWG formats:
   - IFC: IFC files can be imported/exported via Import/Export command
   - SKP: Sketchup files can be imported
   - DGN: there is no import/export to or from DGN
   - SAT: ACIS sat files can be imported/exported via acis-in acis-out
   - IGS, STP, SLDPRT are not supported in plain BricsCAD, but are imported/exported with use of an optional module: BricsCAD Communicator, along with several other 3D MCAD modeling formats. Please read below for complete informations.

RASTER files (also known as BITMAP or IMAGE files):

Import/Attach Strictly speaking, you cannot import a RASTER image into a DWG. The DWG format is, in essence,a Vector and not a Raster format. BricsCAD does, however, enable you to attach a raster file as a background image. This is achieved via the Image Enabler. Supported formats include: .BMP, .TIFF, .JPEG, .GIF, .PNG, .WMF, .EMF, .PCX, .RAS, .TGA V9.1.1 and higher: Support for ECW images. V10.1.4 and higher: support of MrSID images. LizardTech's MrSID is one of the leading geospatial imagery formats. Raster image files can have 'world files' or 'position files' associated to them. A world file is a text file which describes the size (scale), the orientation and the coordinates of the raster map images. Position file are typically used in GIS. Supported position file formats are: .TFW , .RAT, .JPW, .PNGW

EXPORT You can save screen view to raster format via file>export. The SLIDE command also produces raster files (.SLD). If you use a virtual printer like PDFCreator or PDF995, CutePDF, you can produce (large scale) PDF output. Finally, many 'external' programs allow you to save a screen copy.

Symptoms

What formats are supported by BricsCAD? What are the import and export formats?

Cause

Natively BricsCAD only supports DWG/DXF. Settings allows to save a DWG to a specific DWG (DXF) revision. See 'Save as' in the help project. Alternatively the Export command allows to export drawings to a variety of vector and raster formats. See 'Export' in the help project. Apart from DWG format, BricsCAD also able to handle Acis (.SAT) files. BricsCAD can import (/export) .SAT (=Acis) files via the ACISIN, ACISOUT commands respectively. There is no direct support to import STEP/IGES or SolidWorks formats in BricsCAD. However there are many additional applications which will perform data exchange/transformation.

More information

In addition to BricsCAD, the BricsCAD Communicator module can import/export the following file formats: **IMPORT** IGS, IGES: Initial Graphics Exchange Specification STE, STP, STEP: Standard for Exchange of Product data IAM, IPT: Inventor CATIA V4 and V5 PARASOLID PAR, PSM: Solid Edge PRT: NX ASM, PRT: Creo Elements / Pro Engineer SLDASM, SLDPRT: Solidworks VDA: VDA-FS **EXPORT** STE, STP, STEP: Standard for Exchange of Product data IGS, IGES: Initial Graphics Exchange Specification CATIA V4 and V5 3D PDF

1.28.11 Trailing afterimages or ghosts when dragging toolbars and dockable panels.

The problem does not block or delay the user. If the user considers the problem unacceptable the only known way to handle this is to avoid/delay installing Windows 10 Creator Fall update or to uninstall it.

Symptoms

When dragging toolbars or dockable panels there are gray afterimages left behind.As soon as you press the mouse button on a toolbar a gray boundary is drawn around the toolbar. That is normal. The problem now is that this gray rectangle is not or partially undrawn/refreshed as you drag the toolbar. It does not block the user from working and it is a transient problem. but it is inconvenient.

Cause

This started with Windows 10 Creator update 1709, fall 2017. The problem is not limited to BricsCAD. We've found the same problem in AutoCAD toolbars and in Acrobat Reader (element properties). The common attribute appears to be that these are all dialogs that are floating / dockable. At this time it has not been established what the cause is or how it will be fixed.

More information

It may be that a later update of Windows or a display driver may fix the issue. It is possible that a setting in windows can fix it but this has not yet been found. It is possible that we eventually have to adapt the program to work around the problem. It is also possible that this is a preexisting problem which now got much worse. Smaller redrawing problems with the boundary rectangles have been encountered in earlier versions of Windows.

1.28.12 What are the BricsCAD minimum system and hardware requirements?

Recommended hardware requirements are handled in knowledgebase item 390. This item is about minimal hardware.

Operating systems

- Windows 10 or higher (32 and 64 bits)
- Windows 8 or higher (32 and 64 bits)
- Windows 7 (32 and 64 bits)
- Windows Vista with Service Pack 2 or later (all editions except Starter Edition)
- Windows Server 2008 R2
- Windows Server 2008 with Service Pack 2 or later
- Mac OSX 10.9 or higher
- Ubuntu 14.04 or higher (*)
- openSUSE (*)
- Fedora (*)

Processor

- 1 GHz processor or faster, or compatible processor.

Memory

- Next to the memory required by the operating system, BricsCAD requires 256 MB (1 Gb or more recommended).

Hard Drive

- 250 MB for program files + 1 GB free space.

Display

- 1024 x 768 XGA with true color (minimum)
- Supported GPU (**): see below for more information on supported chipsets for GPU acceleration for non-wireframe render modes such as realistic rendering. Please note that non-wireframe render modes and all other features of BricsCAD can still be used without hardware acceleration.

More information

(*) Supported Linux distribution versions: Bricsys has chosen to support Ubuntu, openSUSE and Fedora. BricsCAD (Linux) V16,V17 is built with gcc 4.8.2 on Ubuntu 14.04 LTS. As a result the minimum required c++ runtime (libstdc++) is 4.8.2. In practice, BricsCAD (Linux) can run on almost any distribution that is newer than Ubuntu 14.04. We receive many support requests for other distributions. As a rule of thumb, if we can reproduce the reported issue in one of the supported distributions, we provide support for it. (**) BricsCAD's non-wireframe render modes are based on RedSDK technology from Redway3D®. GPU-acceleration for these render modes requires a supported GPU (graphic processing unit), either on a video card or on the motherboard.

1. On Windows, all graphic cards from NVIDIA, ATI and INTEL are supported by RedSDK
2. On Linux, 3D graphics hardware acceleration in RedSDK is supported on the following video chipsets:
   - NVIDIA cards. It is advised to use the latest recommended driver from NVIDIA.
   - ATI cards. It is advised to use the latest recommended driver from ATI/AMD.
3. 3D graphics hardware acceleration in RedSDK on Linux is NOT supported on:
   - INTEL cards. As opposed to ATI and NVIDIA, 3D hardware acceleration in linux drivers for INTEL cards is relatively new. So far we do not yet officially support 3D hardware acceleration on INTEL cards, although we are working on it.
   - Laptops with dual graphics, although it happens to work on some dual graphics systems using recent versions of proprietary NVIDIA drivers.

1.28.13 What are the recommended system and hardware requirements for BricsCAD

This item is about recommended hardware. See also the minimal hardware requirements. The way dwg files are structured and the way dwg entities depend on each other and affect each other, makes it really difficult to process them using multiple parallel threads. Although BricsCAD meanwhile is capable to e.g. load drawings and regenerate display objects using multiple threads, today the majority of the processing that occurs during a BricsCAD session is still executed in a single-thread. What does this mean? Consider a typical processor with 4 physical cores, which usually will have the double number of logical cores: 8. During most of a BricsCAD (or AutoCAD) session, only 1 of these logical cores will be used, resulting in a maximum processor load of 12.5 % Although we continue our efforts to parallelize more processes, that is how it is today. Hence, the part of processor performance that matters most is the Single Thread performance. There are specialized hardware benchmark sites where such processor information can be found, a good one is the Passmark site - http://www.cpubenchmark.net/overclocked_cpus.html There you can see that e.g. the high-end Xeon E5-2699 v3 @ 2.30GHz with its impressive 18 physical cores and equally impressive 4200 $ price tag, has a Single Thread performance Rating (STR) of 1701. This is significantly lower than e.g. the i7-4790K @ 4.00GHz with its STR of 2530 and its 340 $ price tag. In other words, 50 % better performance and less than 1/10 the price... We recommend to look for a fast i7 processor with 4 physical cores and high STR score. Concerning hard drives: we recommend a fast SSD primary drive combined with a fast SATA as secondary drive. 16 Gb of RAM will be sufficient for most tasks, but given the relatively low price, it may be interesting to install more and be prepared for handling large projects. The GPU's on most graphic cards found in today's computers will handle editing tasks of 3D models using a 'rendered' visual style with satisfactory performance. It is important to know that to update views of 3D models during editing, the bottleneck usually is at the CPU side, not at the GPU side. The time when a high-end GPU will make a significant difference is when using the RENDER command to generate images.

Symptoms

Minimal hardware requirements are handled in knowledgebase item 65. This item is about recommended hardware.

More information

Please be aware that a hardware change is often the last option to consider when addressing performance issues. Changes in settings or changes in drawing structure usually have a far larger impact.

1.28.14 What formats are supported by BricsCAD?

Natively BricsCAD only supports DWG/DXF. Settings allows to save a DWG to a specific DWG (DXF) revision. See 'Save as' in the help project. Alternatively the Export command allows to export drawings to a variety of vector and raster formats. See 'Export' in the help project. Apart from DWG format, BricsCAD is also able to handle Acis (.SAT) files. BricsCAD can import (/export) .SAT (=Acis) files via the ACISIN, ACISOUT commands respectively. There is no direct support to import STEP/IGES or SolidWorks formats in BricsCAD. However there are many additional applications which will perform data exchange/transformation.

VECTOR FILES

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   - open DWG/DXF/DWT
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2. Non-DWG formats:
   - IFC: IFC files can be imported/exported via Import/Export command
   - SKP: Sketchup files can be imported
   - DGN: there is no import/export to or from DGN
   - SAT: ACIS sat files can be imported/exported via acis-in acis-out
   - IGS, STP, SLDPRT are not supported in plain BricsCAD, but are imported/exported with use of an optional module: BricsCAD Communicator, along with several other 3D MCAD modeling formats. Please read below for complete informations.

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EXPORT You can save screen view to raster format via file>export. The SLIDE command also produces raster files (.SLD). If you use a virtual printer like PDFCreator or PDF995, CutePDF, you can produce (large scale) PDF output. Finally, many 'external' programs allow you to save a screen copy.

Symptoms

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Cause

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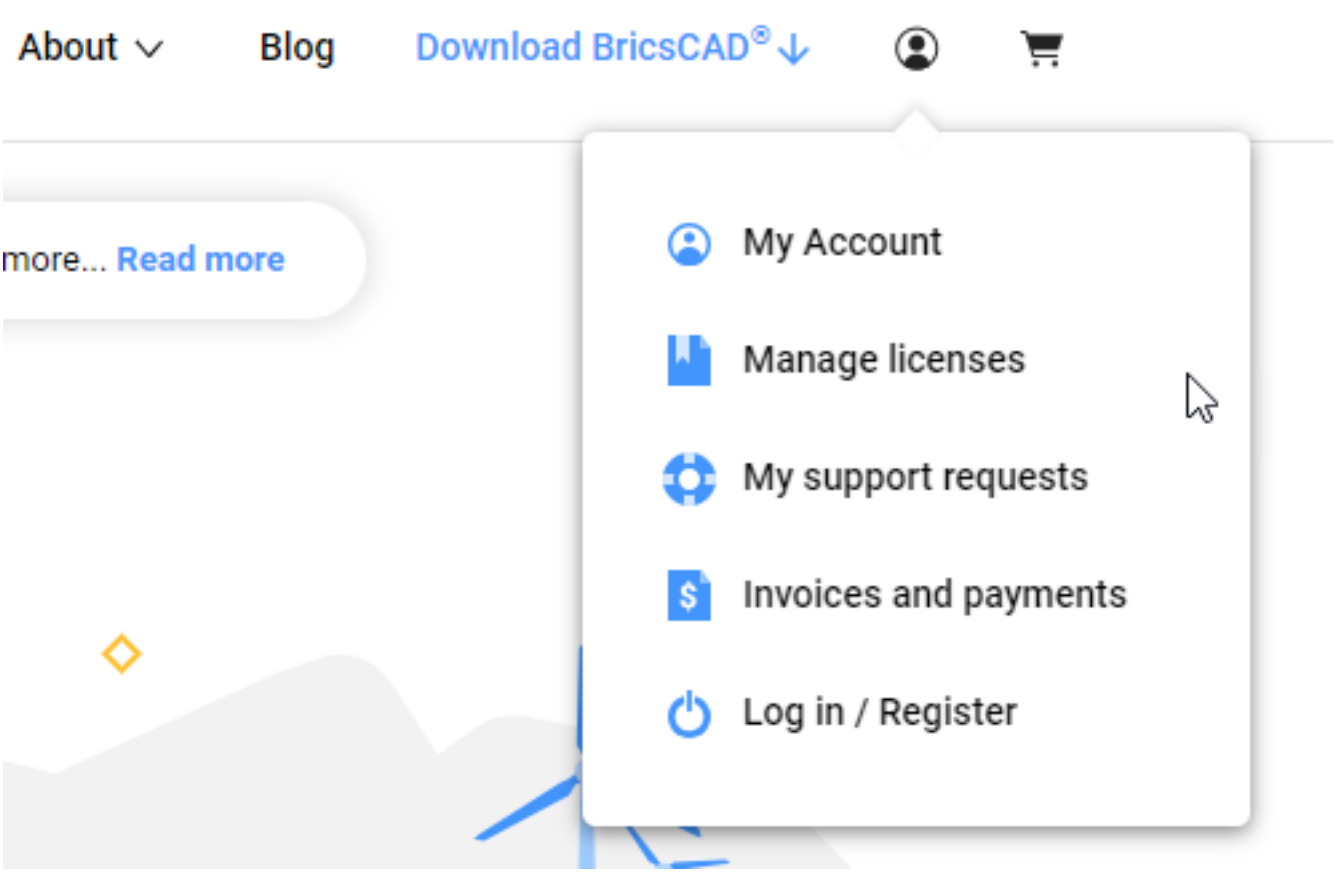
1.28.15 Where can I change my mailing subscriptions?

Your mailing subscriptions can be managed under ”My Account” on the Bricsys web site. Your account can be accessed by clicking on your name (be sure you’re logged in) in the upper right corner of the Bricsys web site. In the menu on the left, click on ”Mailing subscriptions”. There you can check/uncheck the newsletters or release mails you like to receive.

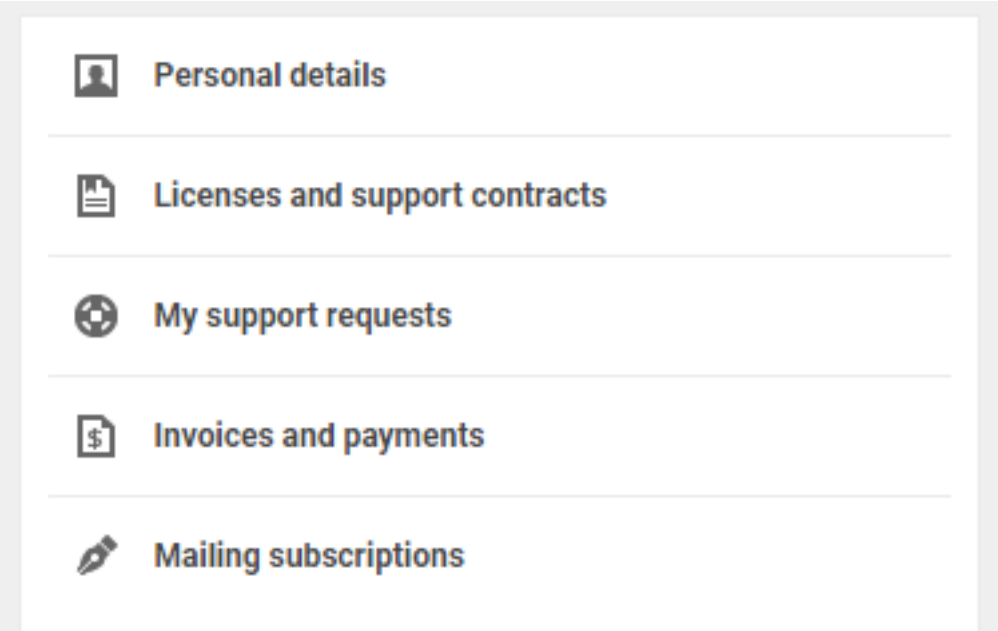
Figure 1.1208



1.28.16 Where can I find my licenses?

On the Bricsys website, log on to your account click on your name in the upper right corner. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/FAQ/LicensesBOAlogin.png}) Then click on Licenses and support contracts to get an overview of your licenses.Only valid licenses are displayed. Upgraded or expired licenses are not displayed. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/FAQ/LicensesBOAlo

Figure 1.1209



1.28.17 Windows 10 Upgrade

Not yet upgraded

We recommend deactivating the license before upgrading to Windows 10. You can do this by starting BricsCAD, and clicking [Help > BricsCAD License...] in the menu bar. Here you will find the deactivate button. Once upgraded, you can activate with your activation key as before. **Note:** this only applies to online activations. Manual activations will not be affected.

Already upgraded

V14 or older If you have upgraded with V14 still activated, you might get an error "The license for this product is locked to another computer or is using an invalid hostid" (error -4). In this knowledgebase article is described how to resolve it.

V15 or newer Upgrading with V15 or higher should not cause any problems. If an activation dialog appears when starting BricsCAD, enter your activation key in the dialog and BricsCAD should start.

Chapter 2

BricsCAD BIM

2.1 Getting Started

2.1.1 Exploring the BricsCAD BIM User Interface

https://www.youtube-nocookie.com/embed/obilBLvybZ0 This tutorial explains the different User Interface elements: Menu bar, toolbar, panels, status bar and commandline. Back to Tutorial Overview

2.1.2 How to use Quad Cursor & Rollover Tips

https://www.youtube-nocookie.com/embed/-UZvYKVOX0Q

This tutorial explains what the Rollover Tips are and how to use them. It also explains how to use the Quad Cursor in different scenarios. Back to Tutorial Overview

2.1.3 How to navigate

https://www.youtube-nocookie.com/embed/WJLSzSCKO-E

This tutorial explains how to navigate in a 3D model space, using the keyboard and mouse shortcuts as well as the LookFrom widget. Back to Tutorial Overview

2.2 Getting Started

2.2.1 Starting a Project

How to set up a new drawing in BricsCAD BIM.

Implementing the proper set up of your BricsCAD settings is a crucial step to produce a drawing that meets your project needs.

BricsCAD Welcome Screen

Folder Setup Start a project by choosing one of the available options on the welcome screen. The welcome dialog box displays when you launch BricsCAD. This screen allows you to define drawing properties such as profile and units before beginning your new drawing.

Get Started The following illustration shows the BricsCAD Get Started tab on the Welcome dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/WelcomeDialogBoxGetStartedTab.png
The Get Started tab includes settings required to start a new drawing in BricsCAD. To start a new drawing, choose one of the available options. The following table presents each option of theGet Started tab.

New drawing (1)	Creates a new drawing using the default template of the current profile.
Open other drawing(2)	Opens an existing drawing.
Start from template (3)	Creates a new drawing using a template of your choice.
Open a recent file (4)	Opens the most recent drawings. Up to 20 files can be listed here.
Set current profile (5)	Choose a user profile in the list, the most recently used profiles will be shown here and the most recently used profile is indicated by a blue frame.
Select other profile (6)	Select a profile in the All profile dialog. The selected profile is added to the Set current profile list.

The following illustration shows the BricsCAD Set current profile list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom
By default the five most recently used profiles display. If necessary, drag the right-hand edge of the Welcome dialog box to see more profiles.

Figure 2.1

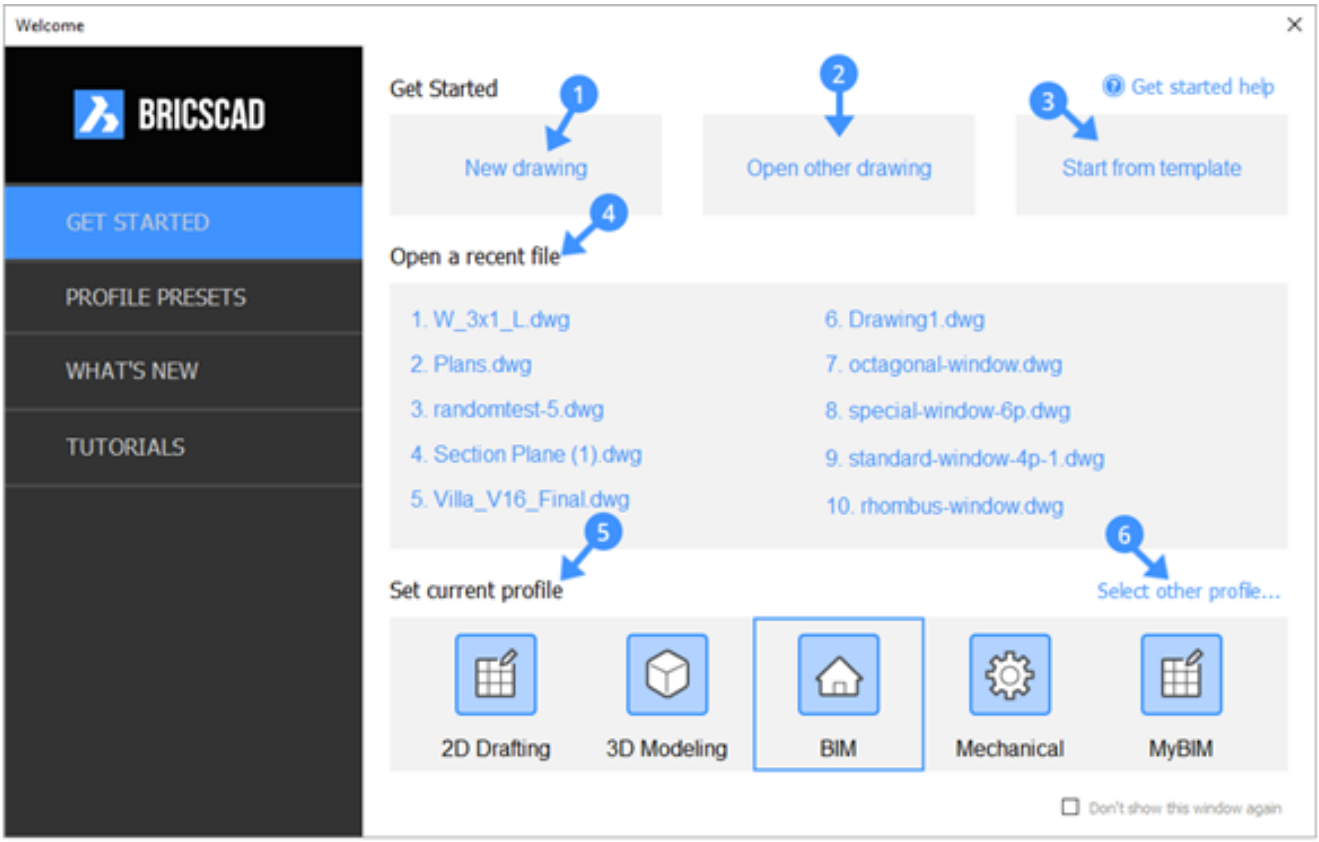
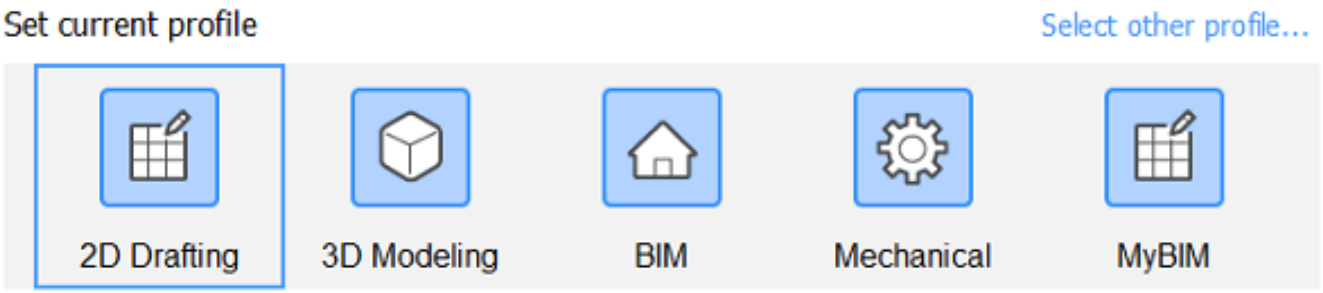


Figure 2.2



Profile Presets The Profile Presets includes four pre-defined profiles andadrawing units bar which letsyou get started on your drawings.Each profile contains workflow user-specific settings e.g. BIM Profilecreates a fresh BIM user profile using-built in, hard-coded default settings.

- Each of the profile presets is associated with a **Workspace (7)**.
- To specify the drawing **units (8)**, choose either inches, mm, cm or m.

The Get Started tab includes settings required to start a new drawing in BricsCAD. To start a new drawing, choose one of the available options. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/Wel
To create a new profile, select one of the available profile and rename that **Profile**. (\vref {fig:C:/Dev/Task-2448/images/staticbr

Figure 2.3

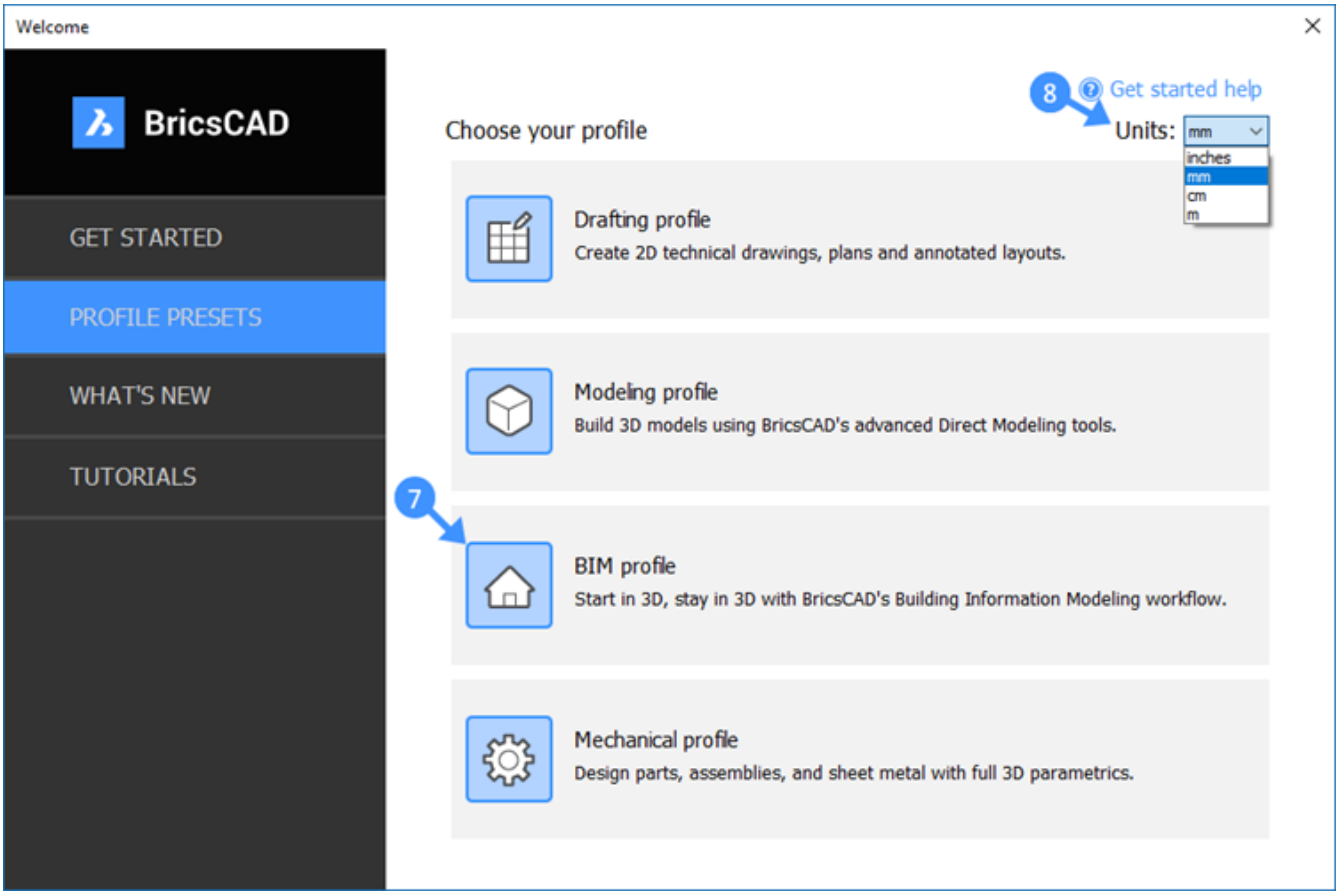
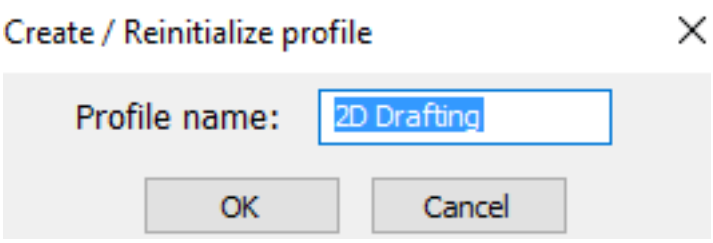


Figure 2.4

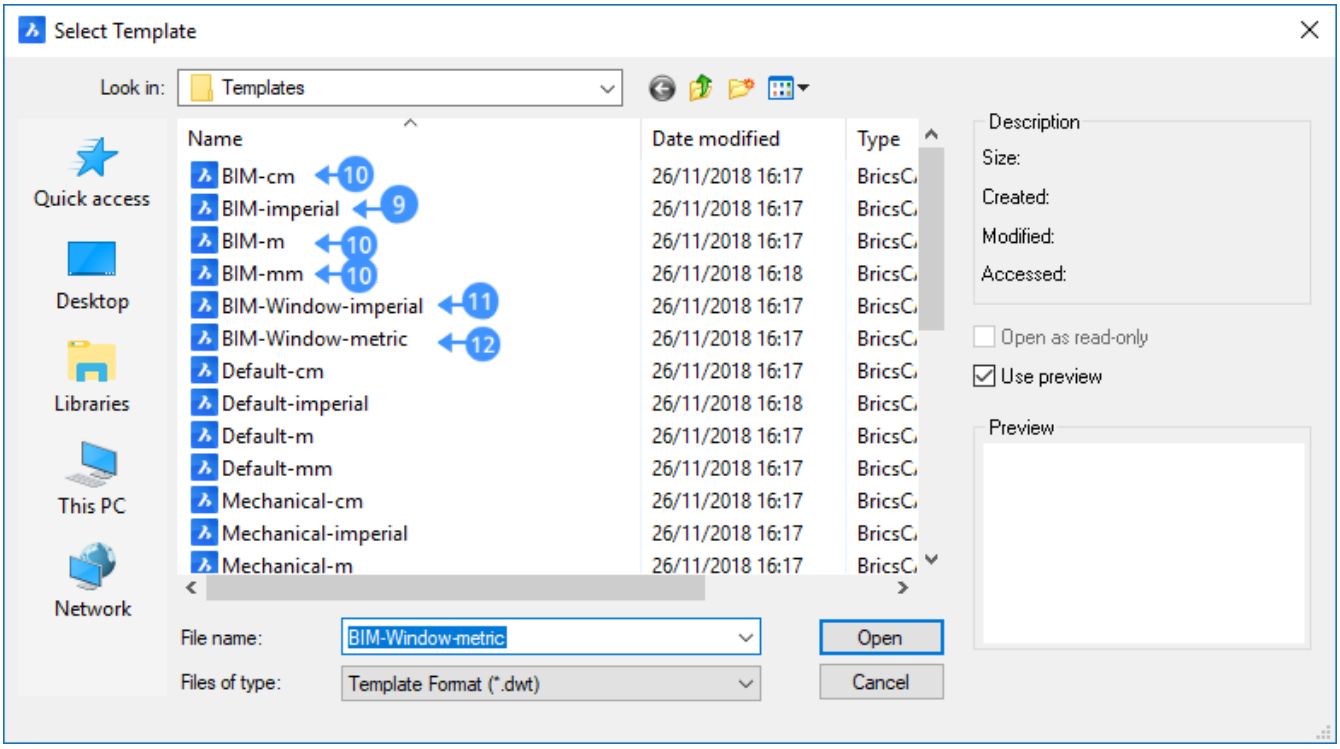


Using Templates

To use a template, click **Start From Template** on the welcome screen. The template dialog box appears. You can choose one of the predefined templates to create your drawing layout. The following illustration shows the Select Template dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/SelectTemplate

BIM Imperial (9)	Allows you to create models using imperial units with the settings of BIM profile.
BIM Metric (10)	Allows you to create models using metric units with the setting of BIM profile.
BIM Window Imperial (11)	Allows you to create the geometry of the window or door using the imperial units with the set of predefined layers.
BIM Window Metric (12)	Allows you to create the geometry of the window or door using the metric units with the set of predefined layers.

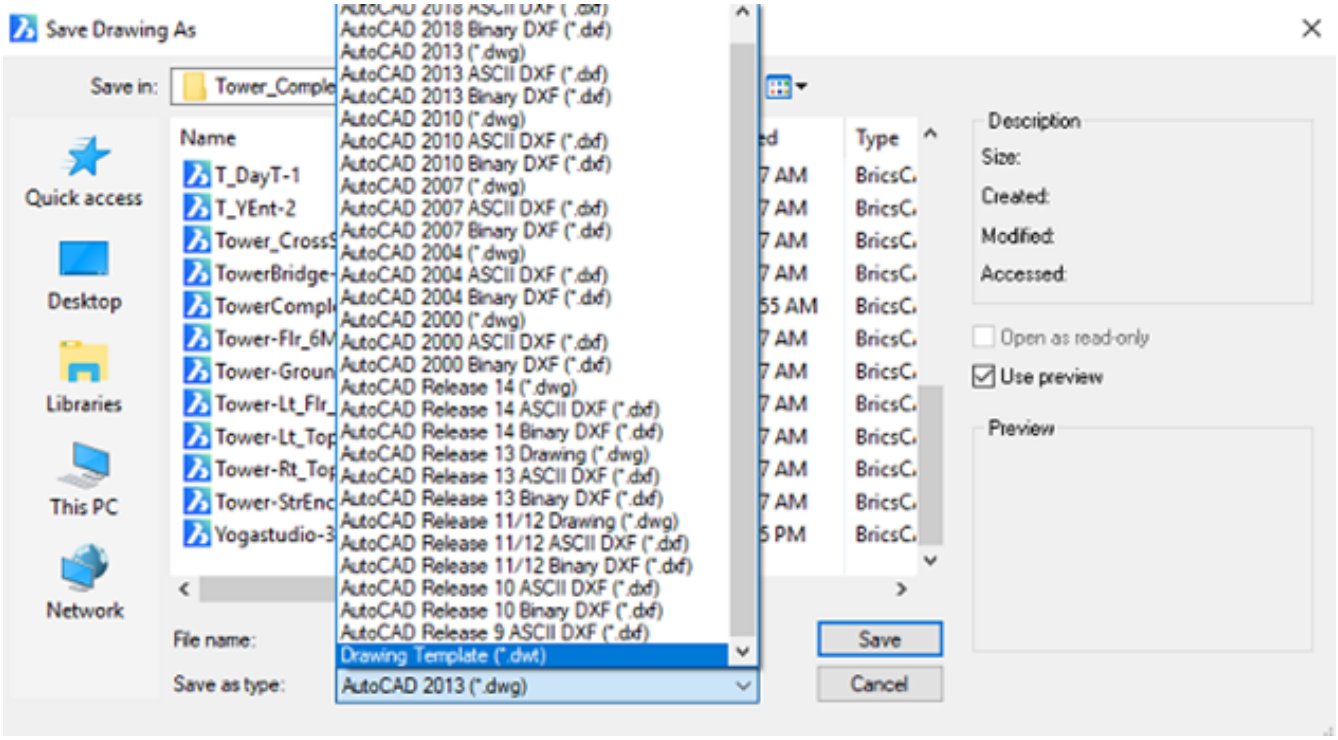
Figure 2.5



Creating a Custom Template

Creating a custom template enables you to keep your pre-loaded settings such as standard layers and standard line types. To create a custom template, open a drawing that has the desired settings you want and then save it as a drawing template (.dwt file). Your newly customized template is available in the 'templates' list. The following illustration shows the Save-As dialog for creating a custom template. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

Figure 2.6


Save Drawing As
Save in:
Tower_Comple
Quick access
Desktop
Libraries
This PC
Network
Name
T_DayT-1
T_YEnt-2
Tower_Cross
TowerBridge
TowerCompl
Tower-Flr_6M
Tower-Groun
Tower-Lt_Flr_
Tower-Lt_Top
Tower-Rt_Top
Tower-StrEnc
Yogastudio-3
Type
7 AM
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7 AM
7 AM
55 AM
7 AM
7 AM
7 AM
7 AM
7 AM
5 PM
BricsC.
Description
Size:
Created:
Modified:
Accessed:
Open as read-only
Use preview
Preview
File name:
Save as type:
AutoCAD 2013 (*.dwg)
Save
Cancel
AutoCAD 2018 Binary DXF (*.dxf)
AutoCAD 2013 (*.dwg)
AutoCAD 2013 ASCII DXF (*.dxf)
AutoCAD 2013 Binary DXF (*.dxf)
AutoCAD 2010 (*.dwg)
AutoCAD 2010 ASCII DXF (*.dxf)
AutoCAD 2010 Binary DXF (*.dxf)
AutoCAD 2007 (*.dwg)
AutoCAD 2007 ASCII DXF (*.dxf)
AutoCAD 2007 Binary DXF (*.dxf)
AutoCAD 2004 (*.dwg)
AutoCAD 2004 ASCII DXF (*.dxf)
AutoCAD 2004 Binary DXF (*.dxf)
AutoCAD 2000 (*.dwg)
AutoCAD 2000 ASCII DXF (*.dxf)
AutoCAD 2000 Binary DXF (*.dxf)
AutoCAD Release 14 (*.dwg)
AutoCAD Release 14 ASCII DXF (*.dxf)
AutoCAD Release 14 Binary DXF (*.dxf)
AutoCAD Release 13 Drawing (*.dwg)
AutoCAD Release 13 ASCII DXF (*.dxf)
AutoCAD Release 13 Binary DXF (*.dxf)
AutoCAD Release 11/12 Drawing (*.dwg)
AutoCAD Release 11/12 ASCII DXF (*.dxf)
AutoCAD Release 11/12 Binary DXF (*.dxf)
AutoCAD Release 10 ASCII DXF (*.dxf)
AutoCAD Release 10 Binary DXF (*.dxf)
AutoCAD Release 9 ASCII DXF (*.dxf)
Drawing Template (*.dwt)

2.2.2 Exploring the Interface

Overview: In this lesson, you explore the BricsCAD BIM user interface. Knowing and understanding the user interface will make you more efficient in your daily workflow. **Lesson Objectives** After completing this lesson, you will be able to:

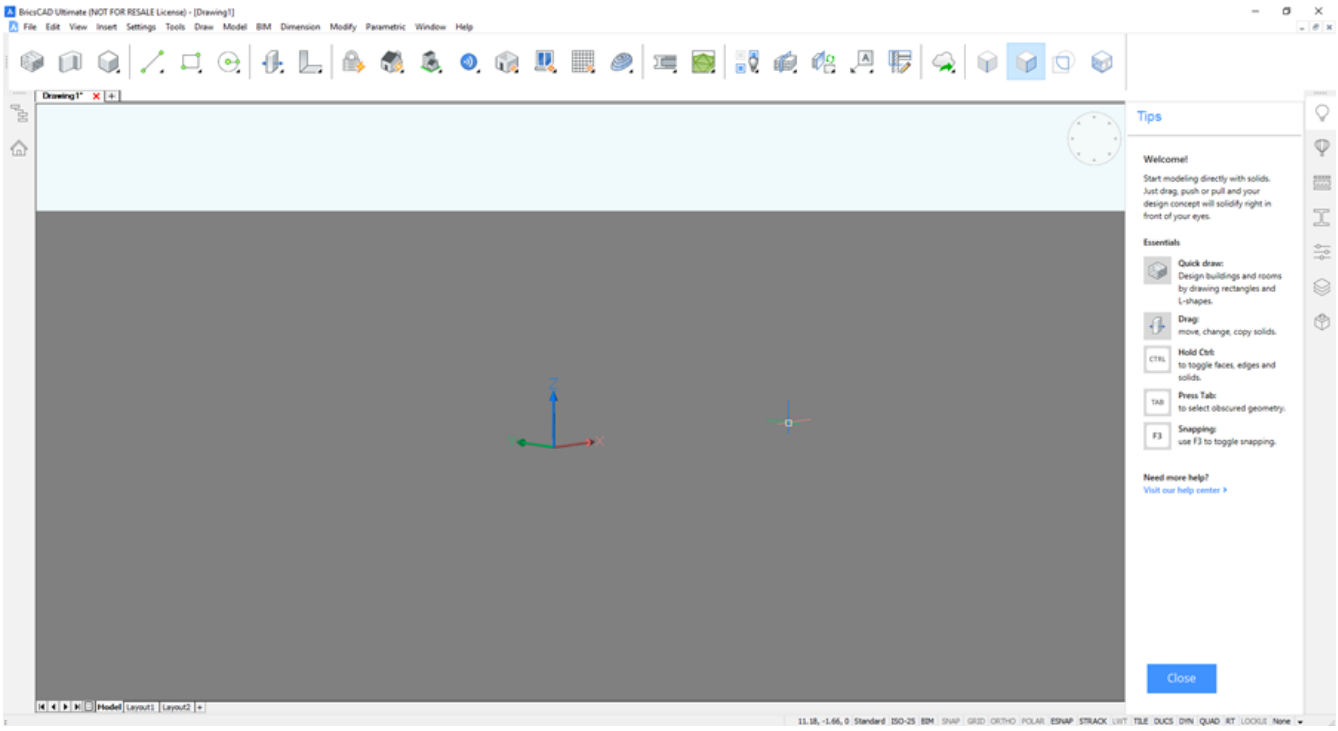
- Identify the important elements of the BricsCAD BIM user interface.
- Use the different methods in the BricsCAD BIM user interface to access different tools.

Tutorial: Exploring the BricsCAD BIM User Interface Click here (2.1.1) to watch

User Interface

The BricsCAD BIM interface includes all tools, commands, and settings required to create and edit building elements. Start by creating building elements, opening an existing drawing or importing a building model. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/BIMUserInterface.png})

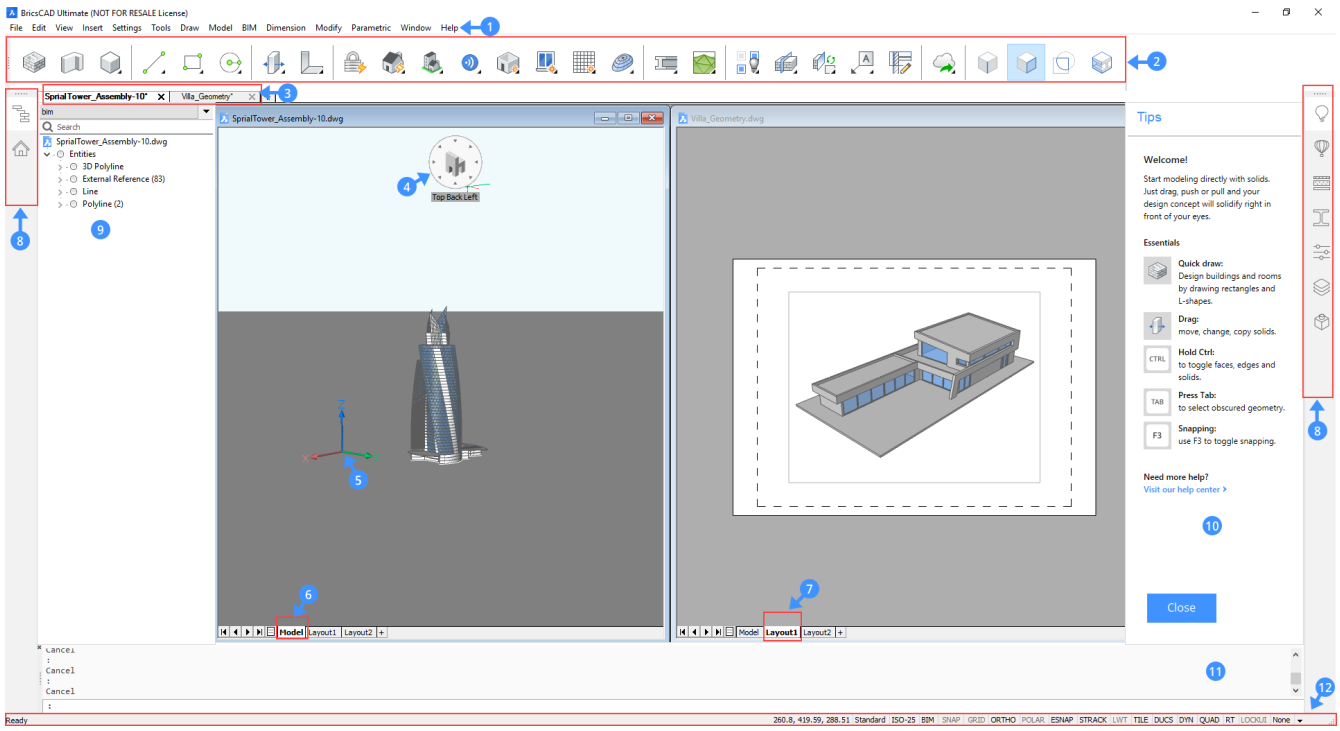
Figure 2.7



Interface Elements

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/BIMUserInterfaceElements.png}

Figure 2.8



1. Menu Bar (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/MenuBar.png}) The menu bar contains access to the most commands for file management, interface configuration, and object creation and editing.

- To display the menu bar, do one of the following: Execute the MENUBAR command and type **1** on the command line to turn it on.
- Right-click on a toolbar or a Drawing or Layout tab, then choose Menu Bar in the context menu. By clicking one of the items in the Menu Bar, you can expand it to display a flyout menu with additional functions. This enables you to access most of the tools from the toolbar. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/

Figure 2.9

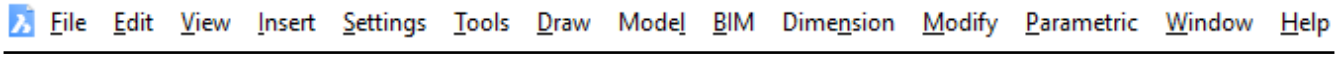
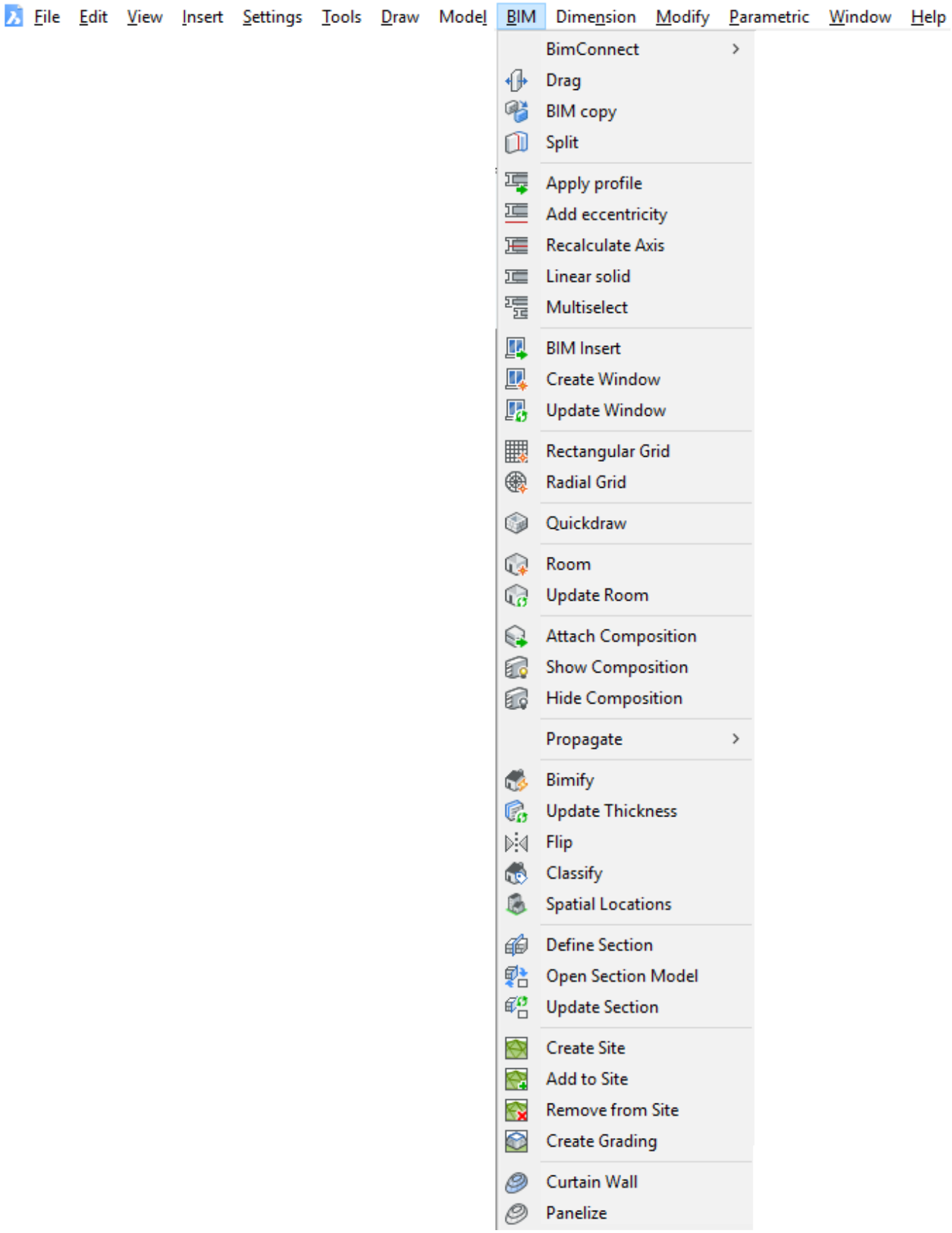


Figure 2.10


File
Edit
View
Insert
Settings
Tools
Draw
Model
BIM
Dimension
Modify
Parametric
Window
Help
BimConnect
Drag
BIM copy
Split
Apply profile
Add eccentricity
Recalculate Axis
Linear solid
Multiselect
BIM Insert
Create Window
Update Window
Rectangular Grid
Radial Grid
Quickdraw
Room
Update Room
Attach Composition
Show Composition
Hide Composition
Propagate
Bimify
Update Thickness
Flip
Classify
Spatial Locations
Define Section
Open Section Model
Update Section
Create Site
Add to Site
Remove from Site
Create Grading
Curtain Wall
Panelize

2. Toolbar In BricsCAD, toolbars are arranged in collections of similar functions within a bar of tools. Each function is displayed with an icon and is categorized by topic. Toolbars are available for all command categories. For example, a BIM toolbar is one of the toolcategoriesavailable in BricsCAD BIM.This toolbar includes the most used BIM tools for modeling. It is possible to customize existing toolbars or create your own custom toolbars. By default, BIM Toolbar is shown under the Menu Bar. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingSta

Figure 2.11

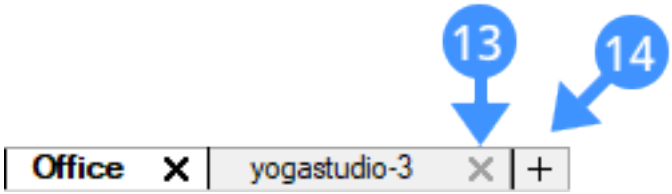


3. Working with Document Tabs Document tabs allow you to:

- Easily switch between all open drawings.
- Drag and drop the tabs to change their order.
- Click the X icon (13) at the right-hand side of the tab to close it.
- Right-click on a document tab to display a context menu.
- Click the + (14) icon at the end of the document tab to open a new drawing.

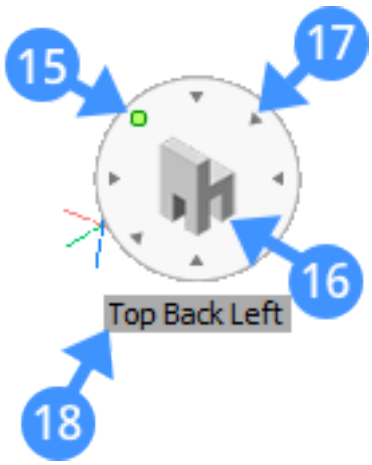
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/DocumentTabs.png})

Figure 2.12



4. Look from Widget The Look From Widget appears in the upper right corner of the drawing area. When you hover the cursor over the widget, a small green rectangle appears (15), as does the preview of a chair (16). Clicking the cursor over one of the triangle shapes (17) shows what the 3D view will look like (18). The default location of the widget is the top right corner of the drawing area. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Getting

Figure 2.13

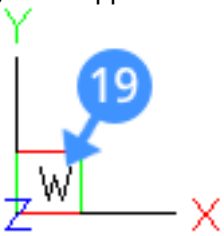


5. UCS Icon UCS stands for User-defined Coordinate System. The UCS icon shows the nature of the current coordinate system. 

The display of the icon is controlled through the UCS icon (6.30.3) system variable and the position of the UCS icon is defined by the UCS icon position system variable.

6. Model Tab When you start a drawing session, your initial working area is called Model Space. Model Space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or User Coordinate System (UCS). You view and work in model space while using the Model tab.

7. Layout Tab Each drawing has at least one layout and each layout is composed of one or more viewports. Each viewport can show a different part of the drawing at a different scale. Paper space is a work environment thatprovides themodel space viewat a given scale, dependingon the sizeof the paper. In each layout, you can add the entities needed to complete a printed copy of the drawing; title blocks, legends, frames, etc. These entities are only visible in the layout when you add. They are not visible in the other layouts or in the model space.

UCS in Layout Mode In paper space of layout modes, a W (19) letter appears in the UCS icon, which means the WCS is active. **Note:** WCS stands for World Coordinate System. 

8. Tool Panel In BricsCAD BIM, the Tool Panel is displayed both sides of the workspace. The Tool panel consists of BIM panels. The BIM panels are dockable and thus they are movable. The panels can be moved to another docked panel using the drag and drop method. The Tool Panel on the left-hand side displays the Structure (20) and Project Browser (21) panels. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/Too
The Tool Panel on the right-hand side displays Tips, Components, BIM Compositions,BIM Profiles, Properties, Layers, and Mechanical Browser panels (default). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Ge
Note: Displaying one of the dockable panels using the context menu or command line will automatically add its icon on the Tool Panel. The following illustration shows the Sheet Sets icon in the Tool Panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/ToolPanelSheetSets.png})

Figure 2.14

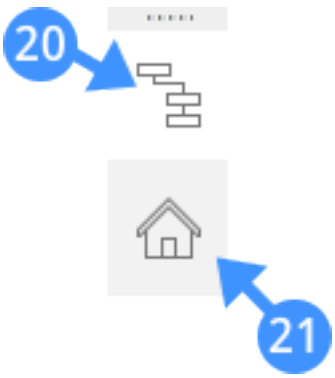


Figure 2.15



Figure 2.16



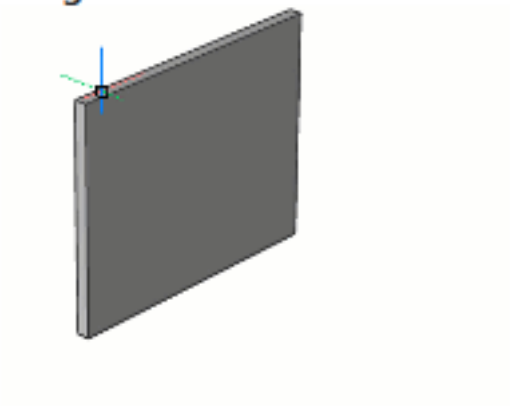
9. Structure Panel The dockable Structurepanel displays a structured tree view of the drawing content. This structured view is fully configurable.Configurations can easily be saved and recalled.

10. Tips Panel Tips Panel gives general information about the tools and commands. The tips can change depending on the active command. The tips can only be seen when the panel is open. The following illustration shows the Tips Panel when the BIMDRAG command is active. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BI

Figure 2.17

Tips

Drag



Select a major solid face, and Drag can:

Drag the selected solid and keep minor faces connected.

Change the thickness of the selected solid.

Create a copy of the selected solid.

Select a minor solid face and Drag can:

Change the height or length of a solid.

Connect the selected face to another solid.

Need more help?

Visit our help center ›

Close

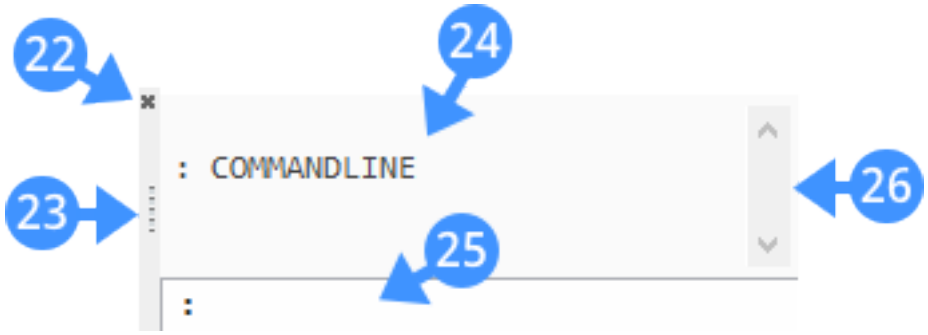
11. Command Bar The command bar consists of two fields.

- In the lower field, you can type the commands and BricsCAD shows prompt, options and other information regarding the execution of commands here. If the command bar is closed, this information shows in the Status Bar.
- The command history displays in the upper field of the command bar.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/CommandBar.png})

Close (22)	Click to close the command bar
Grip (23)	Press and hold the left mouse button to drag the command bar
Command history (24)	Shows the previously typed commands
Command edit (25)	Type the commands and command options here
Scroll buttons (26)	Click the buttons to scroll the command history

Figure 2.18



12. Status Bar The Status Bar sits along the bottom edge of the BricsCAD application window. It contains a lot of information about the settings in the current drawing. The status bar consists of 16 fields. All of these fields are optional, except the Status field(1). If you click the small black down arrow button at the right end of the status bar(2)a list of all field displays. Click a field to toggle its display. Right-click a field to display an option menu for this field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/StatusBar.png})

Status field (27)	Displays the status of the software
List button (28)	Click the arrow to display a list of available Status Bar fields
Fields (29)	Click a field to toggle it ON/OFF. Right-click to display a context menu.

Figure 2.19



Ribbon A ribbon organizes tools in a series of panels which are grouped in different tabs. Each tab is separated by a series of panels, and each panel contains a group of buttons, flyouts, and drop lists. The following illustration shows the ribbon's Model Tab (30) and the File panel (31) which are both indicated in a red rectangle shape. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/Ribbon.png}) **Note:** To display the Ribbon, right-click on a toolbar, then choose Ribbon in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbri

Figure 2.20

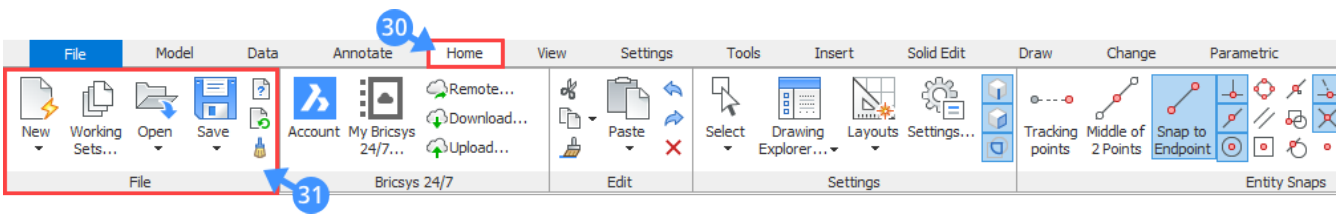
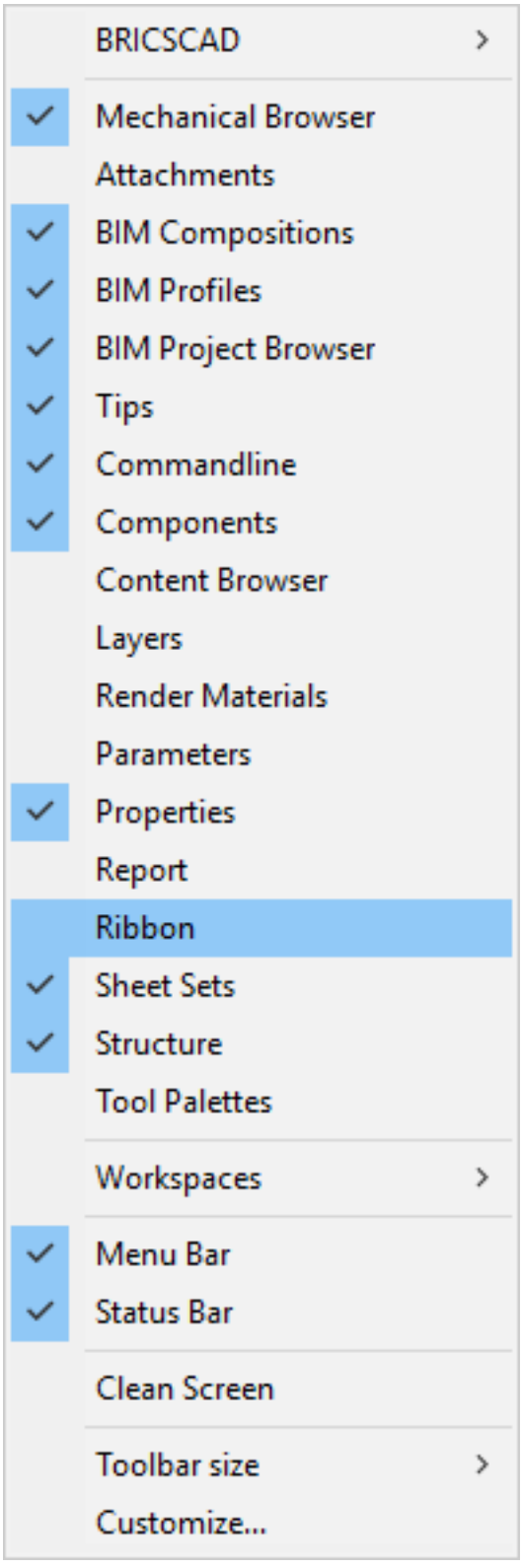


Figure 2.21


BRICSCAD
Mechanical Browser
Attachments
BIM Compositions
BIM Profiles
BIM Project Browser
Tips
Commandline
Components
Content Browser
Layers
Render Materials
Parameters
Properties
Report
Ribbon
Sheet Sets
Structure
Tool Palettes
Workspaces
Menu Bar
Status Bar
Clean Screen
Toolbar size
Customize...

2.2.3 Quad Cursor

Overview: In this lesson, you learn how to display the entity's properties and to quickly and easily modify the 3D solids using the Quad cursor. **Lesson Objectives** After completing this lesson, you will be able to:

- Manage the context-sensitive Quad cursor.

Tutorial: How to use Quad Cursor & Rollover Tips Click here (2.1.2) to watch

About

The Quad cursor menu, or Quad for short, is an alternative to the command line or toolbars, offering a rich set of tools while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs. The rollover tips are a limited set of properties displayed when selecting or hovering over an entity. To toggle the **Quad** on/off, do one of the following:

- Click the Quad field in the Status bar.
- Press the F12 function key.
- Edit the QUADDISPLAY system variable in the Settings dialog.
- Type QUADDISPLAY in the command bar, then choose an option.

To toggle the **Rollover Tips** on/off, do one of the following:

- Click the RT field in the Status bar.
- Type rollovertips in the command bar then set a value.

Quad Layout

The Quad has a floating feature set that adjusts its content, depending on what you are or are not highlighting, and what you may have selected in the current workspace

- Based on the highlighted entity type, the Quad displays the most recently used tool (1) with the entity name(2).

Note: When multiple entities are under the cursor, hit the TAB key to cycle through the different entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADNoRT.png})

- When the Rollover Tips setting is on, a limited set of properties (3) of the entity under the cursor display in the Quad.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADRT.png}) The property values in the Rollover tips can be edited. The following figure shows the color of an arc being changed using the Rollover Tips. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/RTColorChange.pr
Note: To display the Rollover tips, simply click the title bar of the Quad after expanding it.

- When the cursor is moved on the Quad, a number of tool tabs display at the bottom (4).

Note: The number of tool tabs is controlled by the **Quadwidth** user preference (default = 6). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADToolTabs.png})

- When you move the cursor over the tool tabs title bar (5), the tool groups expand (6) under the cursor, which provides you with more tool options.

Figure 2.22

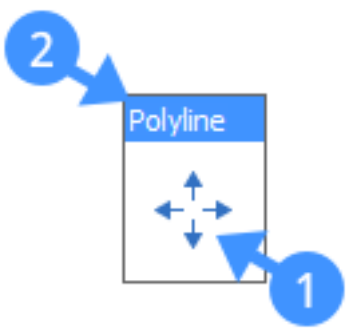


Figure 2.23

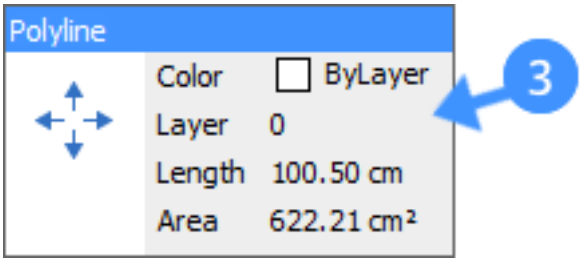


Figure 2.24

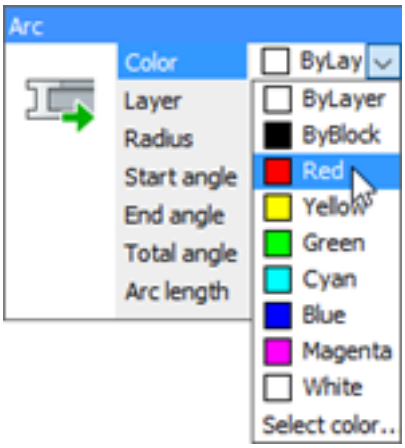


Figure 2.25

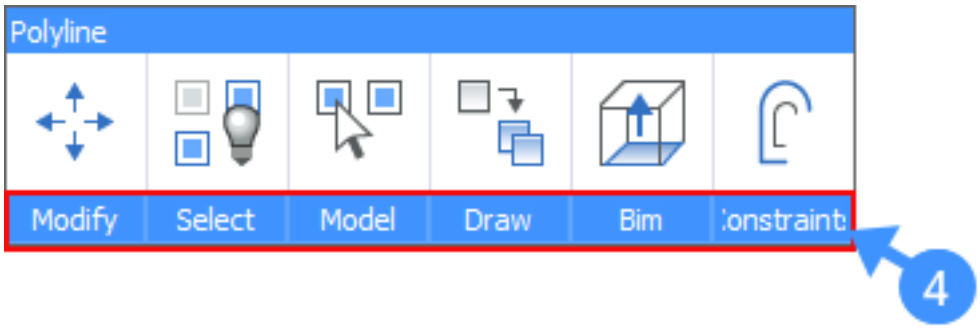


Figure 2.26

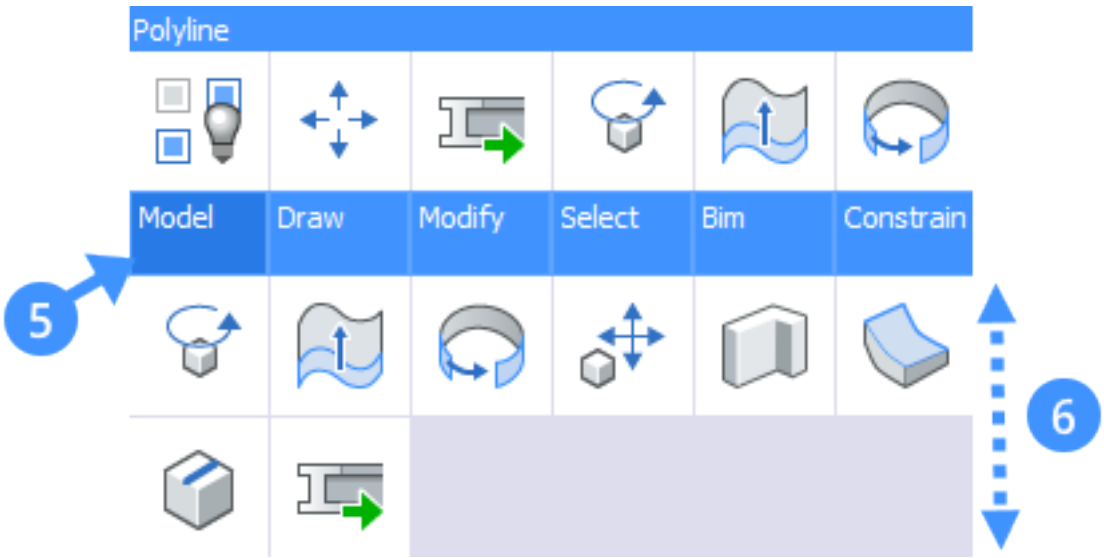


Figure 2.27

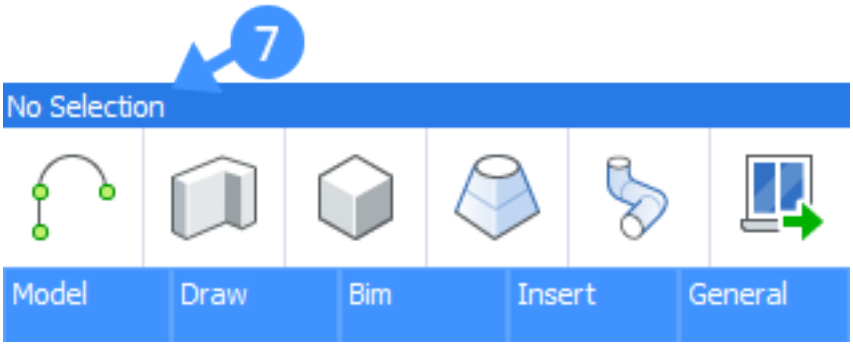
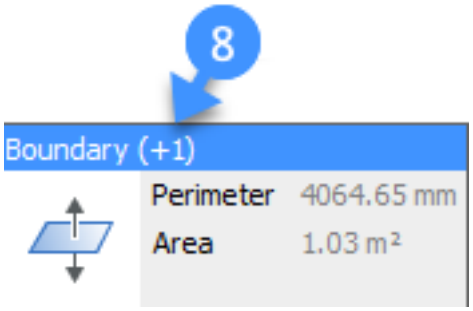


Figure 2.28



Note: The number of available tool groups and the number of tools within a tool group depend on the entity type under the cursor. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADModelTools

- The Quad can also be accessed in the no selection If the cursor is on an entity, hold down the right mouse button longer to display the No Selection Quad (7). Or when there is no selected/highlighted entity in the drawing, right-click to display the No Selection Quad.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADNoSelection.png})

- When an entity is selected and another one is highlighted, next to the entity type (title bar on the top) a plus one (8) appears in the parentheses. The count, regardless of the type, increases each time you select an entity. To reveal the selected the entity types which are hidden in the parentheses, expand the quad by moving your cursor on it.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADMultipleSelected.png})
Note: The Quad also adapts to the type of entities that have been selected. If you select two surfaces, you will be provided with different tool options in the Quad, then if you selected a solid and a surface.

About: Selecting entities

The Quad adapts the type of entities that you have been selected. If you select a surface, you will be provided with different tool options in the Quad, then selecting a solid or edge.

- Selecting an **edge**, the QUAD appears showing the edge length, slope, and elevation (9).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADRTSolidEdge.png})

- Selecting a **solid face**, the surface area is shown in the Quad (10).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADRTFace.png})

- Highlighting a **boundary**, the Quad displays the perimeter and area of this boundary (11).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADRTBoundary.png})

Figure 2.29

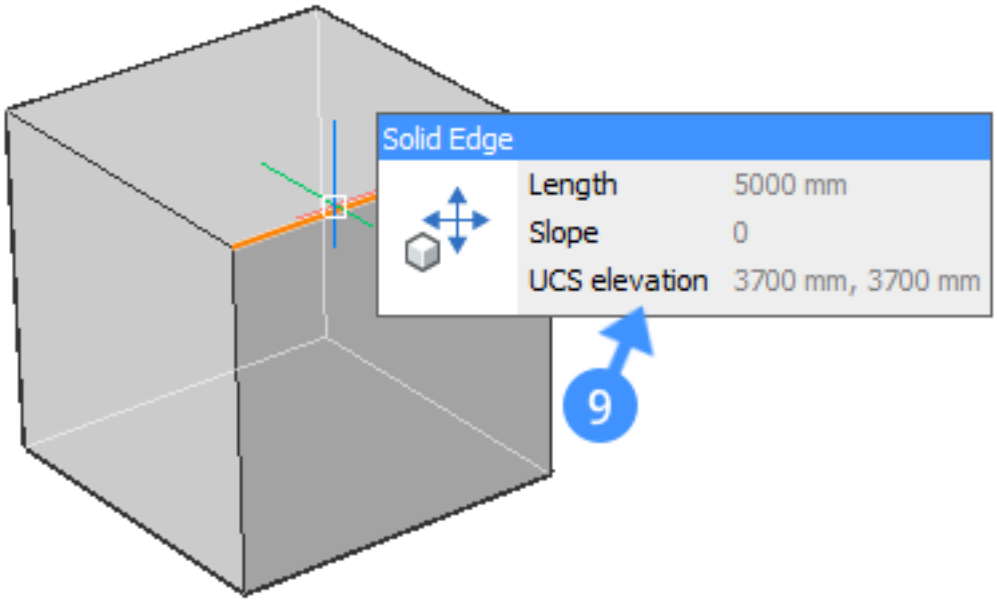
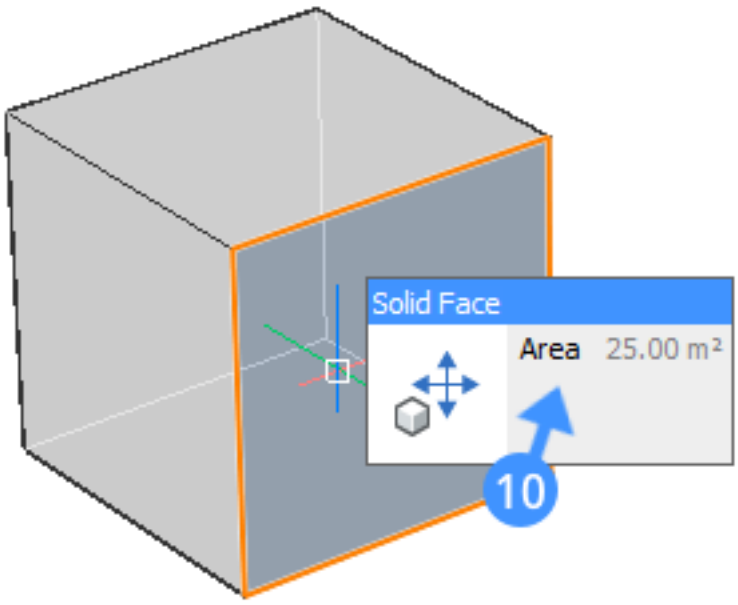


Figure 2.30



- Highlighting a **solid**, the Quad displays the color, layer, and elevation of the solid (12)(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADRTSolid.png})

Tutorial: How to use Quad Cursor & Rollover Tips Click here (2.1.2) to watch

Figure 2.31

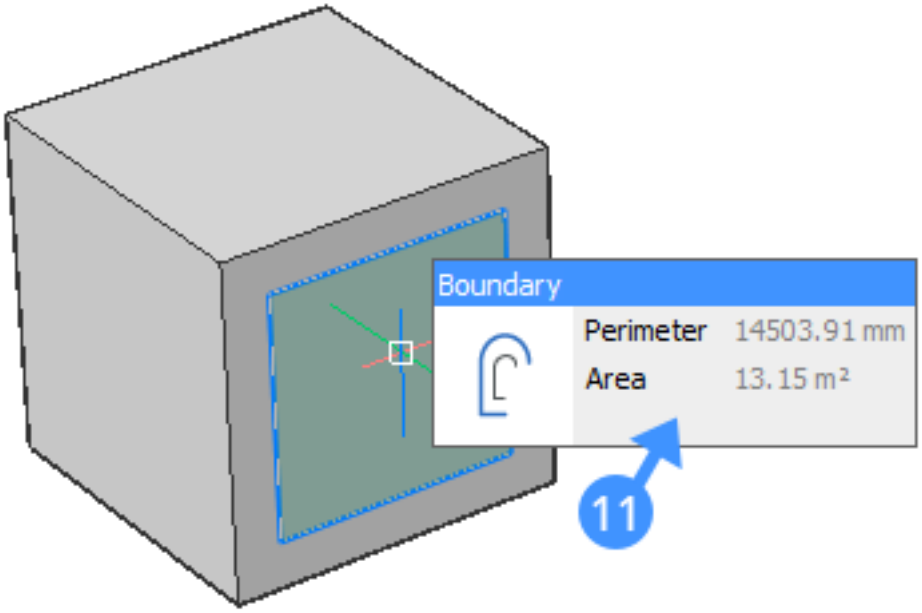
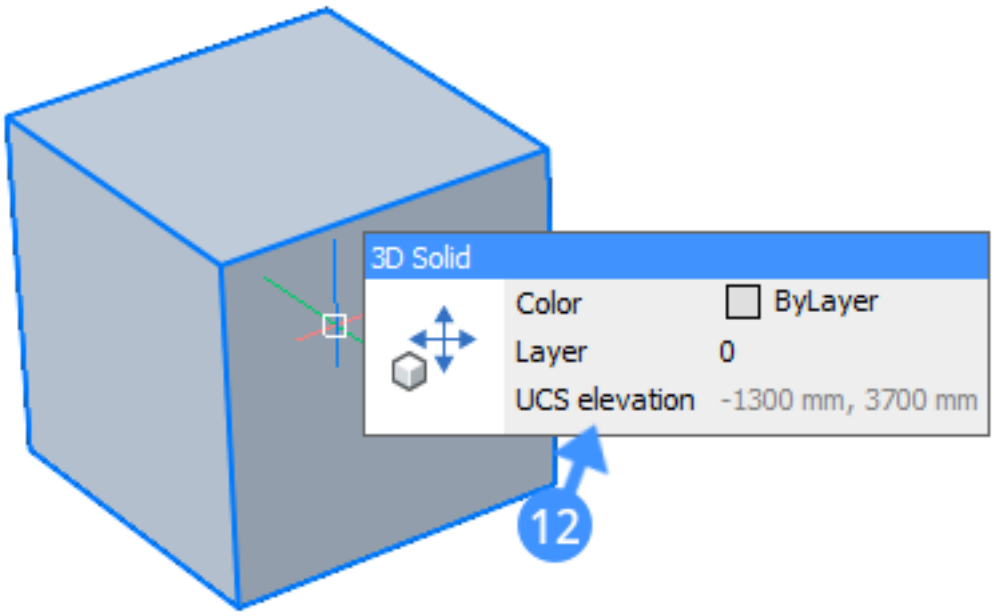


Figure 2.32



Procedure: Using the Quad

The following are general steps to use the Quad cursor. **Note:** The Quad and Rollover tips are both enabled.

1. Hover the cursor over the entity you want to manipulate.

The entity under the cursor highlights. The Quad displays a single icon (13), which is the most recently used tool with this entity type, and a limited set of properties (14) of the highlighted entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADHighlightMostRecent.png})

2. Right-click to launch the recently used tool. Or move the cursor to the single icon to expand the Quad (15), then select one of the available tools to manipulate the entity.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/GettingStarted/QUADHighlightTools.png})

3. Move the cursor over the blue field to further expand the Quad.

All Quad tool groups for the current workspace display. When the first tool group is expanded, click an icon to use a tool or move the cursor to a different tool group.

Figure 2.33

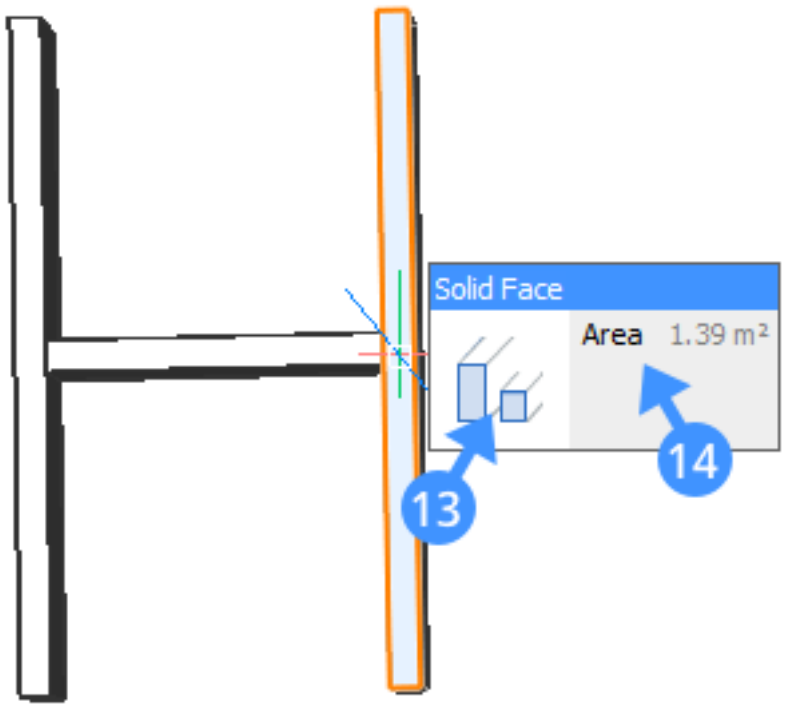
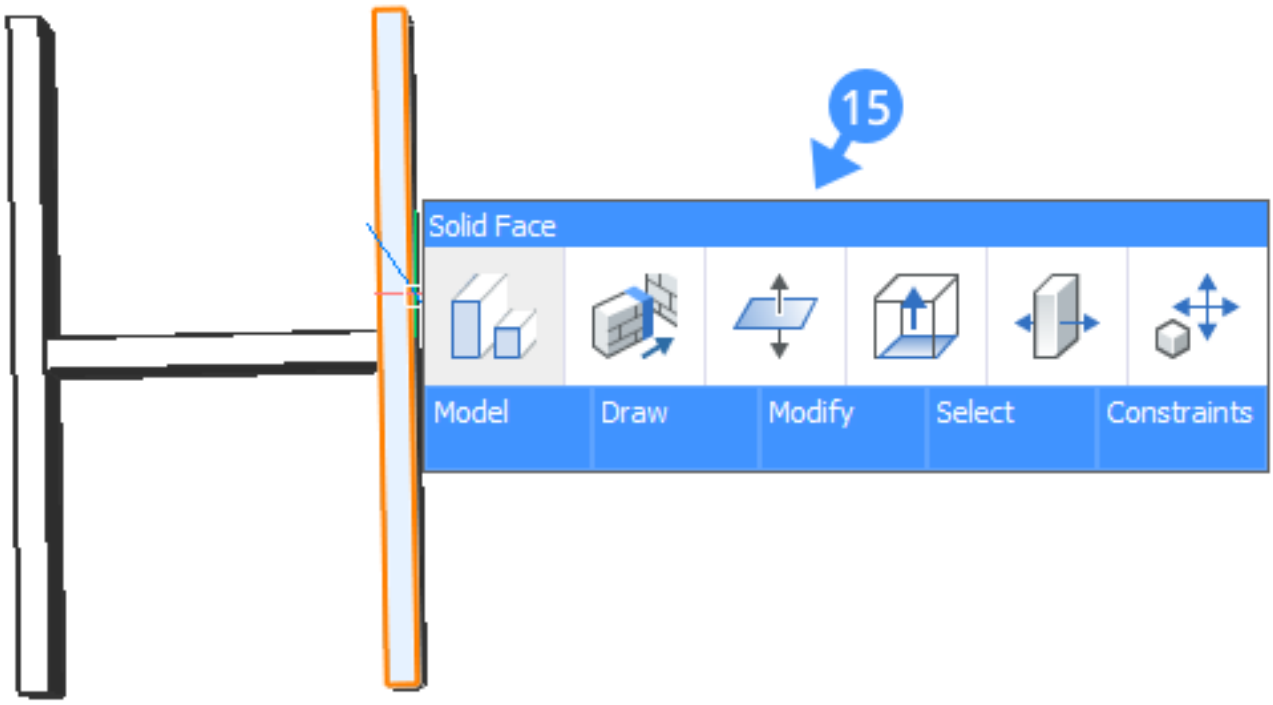


Figure 2.34



2.2.4 Viewing Entities

Command: HideObjects (6.16.11),IsolateObjects (6.17.21),UnisolateObjects (6.30.7) **Overview:** In this lesson, you will learn to use different tools to view and manage your entities in a drawing. **Lesson Objectives** After completing this lesson, you will be able to:

- Hide selected entities.
- Isolate selected entities to keep only those visible.
- Reveal all hidden entities.

Tutorial: How to hide and isolate objects Click here (2.3.8) to watch

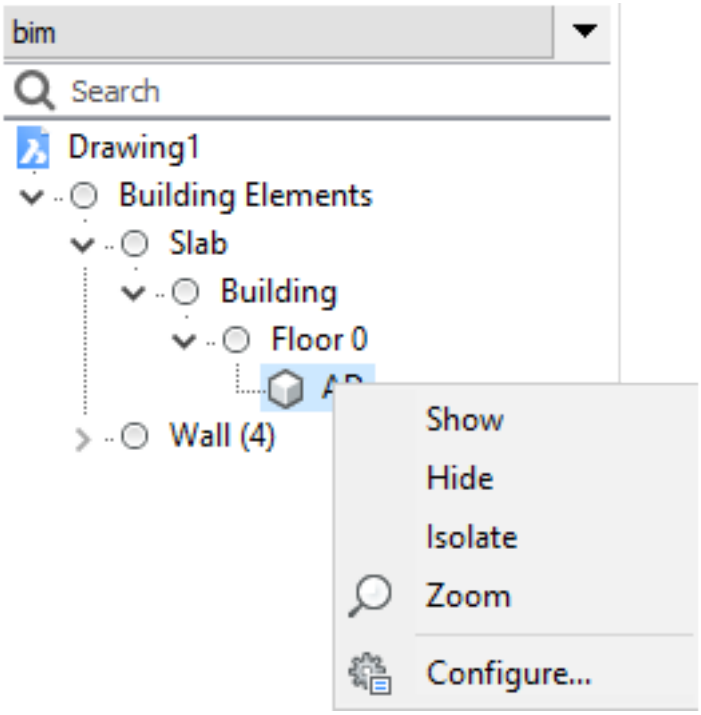
About

If you work with many entities in a drawing, everything that you have in the drawing is visible in the view. When you want to temporarily make only a few entities or a particular entity visible and manage them in a view, you can use the Hide or Isolate tools.

Hiding Entities

The Hide tool temporarily hides the selected entities in the view.  Toolbar: **View > Hide Entities** Quad: **Select Tab > Hide Entities** Keyboard: **HIDEOBJECTS** Ribbon: **Model > View > Hide Entities Optional**Structure Browser can also be used to hide and isolate selected entities as well as revealing all hidden entitiesin the model space.This is an easy way to quickly hide or isolate a certain category of objects or entities with a certain property value etc. To do so, right-click on the entity in the Structure Browser and choose one of the options in the context menu (see below image). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/StructureBrowserVie

Figure 2.35



Procedure: Hiding Entities

1. Select one or more entities in the drawing area.
2. Select **Hide Entities** from the Quad. All entities that have been selected are hidden in the view. Following illustration shows an entire model on the left. In the second image, the roof was temporarily hidden. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/House.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/HouseHiddenRoof.png})

Figure 2.36



Figure 2.37



Isolating Entities



The Isolate tool temporarily hides all entities except those that have been selected. Toolbar: **View > Isolate Entities** Quad: **Select Tab > Isolate Entities** Keyboard: **ISOLATEOBJECTS** Ribbon: **Model > View > Isolate Entities**

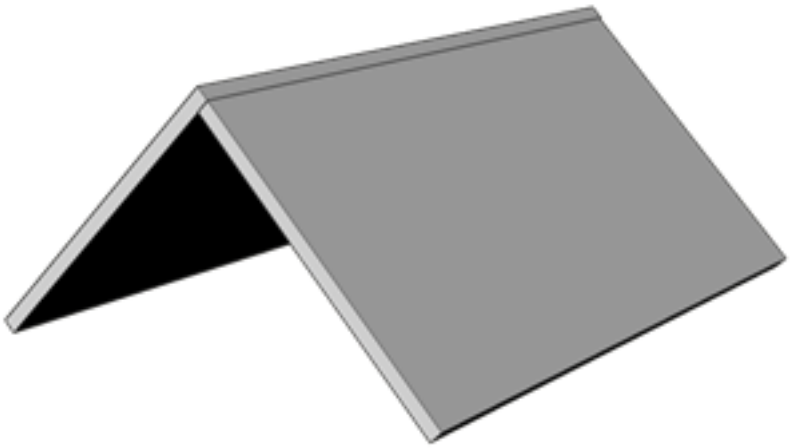
Procedure: Isolating Entities

1. Select one or more entities in the drawing area.
2. Select **Isolate Entities** from the Quad. All entities that have been selected before are isolated in the view. Following illustration shows the same building model from the below. In the second image, the roof was temporarily isolated. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/Hou
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/HouseIsolatedRoof.p

Figure 2.38



Figure 2.39



Showing Entities

The Show Entities tool brings back all hidden entities in the view. Toolbar:**View > Show entities** Quad: **General Tab > Show entities** Keyboard: **UNISOLATE, UNHIDE** Ribbon: **Model > View > Unisolate Entities** If you have any hidden entities in your drawing, you can reveal them in the view using the show entities tool. The following illustration shows the isolated roof elements from the above example. The second image shows the result after using the Show Entities tool. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/HouseIsolate (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/House.png})

Figure 2.40

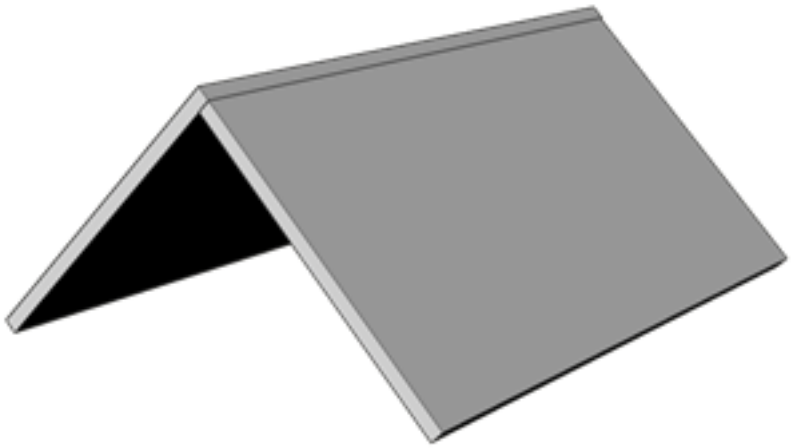


Figure 2.41



2.2.5 Selecting Entities

Overview: In this lesson, you learn how to highlight and select 3D model entities in BricsCAD BIM. **Lesson Objectives** After completing this lesson, you will be able to:

- Identify the difference between highlighting and selecting.
- Use the different selection modes.
- Highlight and select entities and sub-entities.
- Select entities and sub-entities using the selection boxes.

Tutorial: How to select elements Click here (2.3.11) to watch

About

In BricsCAD, when you hover over an object, it will be highlighted. After highlighting an entity, you left click to select the highlighted entity, then it will be selected. **Highlighting**, you only can highlight one entity at a time vs. **Selecting,** you can select as many entities as you want. Most of the modeling process can be done by highlighting the entity and using the Quad. For example, when you highlight a surface, the Quad will open, pick the command you want, without having to select the surface first.

Tutorial: How to detect boundaries in BricsCAD BIM Click here (2.3.6) to watch

Selection Modes

The selection modes allow you to control which sub-entities (faces, edges, and boundaries) should highlight in selection preview, and can be selected. **Ribbon:** Home tab > Settings panel > Selection modes (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RibbonSettingsPanel.png}) **Note:** To display the Ribbon, right-click on a toolbar, then choose Ribbon in the context menu. **Settings:** Program options > Selection > Selection Preview > Selection Modes (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Mo
Keyboard: SELECTIONMODES In the following steps, each selection mode option is given with its figure and explanation.

- **Enable detection of 3D solid edges** ( ): controls whether 3D solid edges are highlighted by selection preview, and can be selected. If you enable this option, you can select the edges of 3D solids by clicking it once.
- **Enable detection of 3D solid faces** ( ): controls whether 3D solid faces are highlighted by selection preview, and can be selected. If you enable this option, you can select the faces of the 3D solids by clicking it once.
- **Enable boundary detection** ( ): controls whether closed boundaries in XY-plane of the current coordinate system or on the face of 3D solids are detected. If you enable this option, you can select closed boundaries from the faces of 3D solids by clicking it once.
- **Enable selection of Sides and Ends** ( ): controls the bimSides and bimEnds visibility of all linear elements.

Note: bimSides/Ends are only available on classified linear elementswith attached library profile.

Figure 2.42

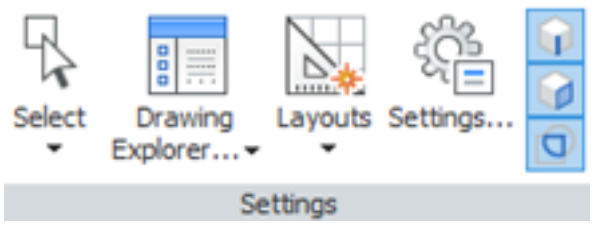


Figure 2.43

Selection modes	0x0007 (7)
1	Select edges
2	Select faces
4	Select detected boundaries

Procedure: Changing selection modes setting

A sub-entity is a face, edge or boundary of a 3D solid. To control which sub-entities you select in your model do one of the following: **Note:** the default value of the selection modes = 6 which means the detection of 3D solid edge is disabled, while the detection of 3D solid faces and boundaries are enabled.

- Click the buttons on the Selection Modes toolbar to edit the value of SELECTIONMODES.

The following illustration shows the sum of the values (1) of the selection modes system variable when the select edges option is disabled (2) after clicking the checkbox once, the select faces and selected detected boundaries are enabled (3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SettingsSelec

- Click the buttons on the settings panel on the Ribbon. The selection mode options can be enabled or disabled by clicking the button once. When one of the selection mode options is enabled this will be indicated in blue.

The following illustration shows when the **select edges** option is inactive (4) whereas the **select faces** and **select boundaries** are active (5).



- Type SELECTIONMODES in the command line and change the current parameter.
- Use the CTRL key on the keyboard to temporarily toggle between the different selection modes. These key functions enable you to select any type of entities.

To display the bimSides/Ends for linear elements you can use ”Display Sides and Ends” system variable in the settings dialog box. The following illustration shows the BIM > General system variable when the ”Display Sides and Ends” is enabled (6) after clicking the checkbox once. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/

Figure 2.44

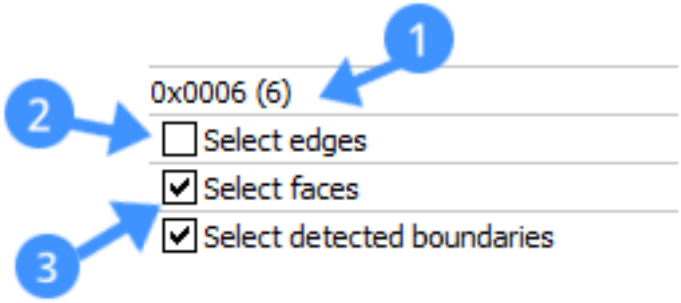
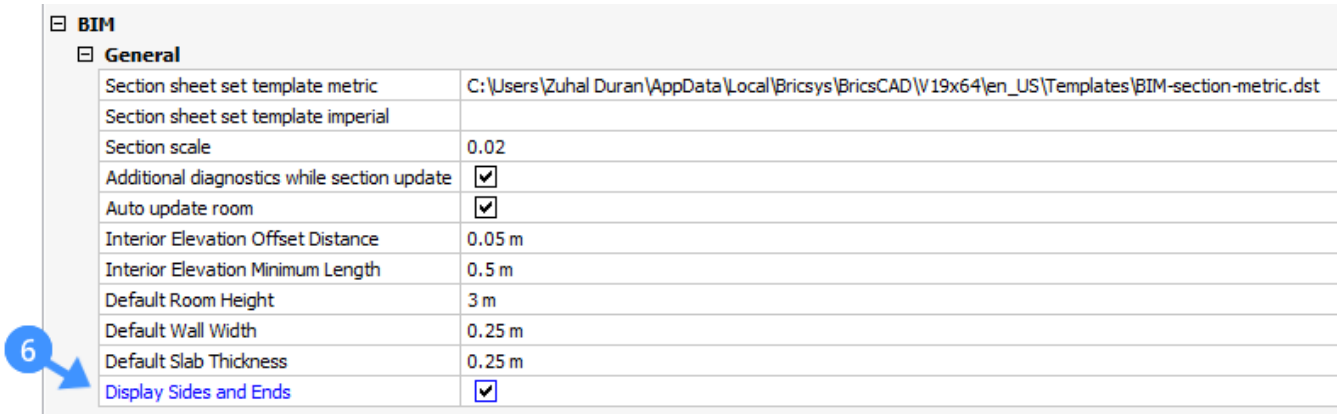


Figure 2.45



Procedure: Selecting sub-entities

When select Edges is off, select Faces is on, Boundary Detection is on (default), do one of the following:

- To highlight a solid face, hover over the face with your mouse cursor. The highlighted face displays in orange (7). To select the face, left-click it when the face is highlighted. To select multiple faces of the solid, left-click them one by one.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectedFace.png})

- To highlight a solid, hover over one of its faces while holding down the CTRL key. The solid displays in blue (8). To select the solid, left-click it while the solid is highlighted. To select multiple solids left-click them one by one.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectedSolid.png})

- To highlight an edge, hover over the edge of the solid while holding down the CTRL key. The edge displays in orange (9). To select the edge, left-click it while the edge is highlighted. To select multiple edges left-click them one by one.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectedSolidEdge.png})

- To highlight a boundary, hover over the detected boundary on the face of the 3D solid.

When the detected boundary is highlighted, it displays with a color. By default, inside of the boundary displays in green (10) and the outer edge of the boundary displays in blue (11). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel
Note: This setting can be changed by typing BOUNDARYCOLOR on the command line (default value = 95).

Figure 2.46

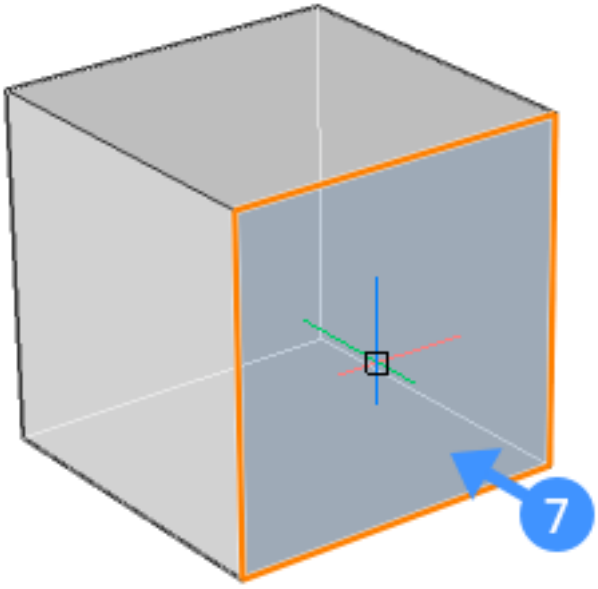


Figure 2.47

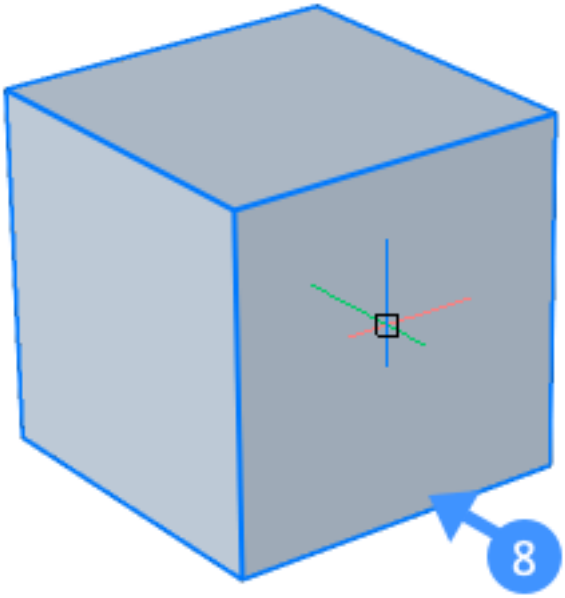


Figure 2.48

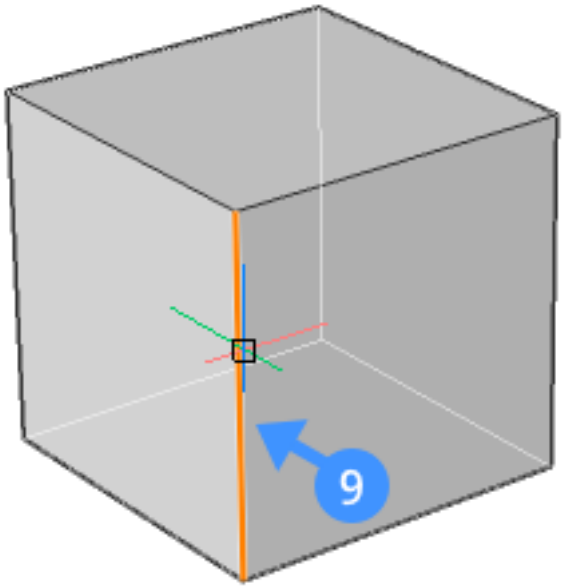
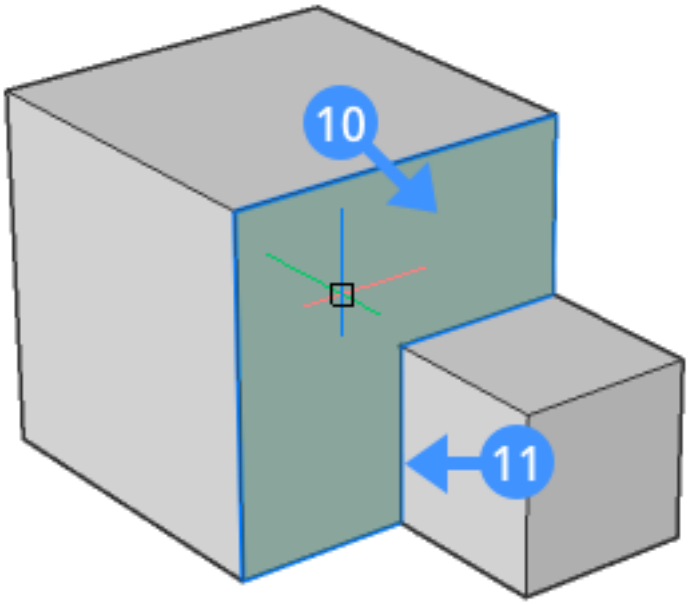


Figure 2.49



Procedure: Selecting obscured sub-entities

In your drawing, some entities or sub-entities (faces, edges, and boundaries) might be hidden behind other entities. The TAB key enables you to cycle through sub-entities. The following steps show how to highlight a hidden face that lies beneath the cursor. When select Edges is off, select Faces is on, Boundary Detection is on (default).

1. Place your cursor on one of the solid faces.

In the example below, the top face of the front solid is highlighted (12).

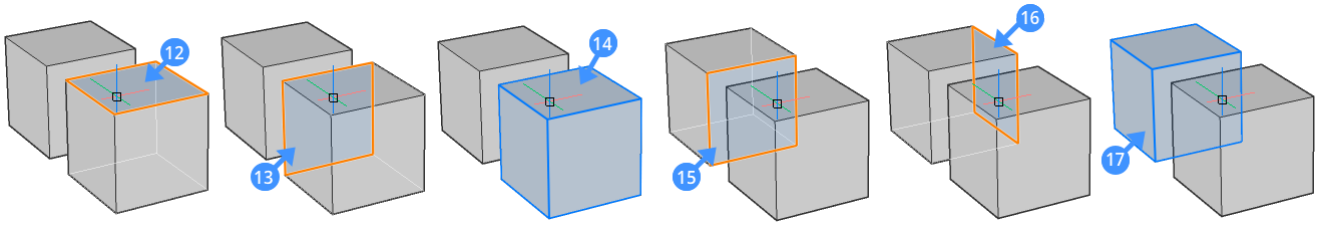
2. Press TAB key to cycle through the hidden entity.

Now, the back face of the front solid is highlighted (13).

3. Repeatedly pressing the TAB key, while keeping the cursor at the same position, allows you to highlight the entire solid in the front (14) and the front face (15) and right face (16) of the rear solid. Finally, the entire rear solid is highlighted (17).
4. When the entity or sub-entity that you want to select is highlighted, left-click it to select. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectingObscuredEntities.png})

Tutorial: How to use selection boxes Click here (2.3.24) to watch

Figure 2.50



Procedure: Selecting multiple entities using the selection boxes

The selection boxes allow you to select one or more than one entity at a time. To use a selection box, left-click on any empty space in the drawing area, and move the cursor until the selection box covers the entities that you want to select. There are two types of selection boxes in BricsCAD; **Blue selection box (18)** appears when you create a box from left to right. The blue selection box will only select entities that are entirely covered by this selection box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BlueSelectionBox.png})

Green selection box (19) appears when you create a box from right to left. The green selection box will select all elements that are entirely covered by the selection box and all the elements that intersect with one of the edges of the box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/GreenSelectionBo
The following steps show how to select multiple entities using the blue selection box: **Note:** by default, selection box only selects entire entities (e.g. solids, lines, polylines, blocks,...) but not sub-entities (e.g. solid faces, solid edges).

1. Click and move the mouse to the right to create a blue box around the entities you want to select. When the entity is completely inside the window, it will be added to the selection set.

The example below shows the blue selection box usage when the selection mode is set to Solids (default). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BlueSelectionBoxSolids.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BlueSelectionBoxSolids2.p
The blue selection box covers the left cube completely and thus only this cube is selected in the model.

2. To select faces of 3D solids and surfaces, press CTRL key once during window selection. The icon (20) next to the cursor indicates the selection mode. The faces that are completely inside the window will be selected when the selection mode is set to Faces. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/M (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BlueSelectionBoxFac

3. To select edges of 3D solids and surfaces, press CTRL key twice during window selection. The 'Edges' icon (21) appears next to the cursor. The edges that are completely inside the window will be selected when the selection mode is set to Edges. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtech (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BlueSelectionBoxEdg

The following steps show how to select multiple entities using the green selection box:

4. Click and move the mouse to the left (crossing window) to create a green box around the entities you want to select. When the entities overlap the window or are completely inside the window, they will be added to the selection set.

The example below shows the green selection box usage when the selection mode is set to default. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/GreenSelectionBoxSolids.png}) (\vref {fig:C:/Dev/ 2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/GreenSelectionBoxSolids2.png}) The green selection box selects both cubes in the model as the selection box overlaps with the two cubes.

5. To select faces of 3D solids and surfaces, making use of the green selection box, press CTRL key once during window selection. The 'Faces' icon appears next to the cursor (22). Any faces that are completely or partially inside the crossing window will be selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM

Figure 2.51



Figure 2.52

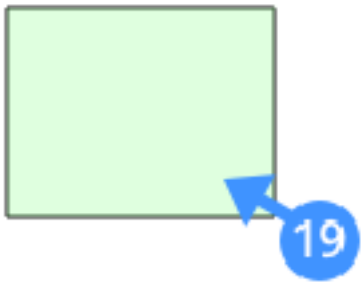


Figure 2.53

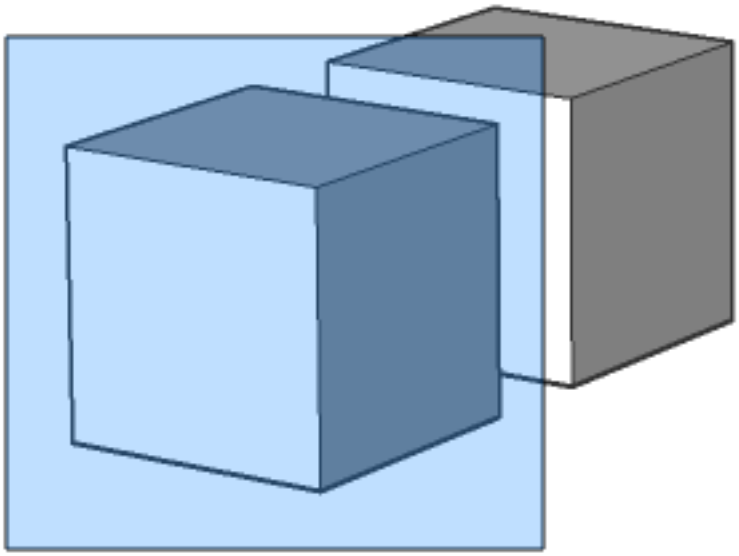


Figure 2.54

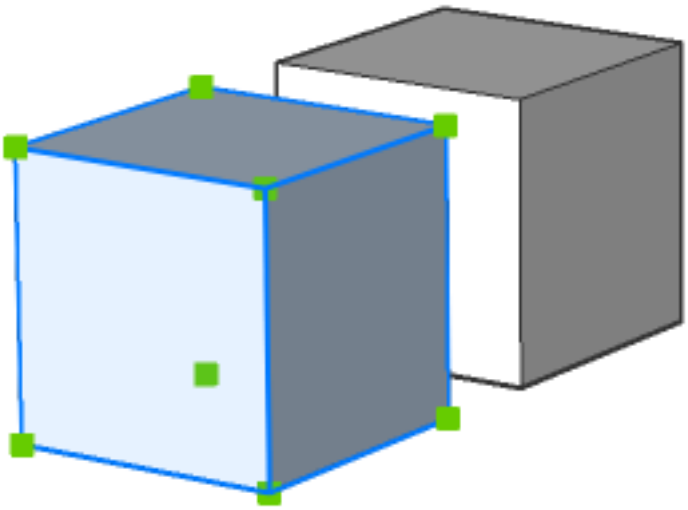


Figure 2.55

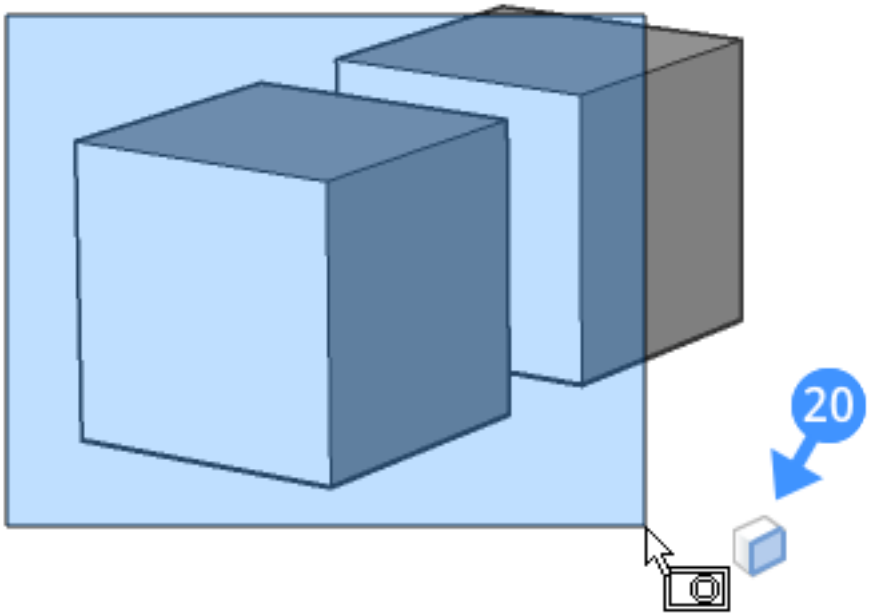


Figure 2.56

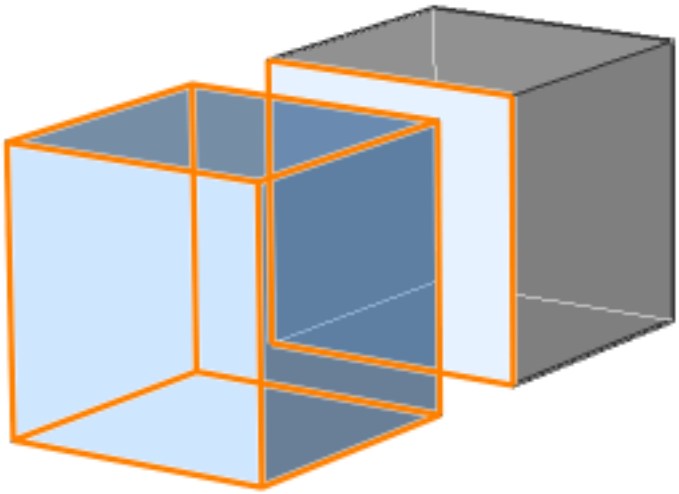


Figure 2.57

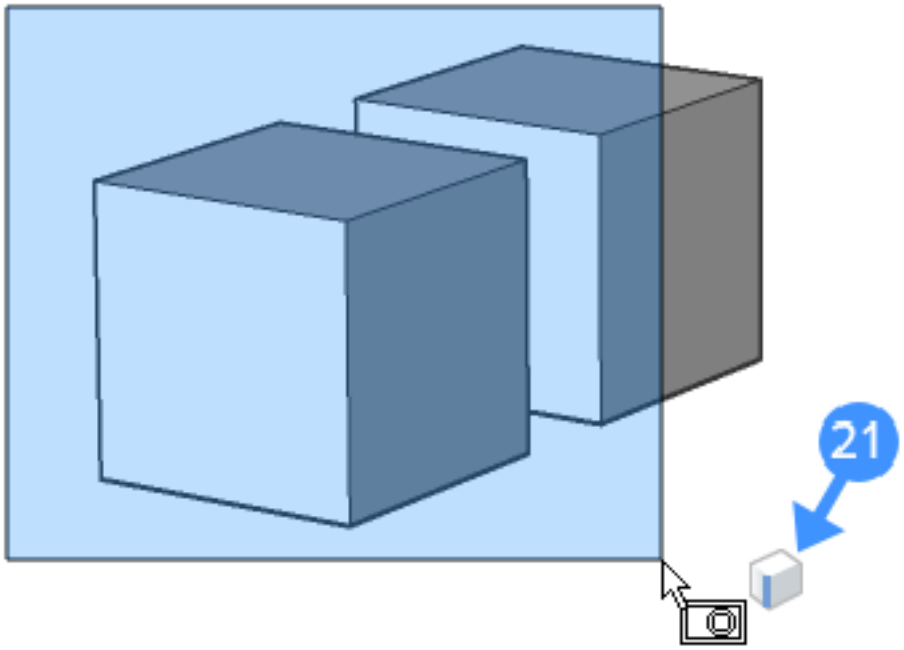


Figure 2.58

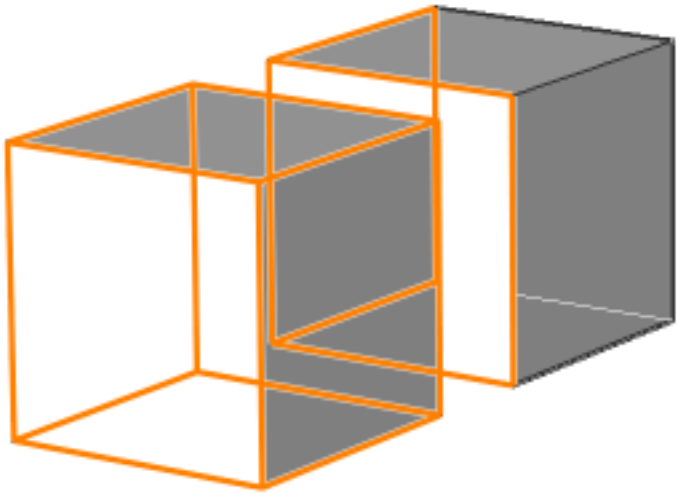


Figure 2.59

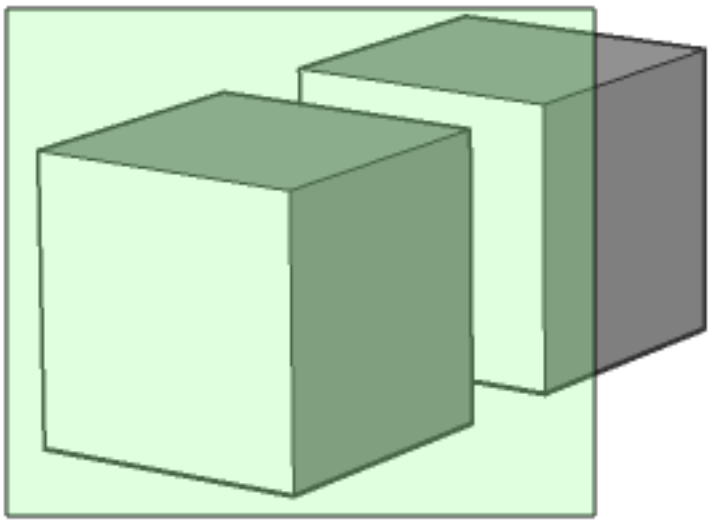
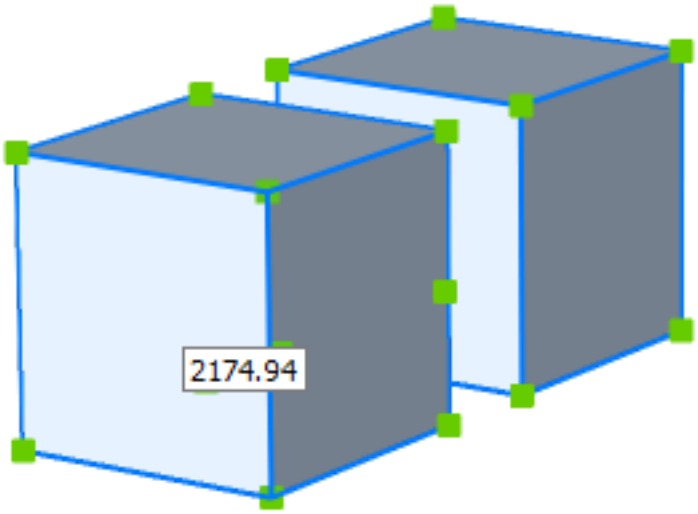


Figure 2.60



(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/GreenSelectionBoxFa

6. To select edges of 3D solids and surfaces, making use of the green selection box, press CTRL key twice during window selection. The 'Edges' icon appears next to the cursor (23). Any edges that are completely or partially inside the crossing window will be selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/GreenSelectionBoxEd

Figure 2.61

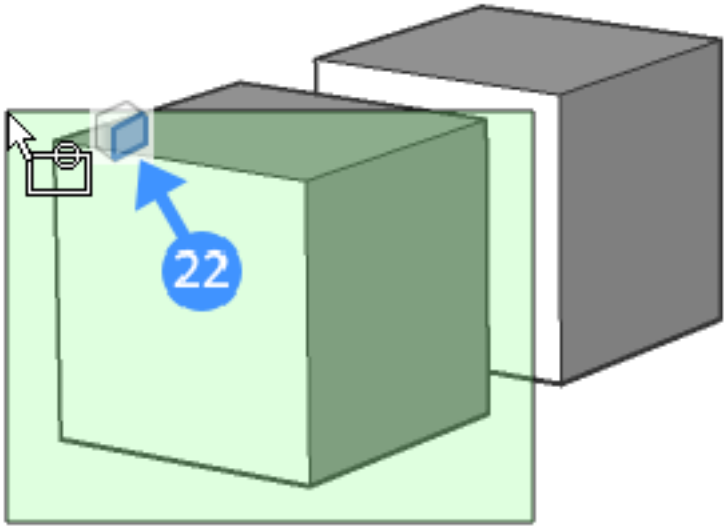


Figure 2.62

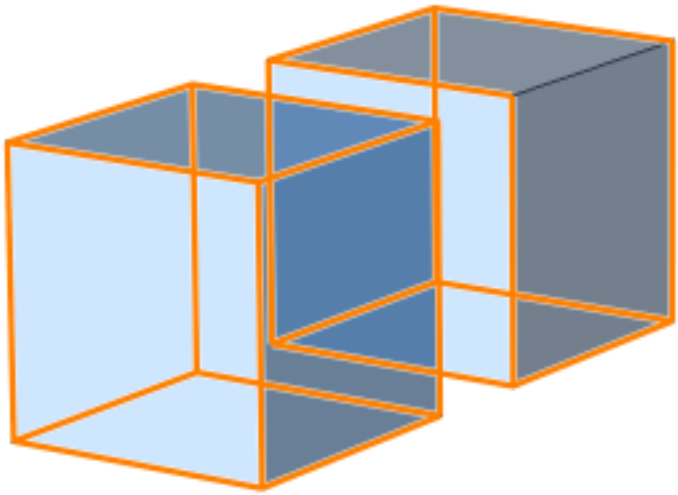


Figure 2.63

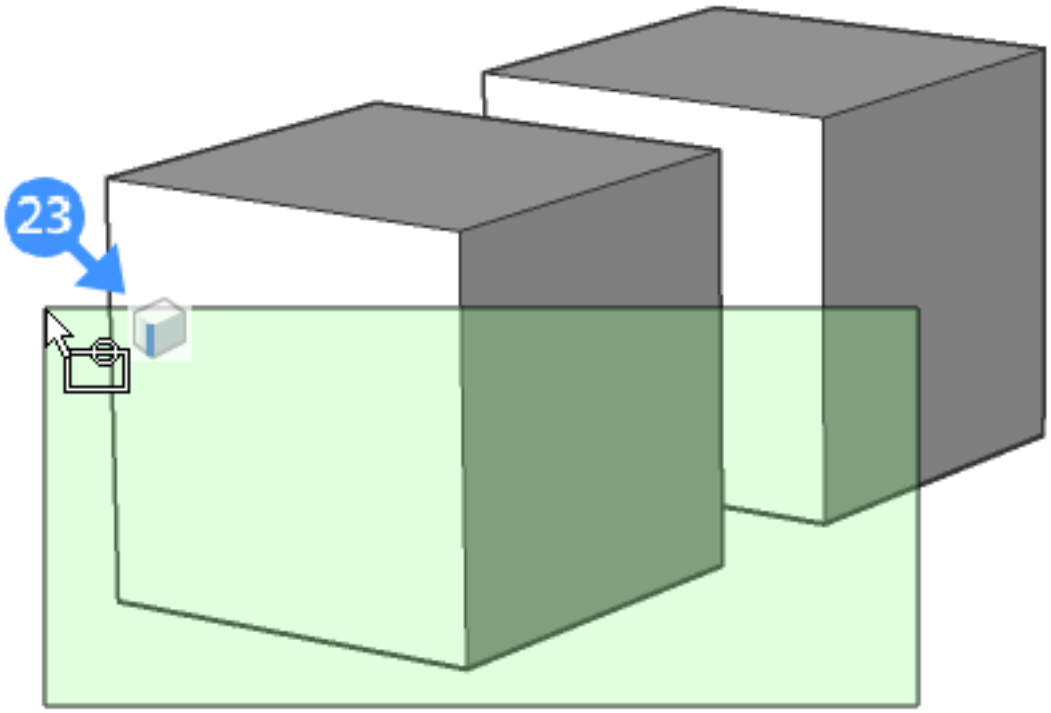
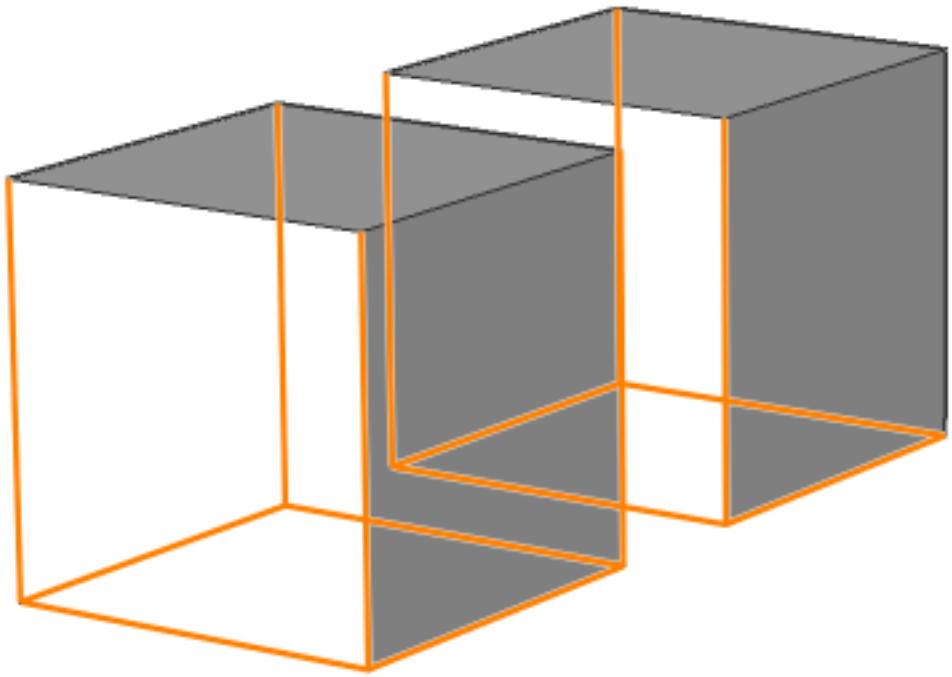


Figure 2.64



Procedure: Selecting/Highlighting Bim Sides and Ends

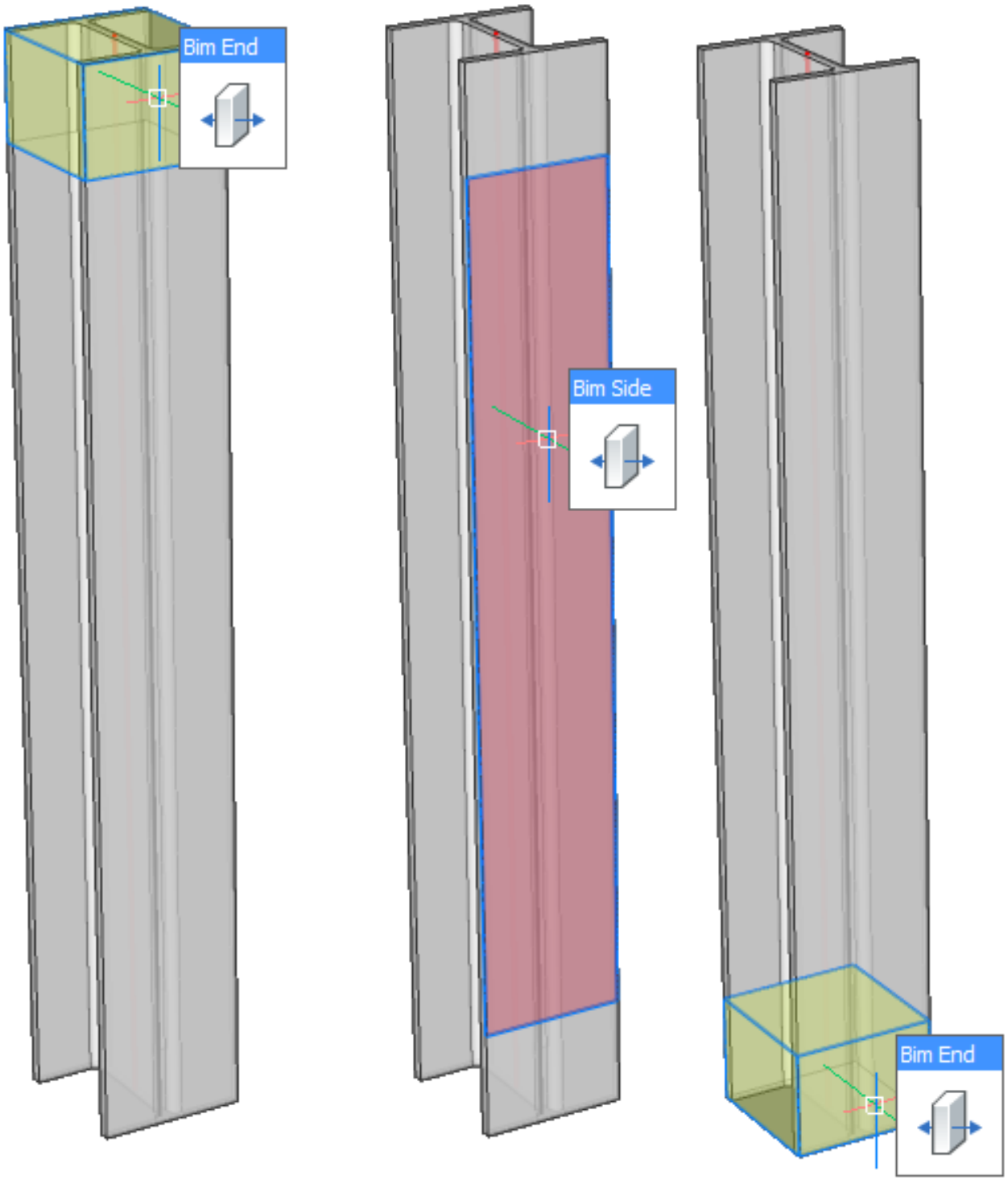
2. Click on **Enable selection of Sides and Ends** icon in the BIM Toolbar (see below image). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectionModesFacesBIMSidesandEnds.pn

3. Hover over the linear solid in the model space when the cursor is on the top or bottom side of linear solid, Bim End face displays in yellow whereas the side face of linear solid displays in red (see below image). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/LinearSolidBimSidesEnds.

Figure 2.65



Figure 2.66



2.3 Modeling Techniques

2.3.1 How to create and edit sections

https://www.youtube-nocookie.com/embed/N_4kG86Llqo

This video shows how Section Planes can be created and used to navigate through your model easily by clipping parts of the model. Back to Tutorial Overview

2.3.2 How to create a pitched roof

https://www.youtube-nocookie.com/embed/2po-JN5Prlc

This video shows how a pitched roof can be created starting from a polyline using the Extrude and Shell tools.
Back to Tutorial Overview

2.3.3 How to create walls

https://www.youtube-nocookie.com/embed/ihQyUxL75ps

This video explains how to use the Polysolid tool to create a wall. Back to Tutorial Overview

2.3.4 How to create wall layouts

https://www.youtube-nocookie.com/embed/xDuDHl36Pc0

This video explains how to use the Polysolid tool to create a series of walls, or to turn existing polylines into a wall layout. Back to Tutorial Overview

2.3.5 How to connect walls

https://www.youtube-nocookie.com/embed/AknK6xhkh7Q

This video explains how to use the different connection tools to easily connect existing walls to each other.
Back to Tutorial Overview

2.3.6 How to detect boundaries in BricsCAD BIM

https://www.youtube-nocookie.com/embed/m5M9QQiJces This tutorial explains how boundaries are automatically detected on flat surfaces, and how these boundaries can be used to create or edit geometry. Back to Tutorial Overview

2.3.7 How to extrude

https://www.youtube-nocookie.com/embed/xHyOhJRx9rM

This tutorial explains what the Extrude tool can be used for. Back to Tutorial Overview

2.3.8 How to hide and isolate objects

https://www.youtube-nocookie.com/embed/JDqhITMz5HI

This video shows how you can temporarily hide or isolate single objects or selection sets. Back to Tutorial Overview

2.3.9 How to offset

https://www.youtube-nocookie.com/embed/SVAGhfuaxHA

This video gives an example of how the Offset tool can be used in combination with other direct modeling tools to model quickly and efficiently. Back to Tutorial Overview

2.3.10 How to push or pull solids

https://www.youtube-nocookie.com/embed/-IJ3VHIGpOs

This video shows how to use Push/Pull to edit the face of a solid. Back to Tutorial Overview

2.3.11 How to select elements

https://www.youtube-nocookie.com/embed/SDYWIz0yYs4 This tutorial explains how to select entities and subentities. It explains the different selection modes and how to toggle between them, and how to use the CTRL and TAB keys to have full control over what you select. Back to Tutorial Overview

2.3.12 How to use Grids

https://www.youtube-nocookie.com/embed/Aas0bb0StV8 This tutorial explains how to create and edit a rectangular or radial grid. Back to Tutorial Overview

2.3.13 How to create Curtain Walls

https://www.youtube-nocookie.com/embed/uP0H3zDrLiQ

This video shows how to create a curtain wall starting from a planar or non-planar face. Back to Tutorial Overview

2.3.14 How to use linear solids

https://www.youtube-nocookie.com/embed/Nt6fyZ_xXMk

This tutorial explains how to create and use linear solids for structural modeling. Back to Tutorial Overview

2.3.15 How to add eccentricities to linear solids

https://www.youtube-nocookie.com/embed/rOYgRPJrlH0

This video shows how to give linear solids and eccentricity with respect to their axis. Back to Tutorial Overview

2.3.16 How to connect linear solids

https://www.youtube-nocookie.com/embed/-gW8L7NDjsE

This video shows how to use the Structural Connect tool to solve nodes of structural steel elements. Back to Tutorial Overview

2.3.17 How to create a Site

https://www.youtube-nocookie.com/embed/fNHXnGfty1s

This video shows how to create a TIN surface or toposurface using three different methods: using existing points, manually adding points, or importing a point file. Back to Tutorial Overview

2.3.18 How to edit a Site

https://www.youtube-nocookie.com/embed/FDl6L2bNqgI This video shows how to add and remove points, breaklines and boundaries to a toposurface. Back to Tutorial Overview

2.3.19 How to create a Grading

https://www.youtube-nocookie.com/embed/x9ILmYT1NPU This video shows how to create a grading starting from a polyline and an existing TIN surface. Back to Tutorial Overview

2.3.20 How to use the BIM Insert tool

https://www.youtube-nocookie.com/embed/1qyoe16tNko

This video shows how to use the BIM Insert tool to insert windows and doors into your model. It explains how to use the dynamic dimensions for accurate placement of the components. Back to Tutorial Overview

2.3.21 How to automatically Parametrize objects

https://www.youtube-nocookie.com/embed/e5GjX33G-k8

This video shows how to parametrize an object automatically, and how to edit these constraints and parameter expressions. Back to Tutorial Overview

2.3.22 How to use Layers

https://www.youtube-nocookie.com/embed/flEr6fZfHAs

This video shows how CAD layers can be used in BricsCAD BIM to organize your model and temporarily hide groups of objects. Back to Tutorial Overview

2.3.23 How to use Quickdraw

https://www.youtube-nocookie.com/embed/9g2bhcxPFbo

This video explains the function of the Quickdraw tool. It shows how the magnetic cursor works, how to create a new room and how to edit existing rooms. Back to Tutorial Overview

2.3.24 How to use selection windows

https://www.youtube-nocookie.com/embed/IO5W4T0ql_c

This video explains the difference between a green and a blue selection window, and how to use the CTRL key to window select solids, faces or edges. Back to Tutorial Overview

2.3.25 How to use the Components Panel

https://www.youtube-nocookie.com/embed/nomfQh5dB5o This video shows how to use the Components Panel to drag and drop components into your model, and how to create new components from scratch. Back to Tutorial Overview

2.3.26 How to use the Window Creator tool

https://www.youtube-nocookie.com/embed/M7WEU7dUO-E

This tutorial shows how to turn a simple polyline or boundary into a fully parametric window component using the Window Creator Tool. Back to Tutorial Overview

2.3.27 How to use BIM Drag

https://www.youtube-nocookie.com/embed/cFQP46P4IQQ

This video shows how to use the Drag tool to move elements. It shows how to toggle off the connectivity with adjoining entities.

Back to Tutorial Overview

2.3.28 How to use the MEP modeling tools

https://www.youtube-nocookie.com/embed/S4OFhcotTeY

This video shows how to use linear solids to create MEP Flow segments, and how to manipulate these segments using different tools. Back to Tutorial Overview

2.4 Modeling Techniques

2.4.1 BIM Insert

Command: BimInsert (6.7.14) **Overview:** In this lesson, you learn how to insert and edit windows and doors in a solid. **Lesson Objectives:** After completing this lesson, you will be able to:

- Insert a window or door as a block reference into your 3D model.
- Edit the parameters of a window or door.
- Move the inserted door or window in the face of a 3D solid.
- Place copies of an insert in the same 3D solid.
- Replace an existing insert with another.

Tutorial: How to use the BIM Insert tool Click here (2.3.20) to watch

About

In BricsCAD, the **BIM Insert** tool allows you to insert windows, doors and other components in any face of a 3D solid and is typically used to insert predefined windows and doors.  Menubar: **BIM > BIM Insert** Toolbar: **BIM > BIM Insert** Quad: **Model > BIM Insert** Keyboard: **BIMINSERT** Ribbon: **Component > Insert**

Controlling the position of the window or door To control the position of the window in the face of the solid (wall), you can use the following drawing aids:

- Dynamic Dimensions (1.15.9): During placement of a window or door on a 3D Solid face, dynamic dimensions are created, starting from the middle of each edge of the bounding rectangle of the opening to the nearest parallel edge on the face of the 3D Solid. Make sure Dynamic Dimensions are active by checking theDYNfield in the status bar andDynamic UCSis active by checking theDUCSfield in the status bar. TheDynamic UCSaligns the window to the face of the solid. The edge of the face by which the cursor enters the face defines the orientation of the X-axis. Hit the Shift key to temporarily lock the UCS, which allows to use reference points outside the face of the solid. Hit the shift key again to unlock.
- Temporary Tracking points (1.15.15): Click the mouse wheel or press theTemporary Tracking Pointstool button () on theEntity Snapstoolbar to start specifying temporary points.
- **OSNAPZ:** If this system variable is ON, the Z-value of any entity snapping point is replaced by the current value of the ELEVATION system variable. If ELEVATION=0, entity snapping points are forced to lie in the face of the solid. OSNAPZ is toggled by theIgnore entity snap elevationtool button () on theEntity Snapstoolbar. The state of the tool button (pressed or not) indicates the current value of OSNAPZ.
- **Entity snap to negative Z-values:** Press theEntity snap to negative-ztool button () on theEntity Snapstoolbar to enable entity snaps on points that lie behind the selected face. If OSNAPZ is ON, the point is projected onto the selected face.

Procedure: Inserting windows or doors

Note: Make sure Dynamic UCS (DUCS) and Dynamic Dimensions (DYN) are active.

1. Select **BIM Insert** from the No Selection Quad.

The file explorer opens (1). By default it should take you to the library of predefined doors and windows. If it does not, this library can be found on the following path: C:\Users\<User>\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\Sup
Note: This is a hidden folder by default, so you may have to show hidden folders first. It is of course also possible to insert windows and doors that you define yourself. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM

2. Select a window or door you want to insert in your drawing.

Figure 2.67

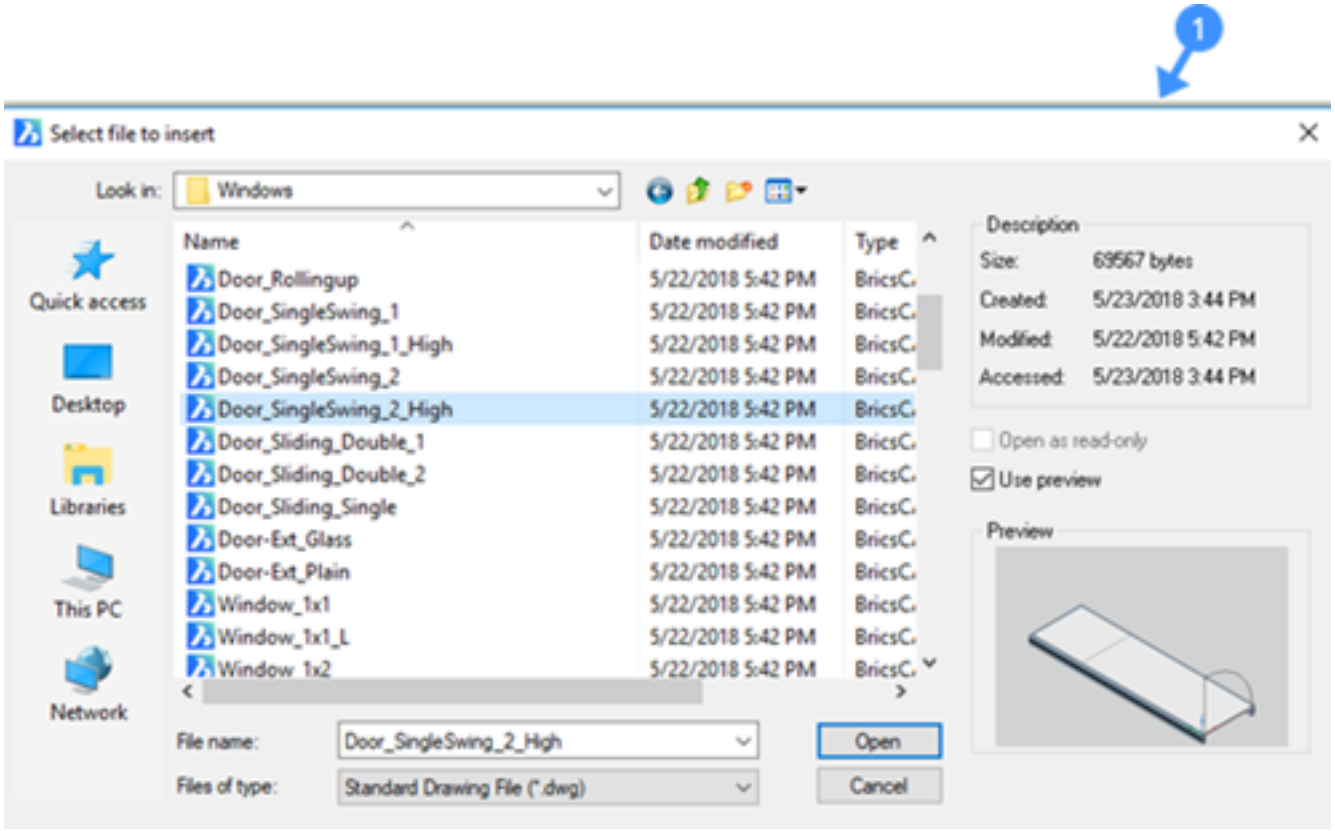


Figure 2.68

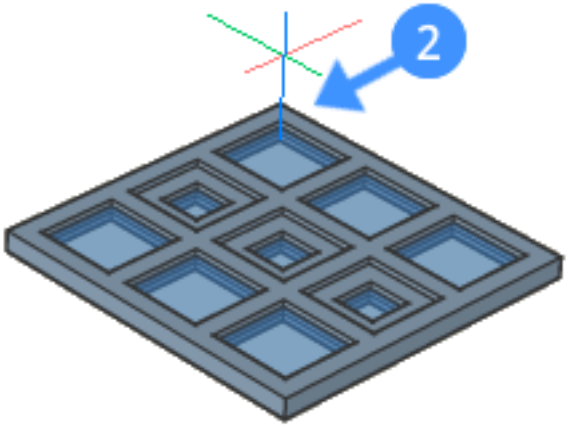


Figure 2.69

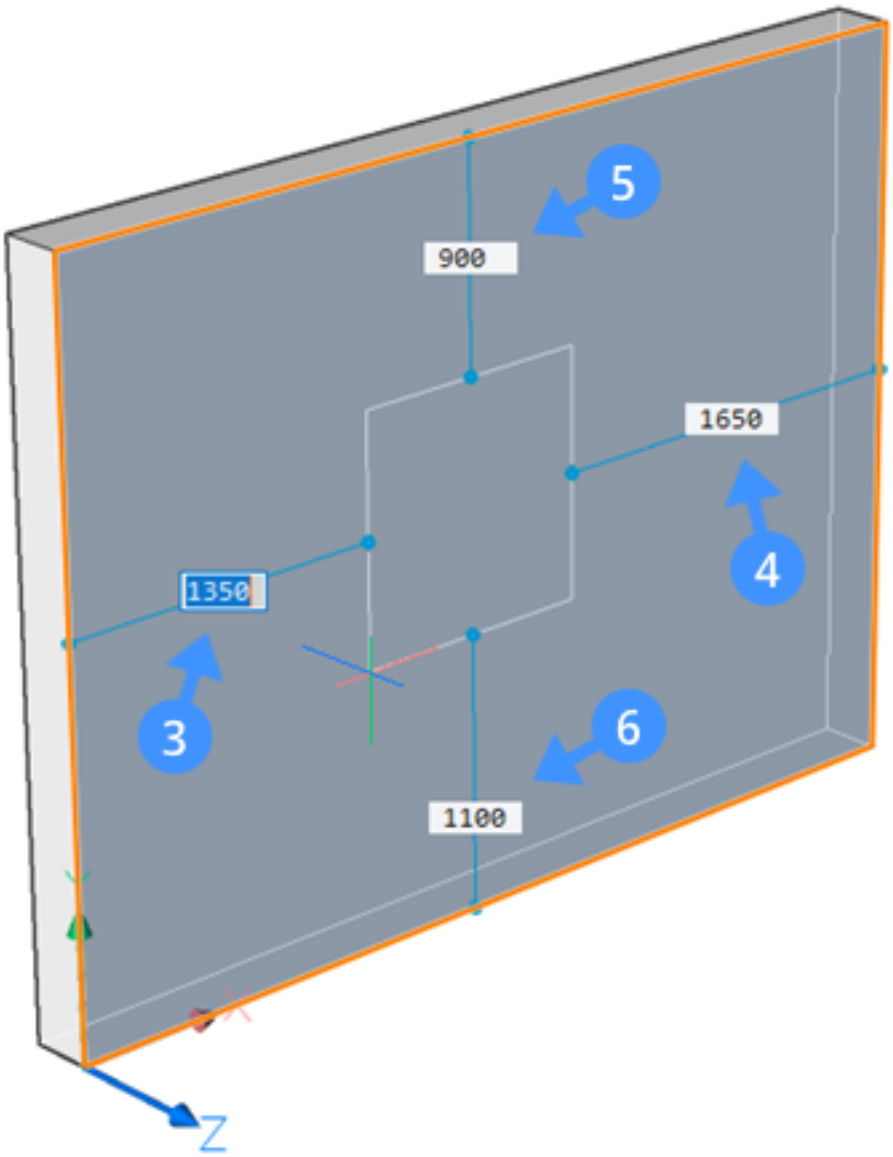


Figure 2.70

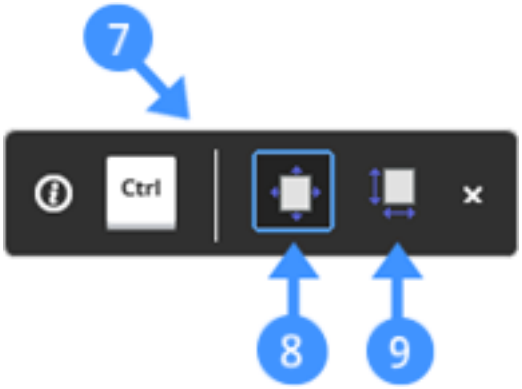
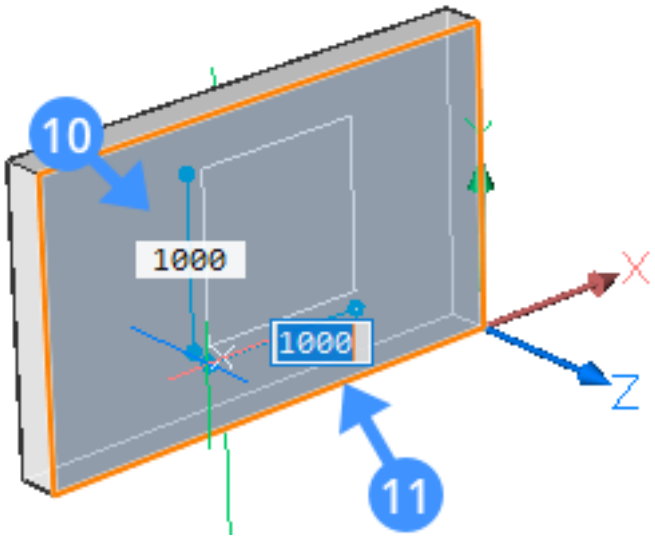


Figure 2.71



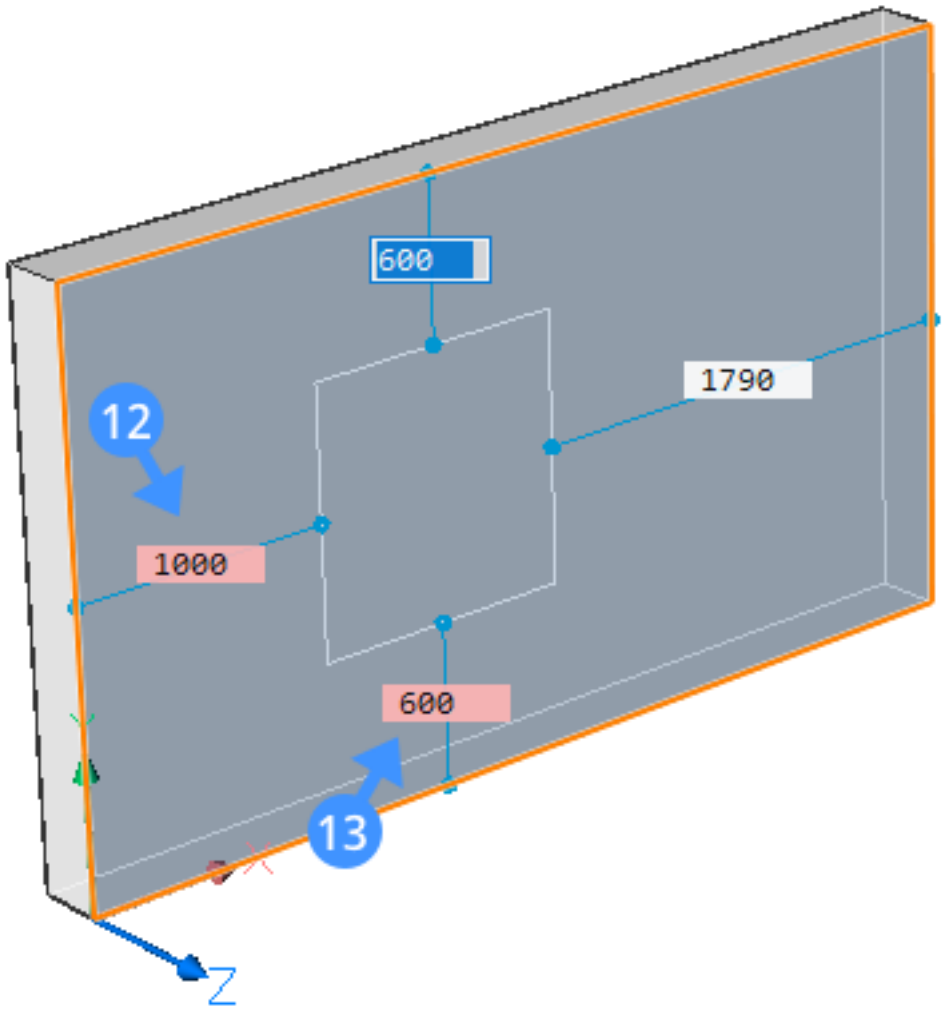
The selected insert is attached to the cursor (2). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/I
When you move over a face of the solid, the insert aligns with the face under the cursor. The distances from the start (3), end (4), top (5) and bottom (6) face of the wall to the placement rectangle of the insert display dynamically. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMInsertPosition.png})
If the HKA is ON, the BIM Insert HotKey Assistant Widget (7) displays at the bottom of the screen to show you the available options. Use 'pick position' (8) option to adjust the insertion point of the wall or door. Use the 'update size' option (9) by hitting the CTRL key to specify the width and height of the insert. The Width (10) and Height (11) field will indicate the current dimensions of the insert. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMInsertSize.png})

3. Do one of the following to place the selected window or door using the dynamic dimension fields:

- Hit the TAB key to select one of the dynamic dimension fields, then type a value and hit the TAB key to lock the dimension. The locked dimension turns red (12).
- Hit the TAB key to select one of the other dynamic dimension fields, then type a value and hit the TAB key to lock the dimension. The locked dimension turns red (13).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMInsertLockedDimensio

Figure 2.72



Procedure: Editing the parameters of an insert

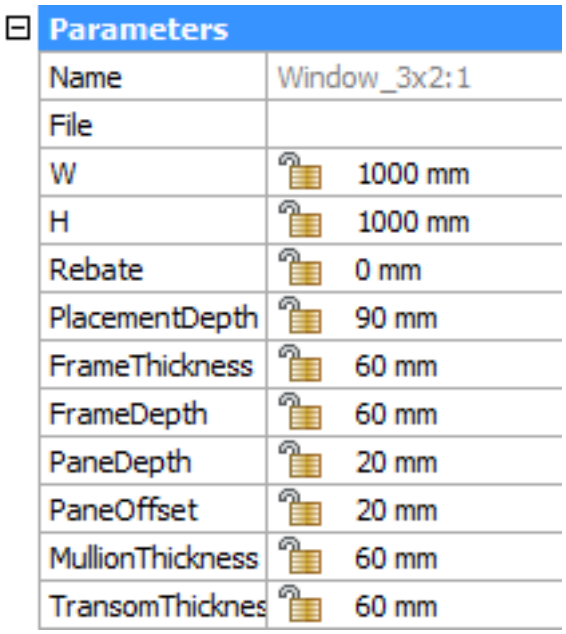
If parameters are defined in the window or door source drawing, these parameters can be used to edit each detail of a window/door separately.

1. Select any door or window in the drawing.

The current values of the various parameters display in the parameters section of the **Properties Panel**. In the following illustration, window parameters are shown in the properties panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

2. Select a parameter you want to change, then type a value in the settings field.

Figure 2.73



Moving an Insert

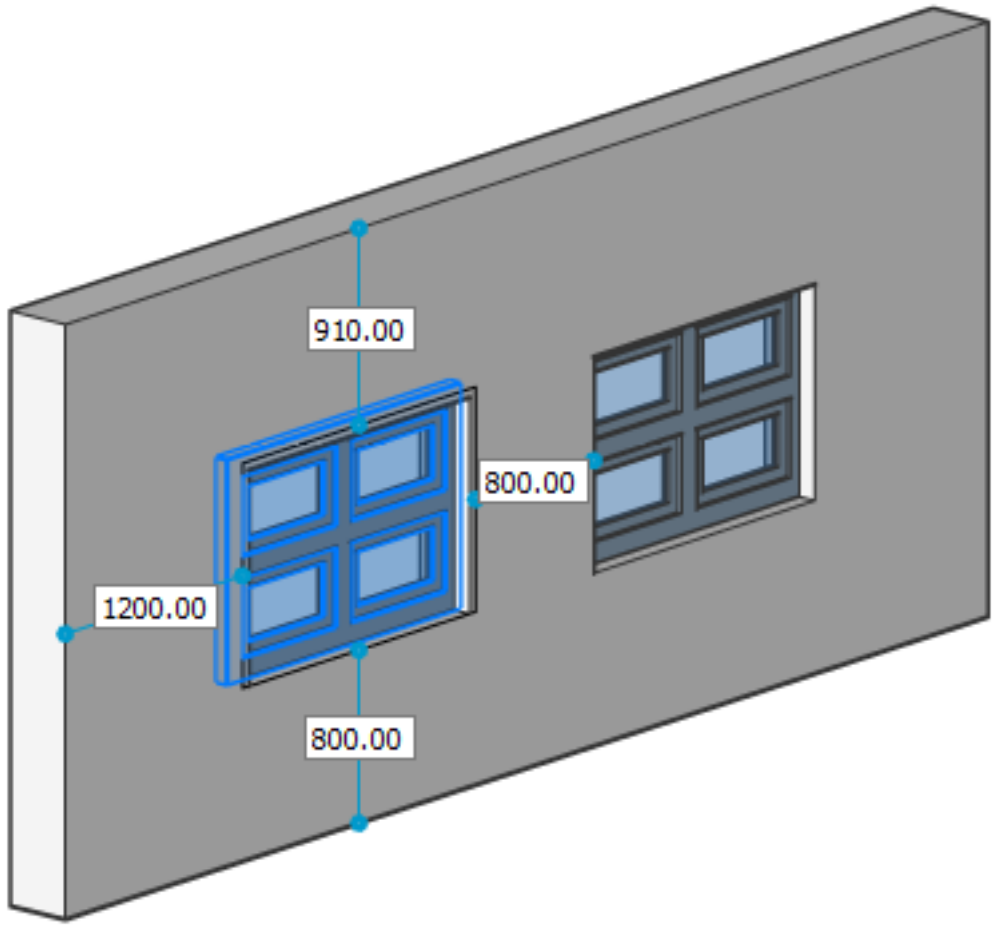
In BricsCAD, there are two methods to move an insert in the face of a solid. The first method uses distance fields after selecting the insert. The second method can be done with the **Push/Pull**tool which enables to move an insert vertically or horizontally when applied to an inner face of the opening made by the insert.

Procedure: Using distance fields

1. Select the insert in the model. Current distances from the insert to the nearest parallel faces of the parent solid display: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/WindowRepo

2. Double click a distance field, then type a value and press Enter.

Figure 2.74

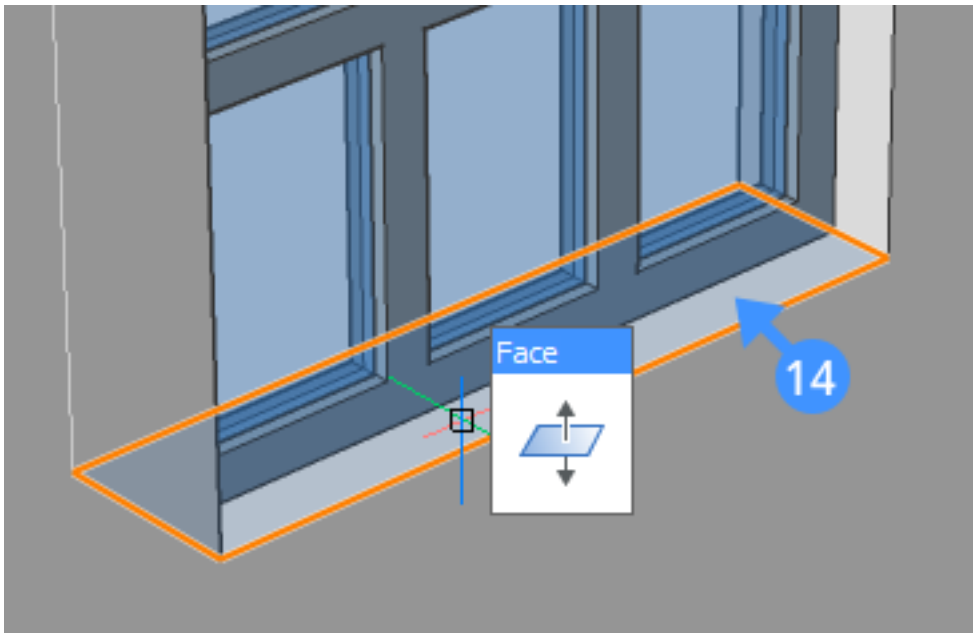


Procedure: Using Push/Pull to move an insert

1. Hover the cursor over the top or the bottom inner face of the opening (14) made by the insert to move the insert vertically. Or to move the insert horizontally, hover the cursor over the left or the right inner face of the opening (15) made by the insert.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/InsertBottomFace.png})
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/InsertRightFace.png})

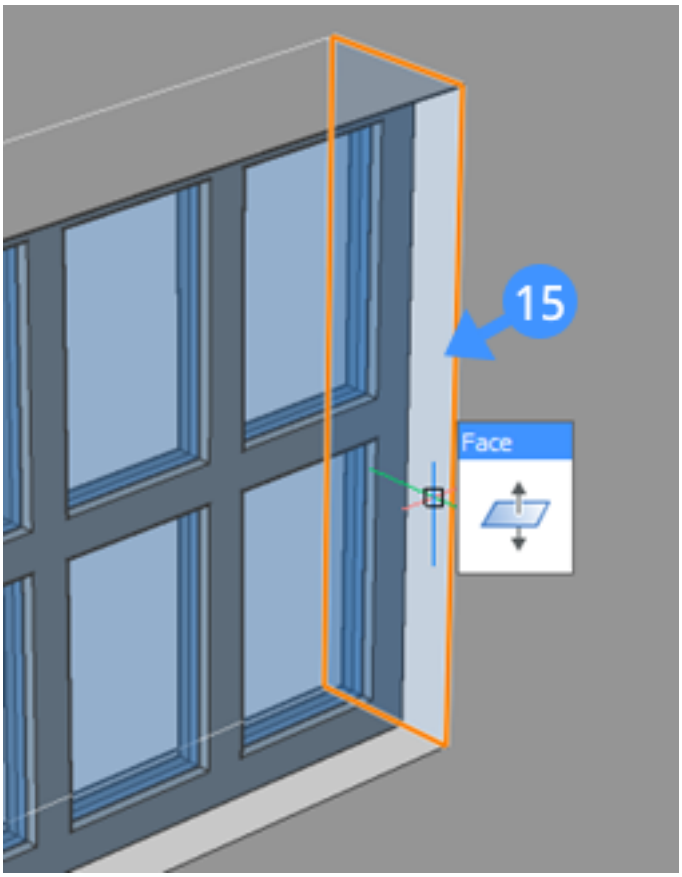
Figure 2.75



2. When the face highlights, select DmPushPull (6.12.20) from the Quad. The insert will move vertically if the top or bottom inner face is highlighted. The insert will move horizontally if the left or right inner face is highlighted.
3. Do one of the following to complete the moving insert process.

- Specify a point.
- Type a distance in the dynamic dimension field.

Figure 2.76



Copying Inserts

Inserting a copy of an insert

1. Hover over the insert. When the insert highlights, chooseBimInsertin theBIMcommand group in the Quad. You are prompted: Select insertion point or [Edit inserted entity] :
2. Proceed with theInserting Windows and Doorsprocedure from step 2 on.

Copying an insert in a solid The **Copy** tool can be used to place copies of an insert in the same solid. Using the repeat option on the copy tool you can create a series of copies at the same interval. **Note:** The movement vector for the copy should be parallel to the face of the wall. If the orientation of the wall, in which the source window or door is placed, is not parallel to either the X- or Y axis of the current UCS, use the**Face**option of the **UCS**command to align the UCS.

Procedure: Copying an insert on the wall face

Note: Make sure that Dynamic UCS (DUCS) and Dynamic Dimensions (DYN) are active.

1. Select an insert using the BIM Insert tool. **You are prompted:** Select insertion point:
2. Place the insert on the wall face. The insert should align with the wall face.
3. Next, hover over the insert to create a copy of it. When the insert highlights, choose **Copy**in the Modify command group in the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniqu
You are prompted: Enter base point [Displacement/ mode] <Displacement>:
4. Pick a point in the drawing area. **You are prompted:** Enter second point <Use base point as displacement>.
5. Type a displacement distance from the coordinates of the base point. **Note:** Displacement specifies the distance to position the copy. You can use the dynamic dimension fields (16,17) to specify the displacement. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/CopyInsertDisplacem
You are prompted: Enter second point [Undo/Repeat/Exit]
6. Optionally, use the **Repeat** option to create multiple copies of the insert keeping the displacement distance of each copy same.
7. Press Enter to accept the copying.

Figure 2.77

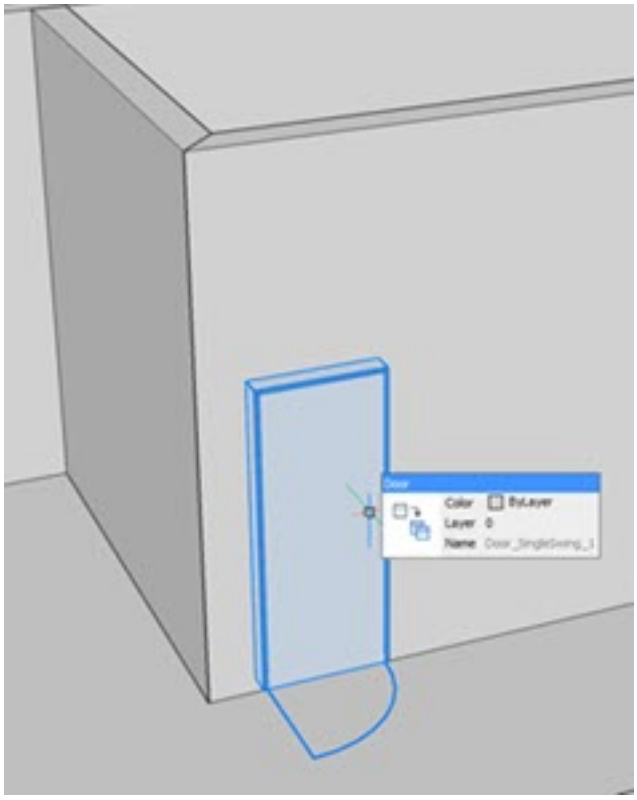
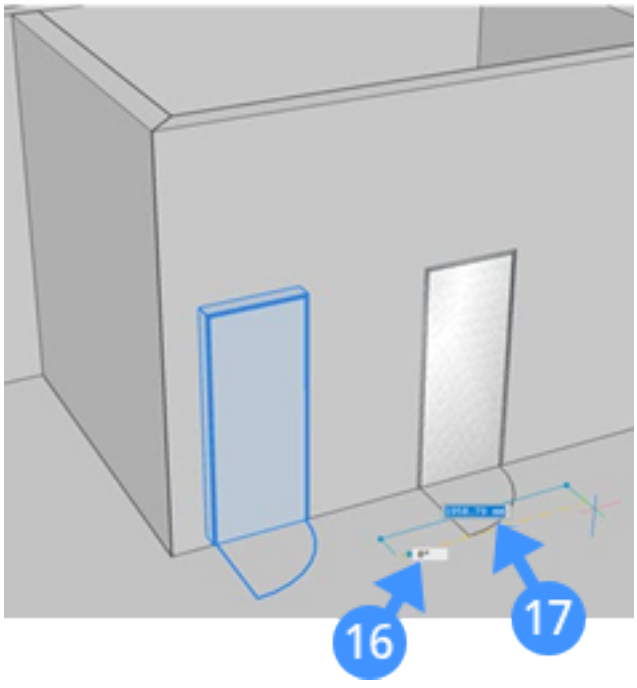


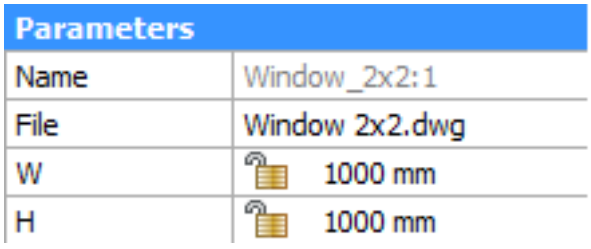
Figure 2.78



Replacing an insert

1. Select the insert. The properties of the selected insert display in theProperties Bar (1.12.4).
2. Select the File property in the Parameters group. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/
3. Click theBrowsebutton (...). AFiledialog displays.
4. Select the replacement file for the insert, then double click or click theOpenbutton on theFiledialog. The insert is replaced.

Figure 2.79



2.4.2 BIM Linear Solid

Command: BimLinearSolid (6.7.15) **Overview:** In this lesson, you learn how to use the Linear Solid tool to create structural members and flow segments. **Lesson Objectives** After completing this lesson, you will be able to:

- Use different structural member profiles and piping profiles in your structural and mechanical modeling.
- Apply and replace profiles of linear solids.
- Add eccentricity to linear solids.
- Recalculate the axis of linear building elements.
- Make structural connections and flow fittings.

About BIM Linear Solid

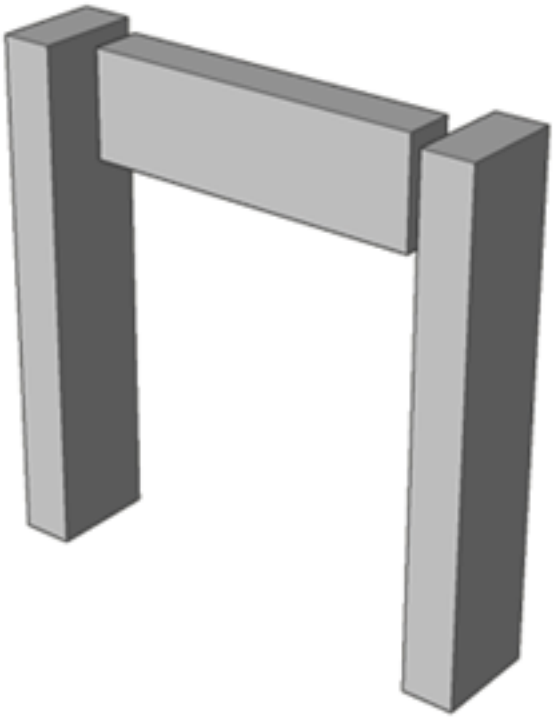
The Linear Solid tool allows you to create structural members or flow segments. The tool can be made use of creating a variety of structural member profiles or MEP elements.  Menubar: **BIM tab > Linear Solid** Toolbar: **BIM > Linear Solid** Quad: **Model Tab > Linear Solid** Keyboard: **bimlinearsolid** Ribbon: **Model tab > Create 3D > Linear Solid**

Creating a single linear solid

Tutorial: How to use linear solids Click here (2.3.14) to watch Before starting the process, make sure that Quad is active. Next, follow these steps:

1. Hover over the Model tab in the Quad and select Linear Solid tool. **You are prompted:** Set start point or [Follow/Rectangular/Circular/Dialog/select in Model/adJust profile] <Last point>:
2. Select a start point. **You are prompted:** Set next point or [Angle/Length/Undo/Quarter turn/Rotate/choose other Profile/adJust profile]:
3. Move the mouse cursor to draw the beam or column.
4. Click to specify the endpoint, or enter a value for the distance using the dynamic dimension field.
5. Press Enter key to accept it. The following illustration shows beam and two columns with the default rectangular profile. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/Beam2Col

Figure 2.80



About: BIM Profiles

BIM Profiles includes steel structural profiles in the library with the installation. These structural profiles can be used to create and edit linear solids. Currently, American, Australian, British, European and Russian structural steel standards are available in the library database.

Tutorial: How to use the Profile Library for linear solids Click here (2.5.9) to watch **BIM Profiles toolbar Note:** If the BIM Profiles toolbar is not displayed on the right-hand side of your screen, right-click on the **Tool Panel**at the right-hand side of your screen, then choose BIM Profiles in the context menu. **Keyboard:** BIMPROFILES The following illustration shows the **BIM Profiles Panel**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Mo
The drop-down box at the top (1) acts as a filter. There are different domains on which you can filter: Generic, Structural Concrete, Structural Steel, HVAC, Piping and Electrical. **Note:** If you don't have any profiles in a certain domain, you will not be able to filter on it. The second drop-down boxes (2) allow you to filter the profiles by the country standards. The search box (3) allows you to filter by name. The menu (4) on the right-hand side allows you to access the BIM Profiles dialog box. Clicking the menu icondisplays a context menu with a couple of options. The following illustration shows the BIM Profiles context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/
Select "**Open profile dialog (5)"** in the menu to open the Profiles Dialog box. The following illustration shows the **Profiles** dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMPr

Filters (6)	Filters the profiles list by Type, Standard, Shape, and Name.
Tools (7)	Use a particular function, e.g. create a new profile
Custom fields (8)	Set the type, standard, name, and size of a profile
Preselect shape (9)	Select a shape
Pick profile in model (10)	Select a closed 2D entity or the boundary entities of an enclosed area
Profile properties (11)	Shows the properties of the selected profile

Figure 2.81

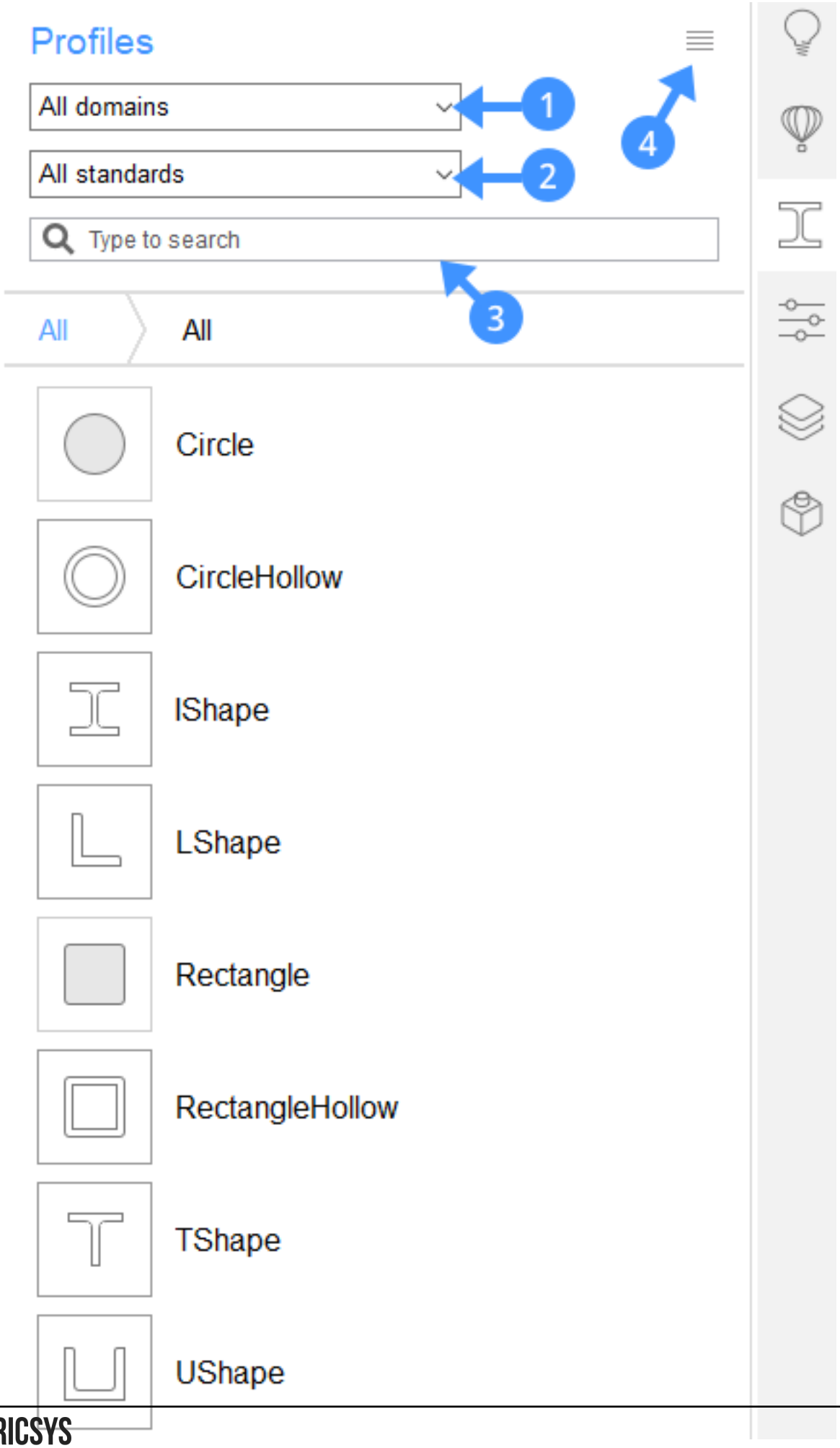

Profiles
All domains
All standards
Type to search
1
2
3
4
All
All
Circle
CircleHollow
IShape
LShape
Rectangle
RectangleHollow
TShape
UShape

Figure 2.82

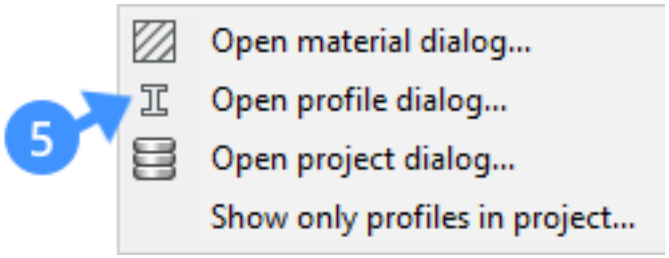
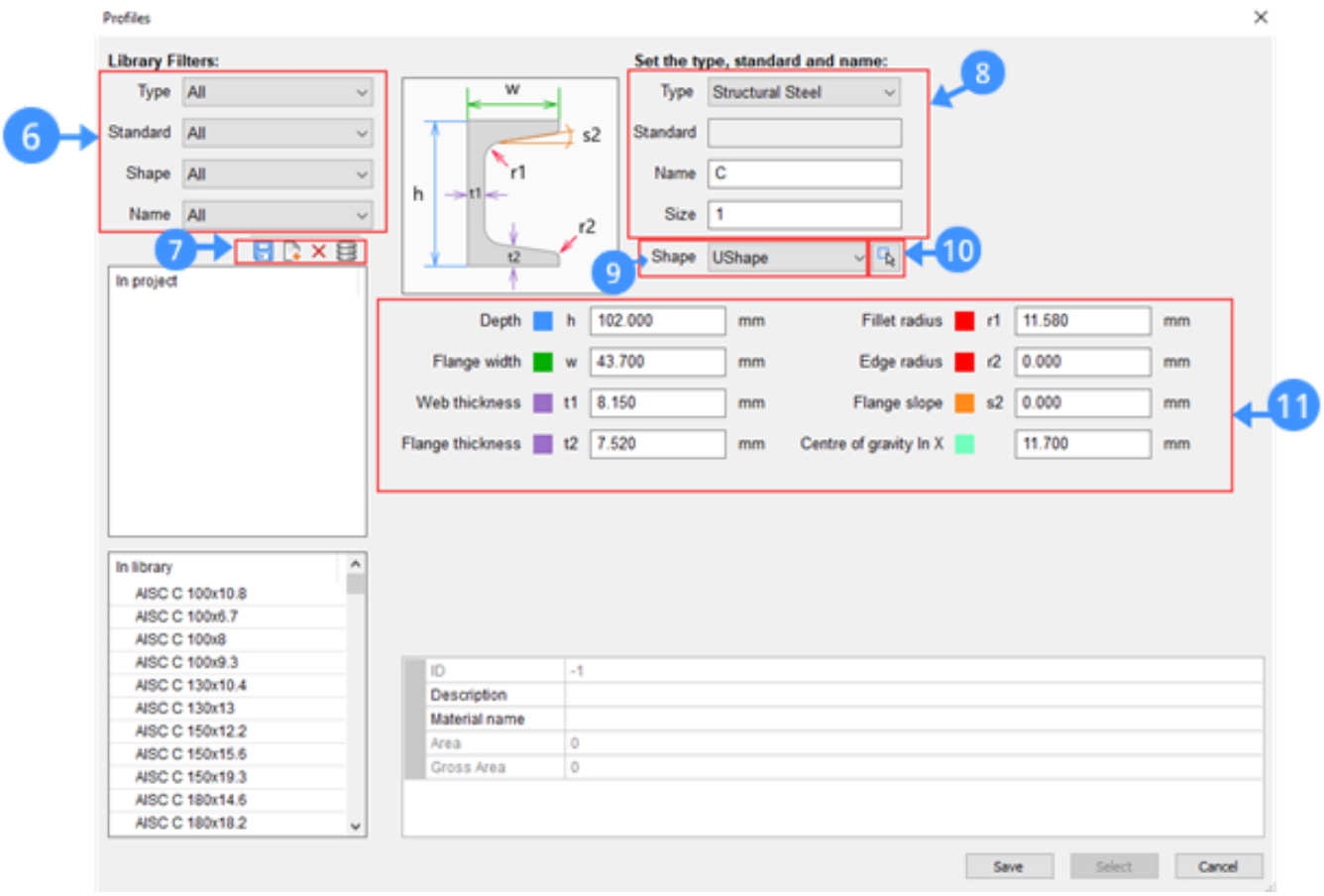


Figure 2.83



About: BIM Apply Profile

The Apply Profile tool allows you to change the profile of linear entities and linear solids. Linear solids have a constant cross-section over a linear extrusion path. The tool recognizes clippings and openings. Using the Apply Profile tool, you can create columns, railings, beams, pipes, ducts and more.  Menubar: **BIM tab> Apply Profile** Quad: **Model tab > Apply Profile** Keyboard: **BIMAPPLYPROFILE** Ribbon: **Data tab > Attach > Attach Profile**

Applying profile to linear solids

1. Launch the Apply Profile tool.
2. Select a linear entity or linear solid in the drawing area. **You are prompted:** Select profile [in Dialog] <in Dialog>:
3. Type 'D' in the command line or click 'in Dialog' option on the prompt menu to open the **Profiles** dialog box. Use the Filters section to sort the profile by type, standard, shape or name. Choose one of the available profiles in the library. Or pick a profile in the model to create a custom shape profile. Click the**Pick profile in Model**button in the dialog box and select an existing profile in the drawing area.
4. Press Enter to accept changes.

About: BIM Add Eccentricity

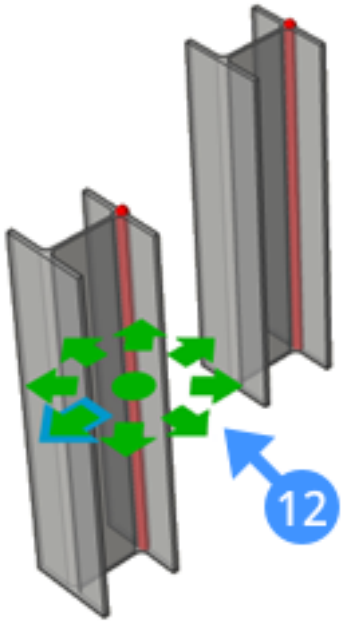
BIM Add Eccentricity allows you to control the relative position of the axis of a linear solid.

Tutorial: How to add eccentricities to linear solids Click here (2.3.15) to watch  Menubar: **BIM tab >Add Eccentricity** Quad: **Model tab > Add Eccentricity** Keyboard: **BIMADDECCENTRICITY** Ribbon: **Data tab > Attach > Add Eccentricity**

Add eccentricity to linear solids

1. Select an existing linear solid in the drawing area.
2. Initiate the Add Eccentricity tool. The eccentricity widget appears with the possible locations.
3. To set an eccentricity, click one of the nine locations from the widget, then right click to confirm.
4. Optionally, type a location in the command line. The options of the locations are: **TR:** Top Right, **TM**: Top Middle, **TL**: Top Left, **MR**: Middle Right, **C**: Center (resets the axes of the selection set to the default location), **ML**: Middle Left, **BR**: Bottom Right, **BM**: Bottom Middle, **BL**: Bottom left
5. Press Enter to accept changes. The following illustration shows the **Eccentricity** widget (12) on the linear solid profile. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/AddEccentric

Figure 2.84



About: BIM Recalculate Axis

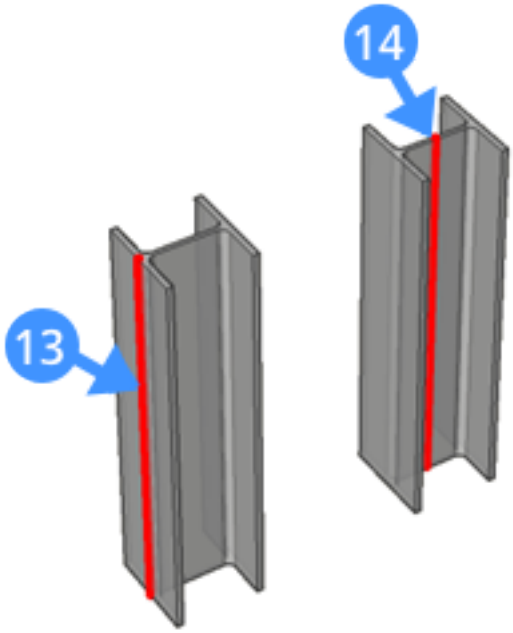
The Recalculate Axis tool recalculates the axis of linear building elements and repositions it back to the center line of a linear element.  Menubar: **BIM tab >Recalculate Axis** Quad: **Model tab> Recalculate Axis** Keyboard: **BIMCALCULATEAXIS Ribbon: Data tab > Attach > Recalculate Axis**

Recalculate the axis of linear building elements

1. Select any linear building element in your model
2. Initiate the Recalculate Axis tool
3. The axis of the element is moved to the default position.

The following illustration shows the result of using BIM recalculate axis on the linear solid. The axis (13) is moved to the center line (14) of the linear element. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelin

Figure 2.85



About: Structural Connect Tool

The BIM Connect tool can be used to connect linear solids. It makes a connection between the selected linear solids withtheoptionofsmooth or planar cuts.

Tutorial: How to connect linear solids Click here (2.3.16) to watch  Menubar: **BIM > BimConnect > StructuralConnect** Quad: **BIM tab > StructuralConnect** Keyboard: **BIMSTRUCTURALCONNECT** Ribbon: **Model tab > Modify 3D > Connect > Structural Connect**

Making a structural connection to linear solids

1. Select the linear solids that need to be connected
2. Initiate the Connect tool
3. The Hotkey Assistant appears at the bottom of the screen to remind you of different options. The following illustration shows the BimConnect tips widget. The default connection type is indicated by a blue frame. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/HKAStructuralConnect.png

Smooth cut (15)	One solid is fully extended to connect to all the other solid's faces
Planar cut (16)	Solids are cut with straight planar cuts

Press CTRL key to cycle through the connection options.

4. Select one of the connection options and press Enter to accept the connection.

The following illustration shows the smooth cut (17) and planar cut (18) on an existing structural model. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/StructuralConnectSmoothCut.pn
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/StructuralConnectPlanarC

Figure 2.86



Figure 2.87

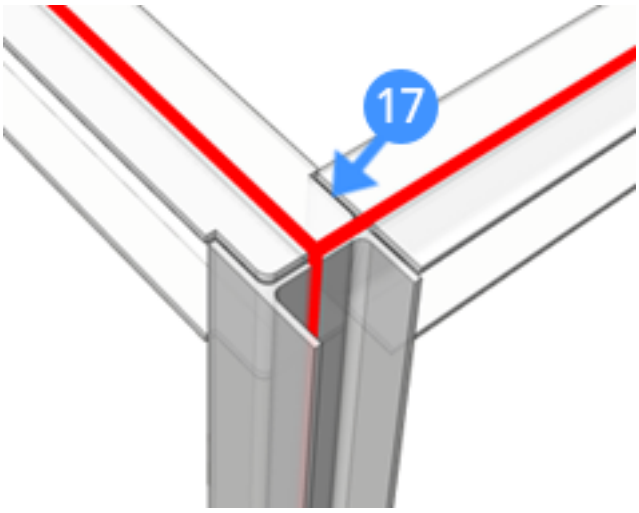
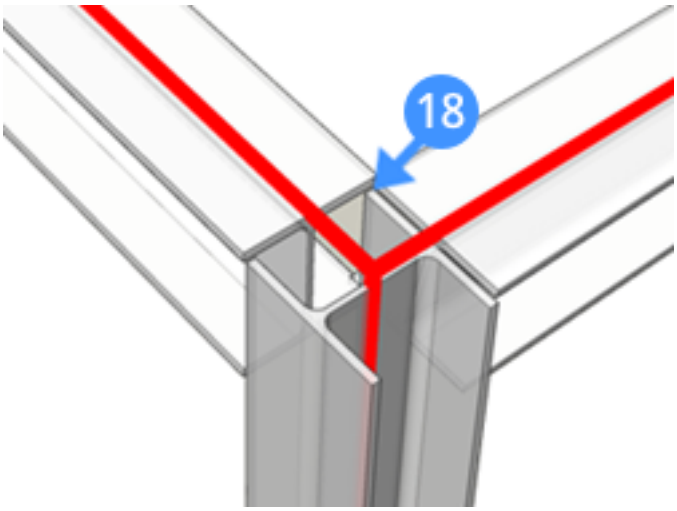


Figure 2.88



Related Commands

BimApplyProfile (6.7.2) - Applies a profile to linear entities and linear solids.

2.4.3 BIM Window Create

Overview: In this lesson, you learn how to create a fully parametric window based on a closed contour in your BIM model. When you have a closed 2D entity or boundary on the face of solid (wall), the window creator tool enables you to use these closed entities to make your window. Using the window creator tool, you can select any window styles in the dialog box, and change the properties of this created window in a BIM model. **Lesson Objectives** After completing this lesson, you will be able to:

- Create a parametric window from a profile on a wall.
- Change the parameters of the created window.

Tutorial: How to use the Window Creator tool Click here (2.3.26) to watch

About

In BricsCAD, the **Window Creator** tool allows you to turn a different set of design ideas into a window. The tool can be used on windows of all shapes and sizes, giving you the opportunity to utilize a significant amount of window styles in your BIM project.  Menubar: **BIM > Create Window** Toolbar: **BIM > Create Window** Quad: **Model > Create Window** Keyboard: **BIMWINDOWCREATE**

Procedure: Creating a parametric window using a boundary

1. Draw a polygon (1) on the face of a solid (wall).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/DrawPolygon.png})
Based on this base profile, the window opening will be created.

2. Make sure Boundary Detection is enabled. Hover over the inside of the polygon and select the Create Window tool from the Quad.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/CreateWindowBoundary.pn
A file dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/W

3. Select a window style in the dialog box

The window is created on the wall with the selected style. The following illustration shows the triple panel vertical window style (2) which was selected to create a window on the face of the wall. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/TriplePanelVerticalWindow.png})

4. Optionally, you can refine the window design by changing the default values of the parameters. To do this, select the window and change the desired properties in the properties panel.

In the following illustration, the default parameters of the triple panel vertical window are shown. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/PropertiesPanelWindowParameters.png})
To change one of the default parameters of this window, type a new value in the property field.

5. Press enter to accept the new value of the parameter. The window should be updated automatically.

Figure 2.89

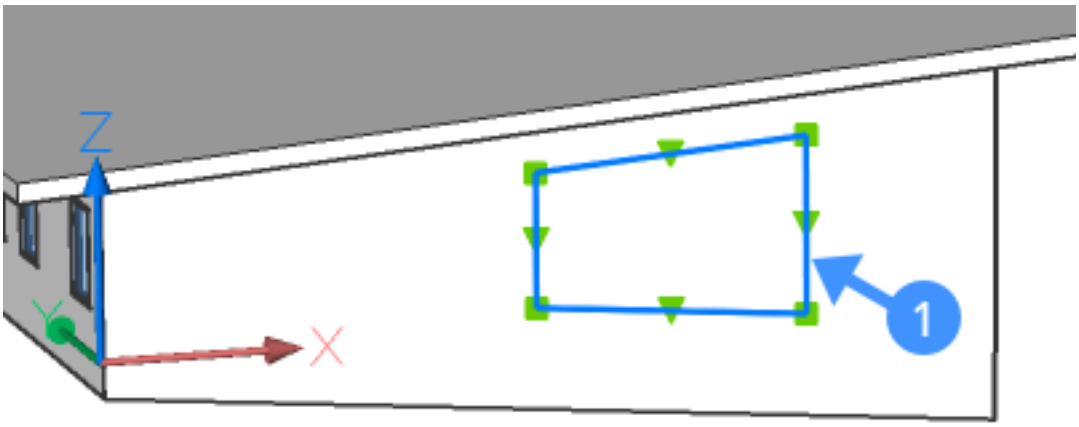


Figure 2.90

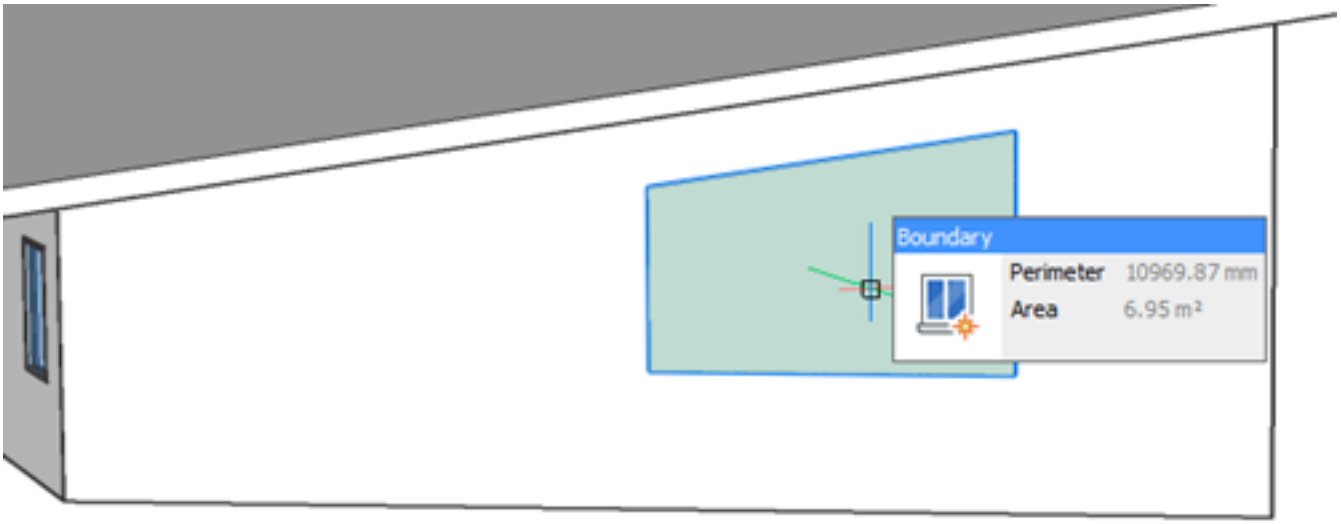


Figure 2.91

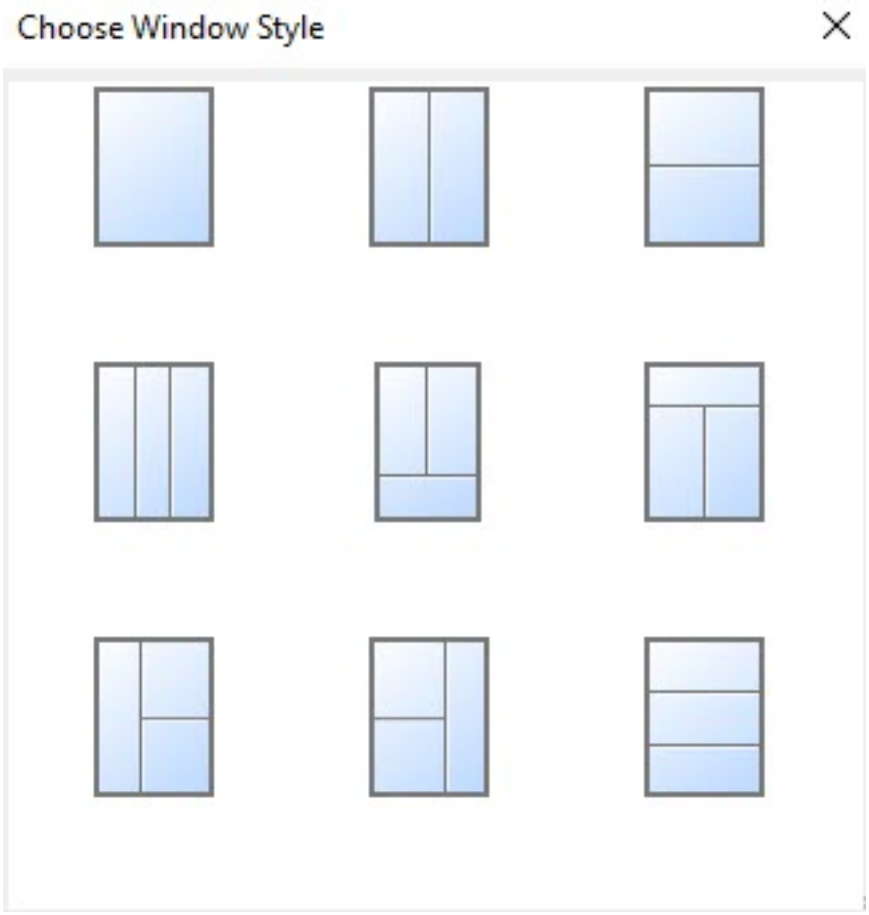


Figure 2.92

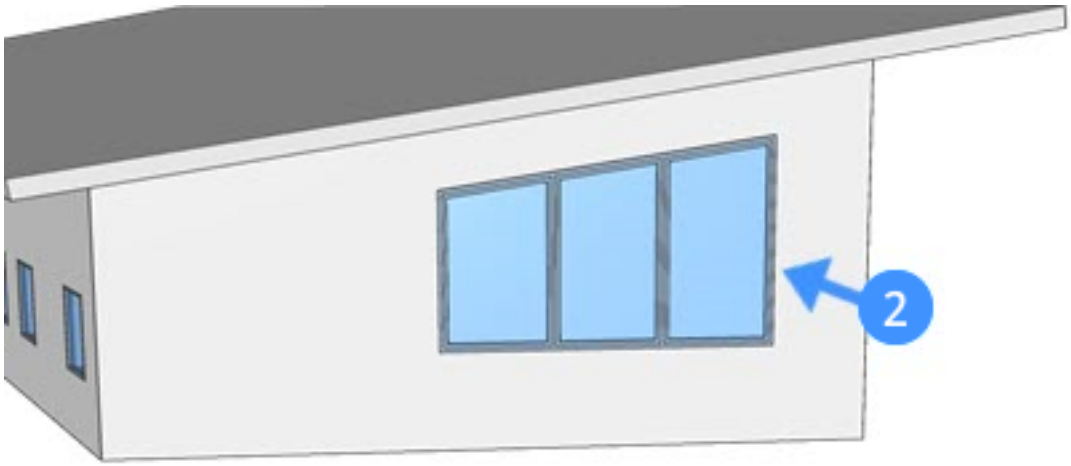


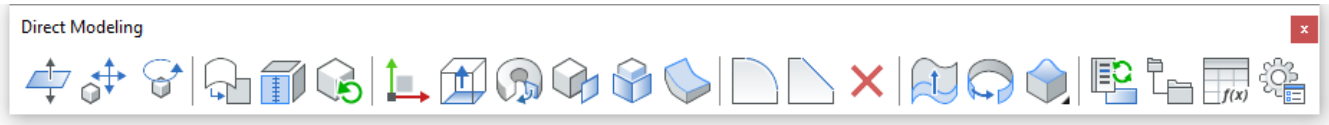
Figure 2.93

Parameters	
Name	Triple Panel Vertical Window:2
File	
LiningThickness	60
LiningDepth	60
MullionThickness	60
FirstMullionOffset	0.33333333
SecondMullionOffset	0.66666667
GlazingDepth	30
GlazingOffset	30
PlacementDepth	50
LeftPanelThickness	60
LeftPanelDepth	60
MiddlePanelThickness	60
MiddlePanelDepth	60
RightPanelThickness	60
RightPanelDepth	60

Figure 2.94



Figure 2.95



2.4.4 Connecting Solids

Since solids, like any other BIM entity, are created as native 3D solids, any of the 3D Solids Editing commands or Direct Modeling (1.18.2) commands can be used to modify solids. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/DirectModelingToolbar.png
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIM-ConnectToolbar.png})
The LConnect (6.20.19) command ( ) creates an L-connection (bisector or parallel) between two solids. The TConnect (6.29.10) command ( ) creates a T-connection between two solids. The BimStructuralConnect (6.7.30) command ( ) connects structural profile solids such as beams and columns. The BimFlowConnect (6.7.11) command ( ) creates a connection between flow segments, such as pipes or HVAC ducts. The BimSplit (6.7.32) command ( )splits segmented solids into separated solids and allows to split a solid using a selection of cutting faces. The Slice (6.27.36) command ( ) cuts a 3D solid with a plane into two parts. The Push/Pull (6.12.20) command ( ) modifies the thickness, height or length of a solid. The BimCopy ( ) command creates a parallel copy of a solid or copies part of a solid, defined by a detected boundary (if the Boundary Detection option of the SELECTIONMODES system variable is checked). The BimDrag (6.7.8) command ( ) moves the solid of elements. The procedures below use the Quad cursor menu (1.12.5).

Tutorial: How to connect walls Click here (2.3.5) to watch

Figure 2.96



Creating L-connections

1. Make sure SELECTIONMODES = 0 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelin
2. Select the first solid. The solid highlights.
3. Hover over the second solid. When the solid highlights, choose LConnectin the Modelcommand group of the Quad. The selected solids are connected: the default connection type is L-bisector. If HKA (1.13.11) is ON, the Hotkey Assistantwidget displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelin
   You are prompted: Accept connection <Ok>/Next
4. Do one of the following:
5. Press Enter or right-click to accept the default L-bisector connection.
6. Hit the Ctrl key to cycle between L-parallel and L-bisector connections and press Enter or right-click when the desired connection displays. The 3D model updates dynamically when cycling the L-connection types.

Note: L-connections can also be created between non-vertical solids such as roof slabs.

Figure 2.97

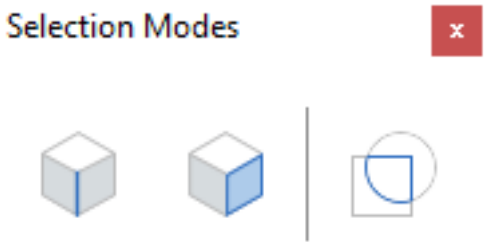


Figure 2.98



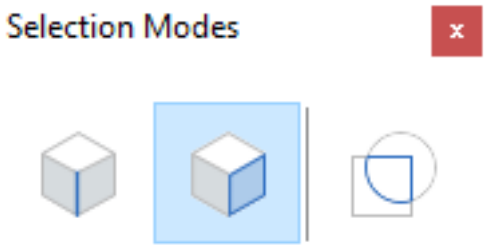
Creating T-connections

The below procedure allows to connect minor faces of a solid to the major faces of other solids, regardless what the orientation of the faces is. As a result, you can use **TCONNECT** command to T-connect solids, for instance, to connect the top or bottom face of a solid to a roof or floor slab or to connect the side faces of a slab to solids.

1. Make sure the Select Faces option of the SELECTIONMODES system variable is selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectionModesFace.png})
2. Move the cursor over the minor face of the first solid at the end that needs to be connected. Hit the TAB key to highlight obscured faces. Click when the face highlights.
3. Repeat the previous step to select more faces. To select the last face, go to the next step.
4. Move the cursor over the face. If necessary, hit the TAB key to highlight an obscured face. When the face highlights, choosein the Modelcommand group of the Quad.
5. Prompts you: Select entities to connect to <ENTER to connect to nearest>:
6. Optionally: Select the solid(s) to connect to. The selected face is connected to selected face or the nearest face of the selected solid.
7. Right click or press Enter to accept the connection.

TIP: Hover over the minor face of the solid, then choose Connect with Nearest ( ) in the Model command group of the Quad.

Figure 2.99



(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/staticbricsys1.png})	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtec
3D Model	**Calculated Section**

Creating T-connections ply-by-ply To control the connection of composition plies between two building elements, you can execute Push/Pull (6.12.20) on a ply in a clipped BimSection (6.7.27)ifShow Composition ( )of the solids is ON. To create a T-connection ply-by-ply in a section, do the following:

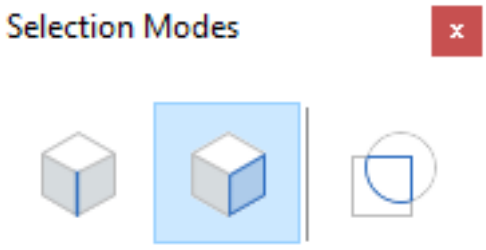
1. Place the cursor over the face of the ply you want to connect. If necessary, repeatedly press the TAB-key to select an obscured face.
2. When the ply face highlights, choose Push/Pullin the Quad.
3. Specify to which point the selected ply should reach by moving the cursor or typing in the distance.
4. Right click or press enter to confirm. The ply of the first solid is connected to the ply of the second solid. The connected ply is subtracted from the second solid.

Modifying the thickness of a solid

1. Make sure the Select Faces option of the SELECTIONMODES system variable is selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectionModesFace.png})
2. Move the cursor over the face of a solid If necessary, hit the TAB key to highlight an obscured face
3. When the face highlights, choose Push/Pull in the Modelcommand group of the Quad.
4. The selected face of the solid is dragged The dynamic dimension field displays the relative displacement of the face (= with respect to its original position)
5. (option) By hitting tab, it is possible to change theface selected as the reference face for the dynamic dimension

Note: When a composition with locked thicknesses (non-variable thickness) is applied to a solid and if the thickness of the solid is not equal to the total amount of the composition thickness, the solid displays with red hatches. Execute the BimUpdateThickness (6.7.35) command to re-apply the composition to such solid.

Figure 2.100

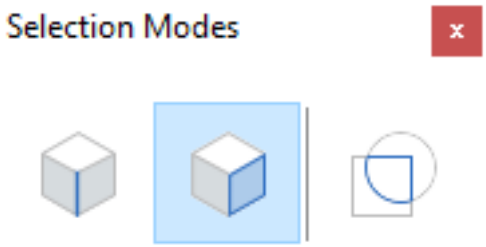


Modifying the height of a wall

Modifying the dimensions of a single wall

1. Make sure the Select Faces option of the SELECTIONMODES system variable is selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectionModesFace.png})
2. Move the cursor over the top face of the wall. If necessary, hit the TAB key to highlight an obscured face.
3. When the face highlights, choosePush/Pullin the Model command group of the Quad. The selected face moves dynamically with the cursor. The dynamic dimension fields displays the incremental height.
4. Type a value in the dynamic dimension field and press Enter.
5. Hit the TAB key to use another reference face, e.g. the bottom face of the solid, then type a value in the dynamic dimension field.

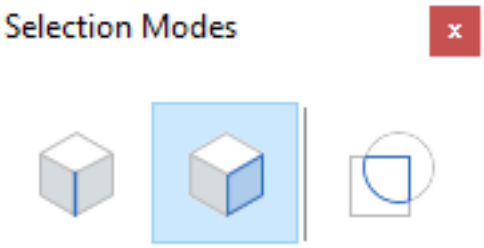
Figure 2.101



Modifying the height of multiple walls Using the SelectAlignedFaces (6.27.16) command (Quad > Select:) it is easy to select the top face of all solids that lie in the same horizontal plane.

1. Make sure the Select Faces option of the SELECTIONMODES system variable is selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectionModesFace.png})
2. Move the cursor over the top face of one of the walls. If necessary, hit the TAB key to highlight an obscured face.
3. When the face highlights, choose Select Aligned Facesin the Solid/Face Selection command group in the Quad. All faces lying in the plane of the selected face are selected.
4. Now follow the same steps as for a single wall

Figure 2.102



Modifying the height ply-by-ply When the Display Composition (2.6.2) property of a solid is set, and the Select Faces option of the SELECTIONMODES system variable is selected, the height or length of the composition plies can be modified separately.

1. Move the cursor over the ply you want to edit and choose Push/Pullin the Model command group of the Quad when the ply highlights. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniqu
2. Type value in the dynamic distance field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/M
3. Press Enter to accept the distance. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelin

Figure 2.103

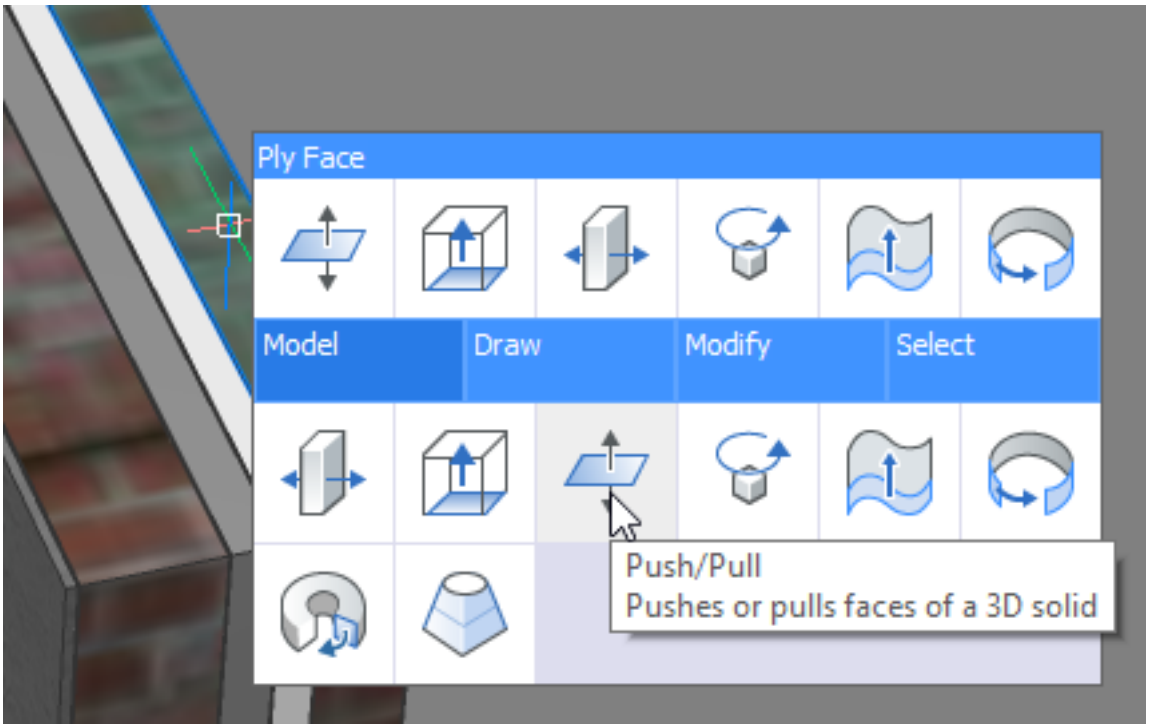


Figure 2.104

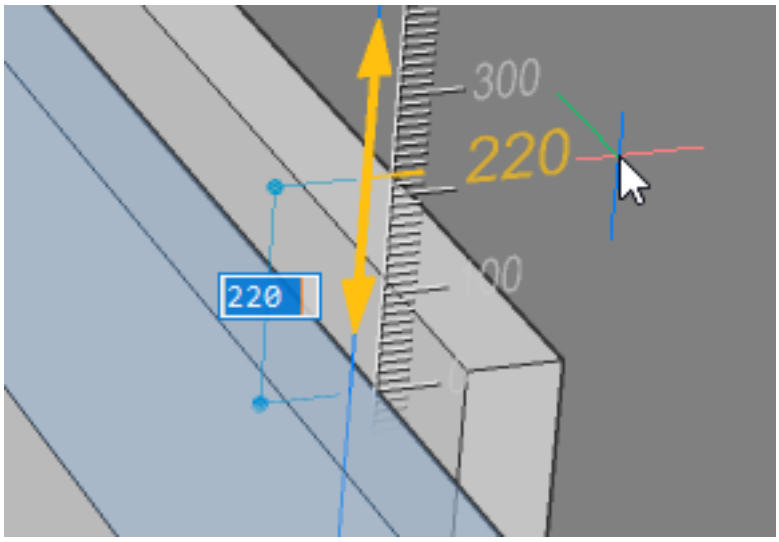
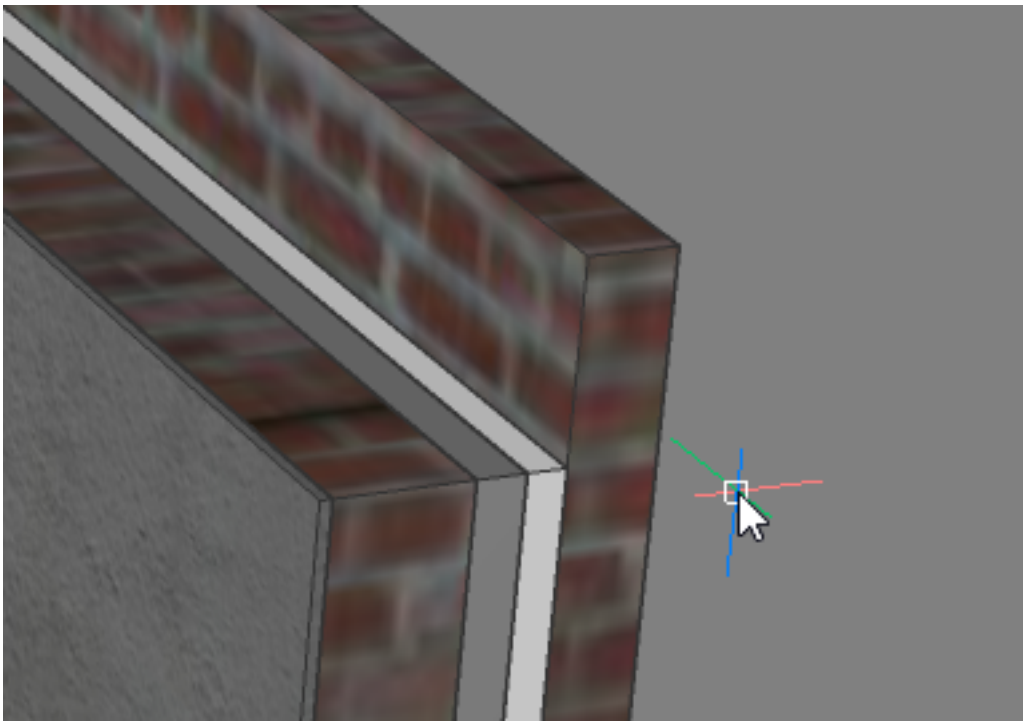


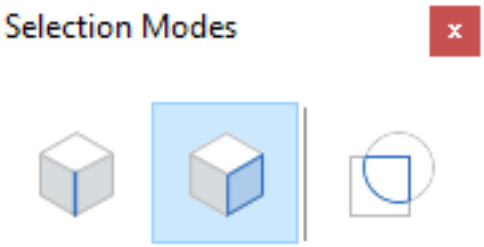
Figure 2.105



Setting the distance between parallel solids

1. Make sure the Select Faces option of the SELECTIONMODES system variable is selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectionModesFace.png})
2. Move the cursor over a face of the solid to be moved.
3. When the face highlights, choose BimDrag in the Modelcommand group of the Quad. The dynamic dimension field displays the relative displacement of the solid (= with respect to the original position of the selected face).
4. Repeatedly hit the TAB key until the desired reference face on a parallel solid highlights. Press and hold the Shift key while hitting the TAB key to cycle reference faces in reversed order. The dynamic entry field displays the distance with respect to the currently highlighted reference face.
5. Type a value in the dynamic entry field, then press Enter.

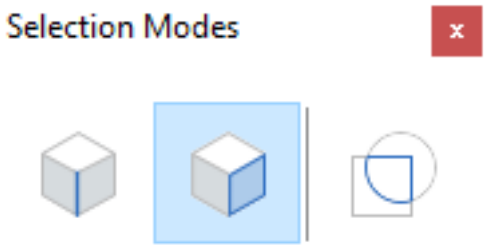
Figure 2.106



Creating a parallel copy of a solid

1. Make sure the Select Faces option of the SELECTIONMODES system variable is selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectionModesFace.png})
2. Move the cursor over a major face of the solid to be copied.
3. When the face highlights, choose BimCopyin the Modelcommand group of the Quad.
4. Repeatedly hit the TAB key until the desired reference face on a parallel solid highlights. Press and hold the Shift key while hitting the TAB key to cycle reference faces in reversed order. The dynamic entry field displays the distance with respect to the currently highlighted reference face.
5. Type a value in the dynamic entry field, then press Enter.

Figure 2.107



Creating a partial copy of a solid

1. Make sure the Enable boundary detection option of the SELECTIONMODES system variable is selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SelectionModesBoundary.p

2. Highlight the part of the boundary for the part of the solid you want to copy.(\vref {fig:C:/Dev/Task-2448/images/staticbric

3. When the face highlights, choose BimCopy in the BIM command group of the Quad.

4. Repeatedly hit the TAB key until the desired reference face on parallel solid highlights Press and hold the Shift key while hitting the TAB key to cycle reference faces in reversed order. The dynamic entry field displays the distance with respect to the currently highlighted reference face; Type a value in the dynamic entry field, then press Enter. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/CopySol

Figure 2.108

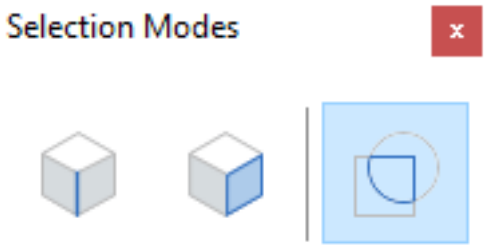


Figure 2.109

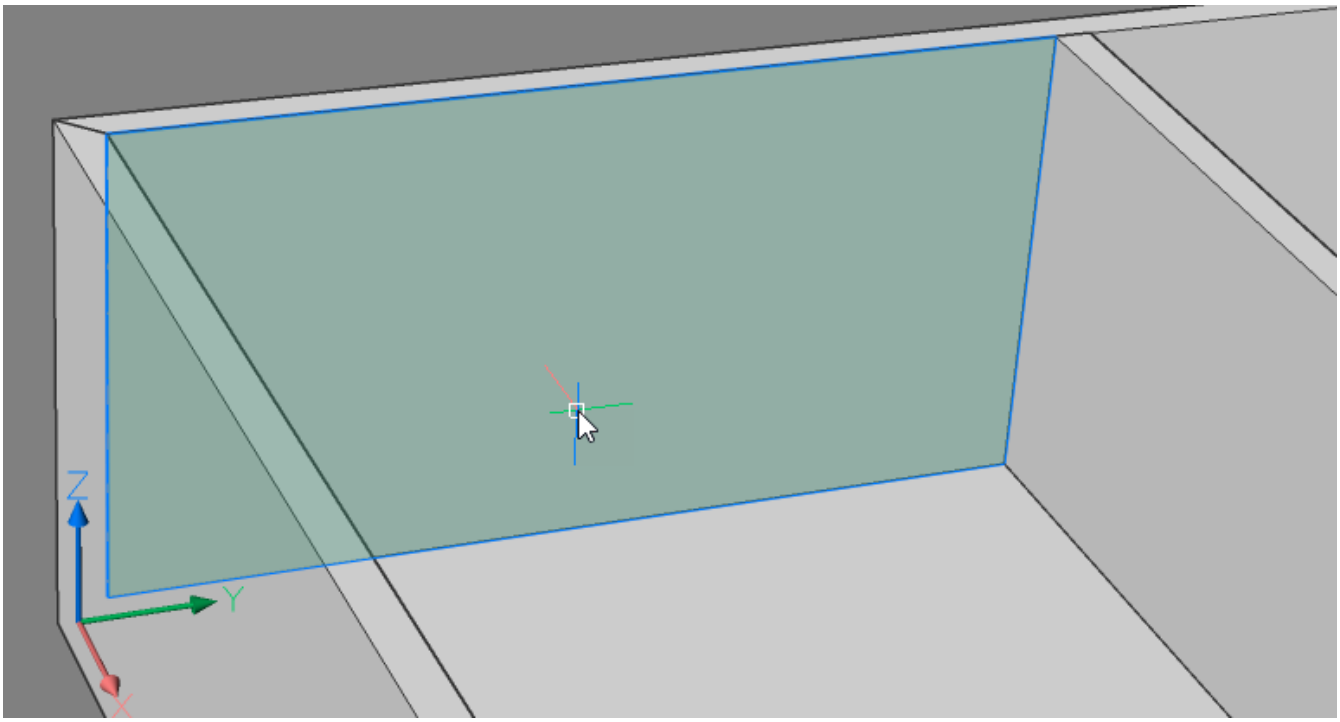


Figure 2.110

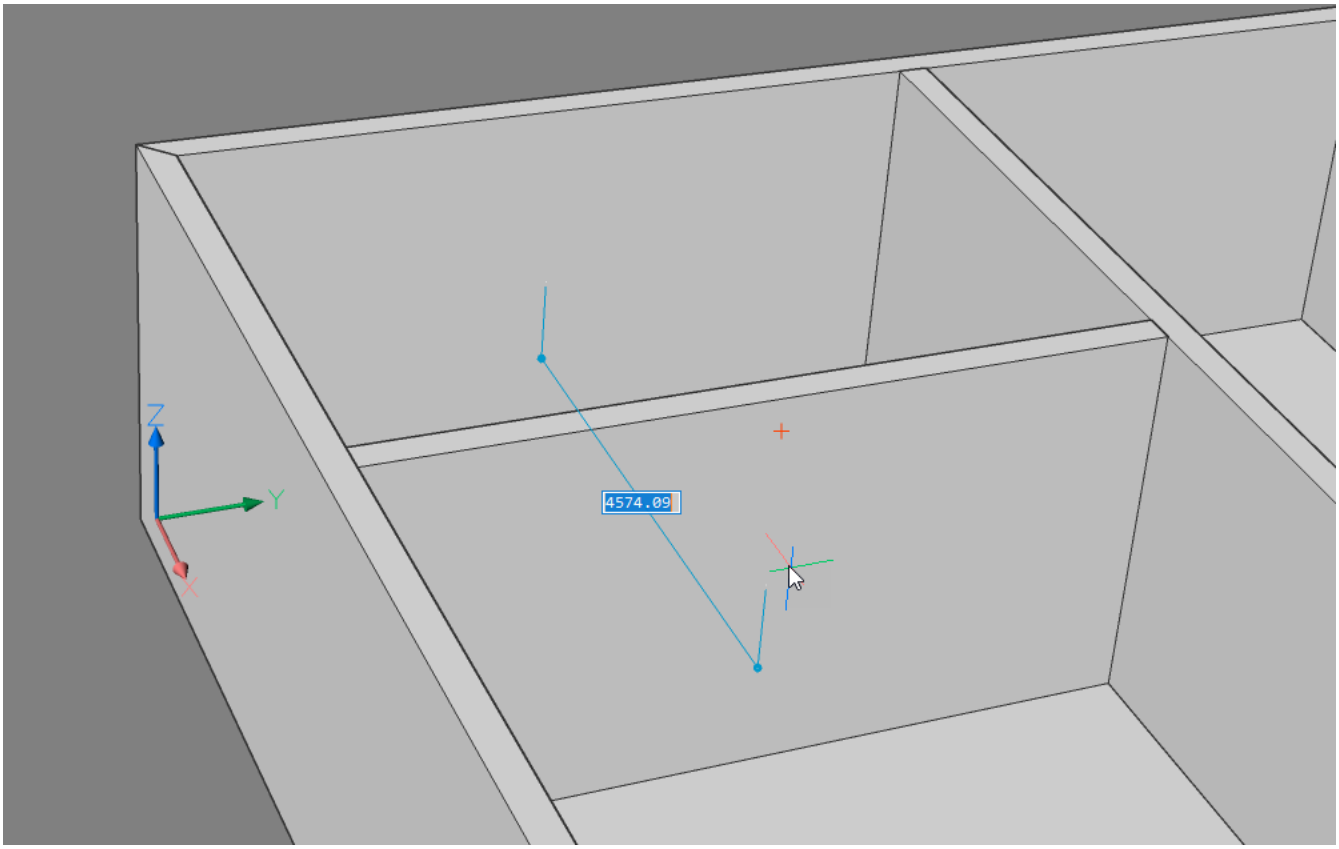


Figure 2.111

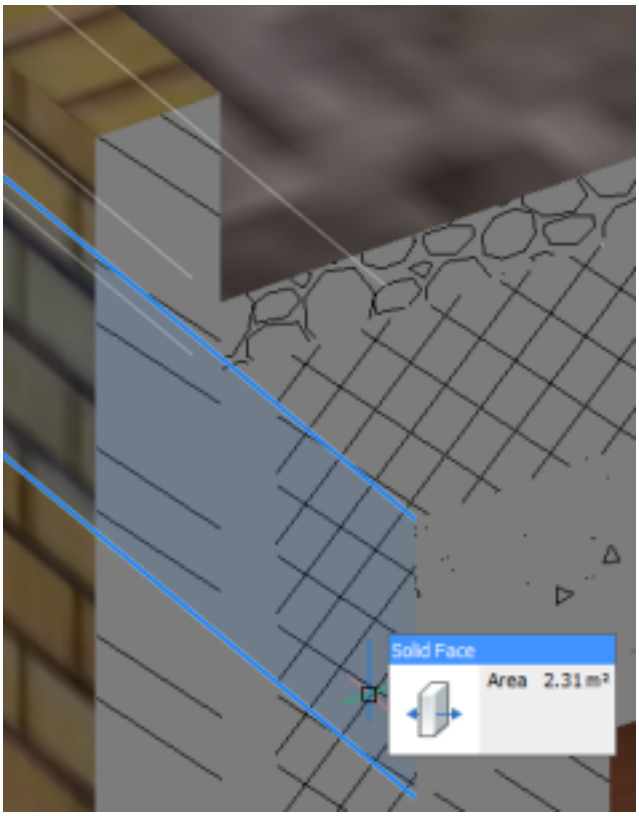
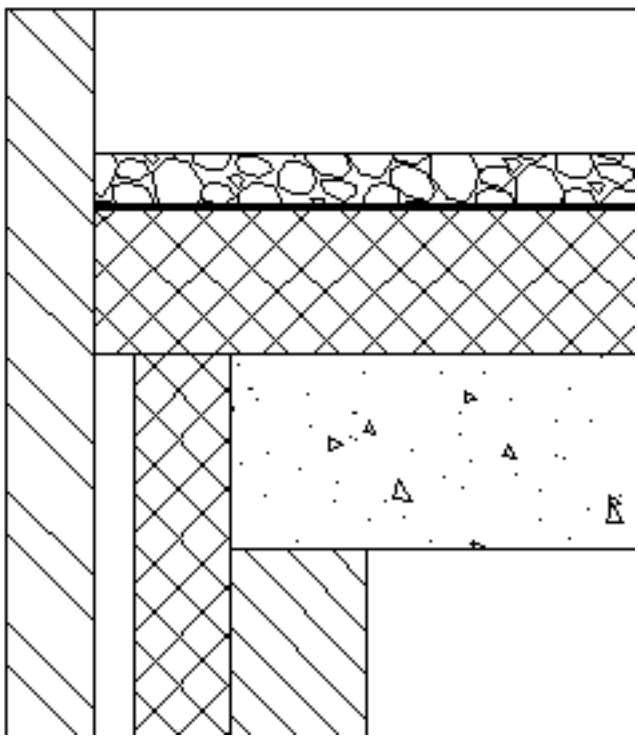


Figure 2.112



2.4.5 Creating a Custom Window

Overview: In this lesson, you learn how to create a custom window to insert in your drawings. **Lesson Objectives** After completing this lesson, you will be able to:

- Create the 3D geometry of a window element.
- Utilize the pre-defined templates.
- Understand how the subtractor solid can be used in creating an opening for insertion.

Creating a Custom Window

In BricsCAD, you can create any geometry or component from scratch. Once you have your geometry in 3D you can insert and use it in your BIM model. You can make changes or add more details to your windowat any time.

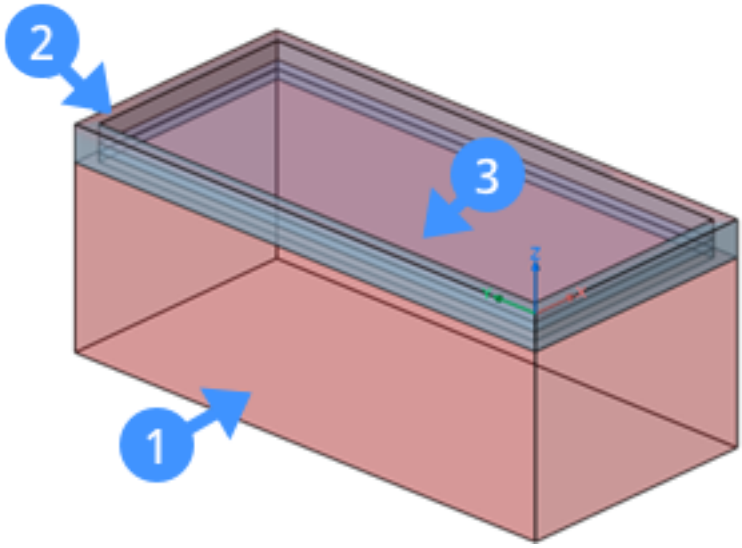
Creating a simple window

This process is explained in 4 parts to make the steps of creating a 3D window geometry straightforward.

- Create the file
- Create the subtractor solid (1)
- Create the window frame (2)
- Create the (Glass) panes (3)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/CreatingSimpleWindow.pn

Figure 2.113



Step 1: Creating the file

1. Click **New** in the **File** menu
2. Select one of the **pre-defined templates** to create a simple window specifying the unit system of your drawing. **Note:** The pre-defined templates contain the layers needed to create the geometry of the window or door. These .dwt file format templates are BIM-Window-imperial (4) and BIM-Window-metric (5) whichassure the required layers and properties are displayed in your drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
3. Set the **BIM_Subtract** layer current (6) either using the drawing explorer or layer field in the properties panel or Layers Panel on the right-hand side. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modeli
4. Create a closed polyline (e.g. a rectangle) that represents the form of the opening for the insert. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/WindowDrawPolyline.png}
5. Save the file in any location. The library of predefined windows and doors is located in the support folder. By default this is: **C:\Users<user_name>\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\Support\Bim\Component** This path can also easily be found by typing SUPPORTFOLDER in the command line.

Figure 2.114

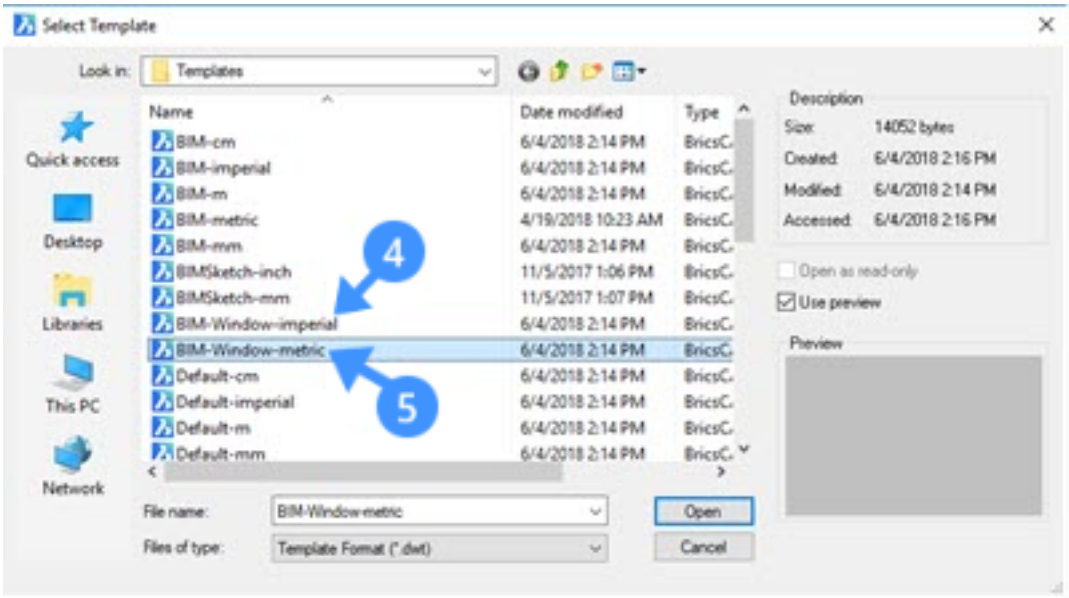


Figure 2.115

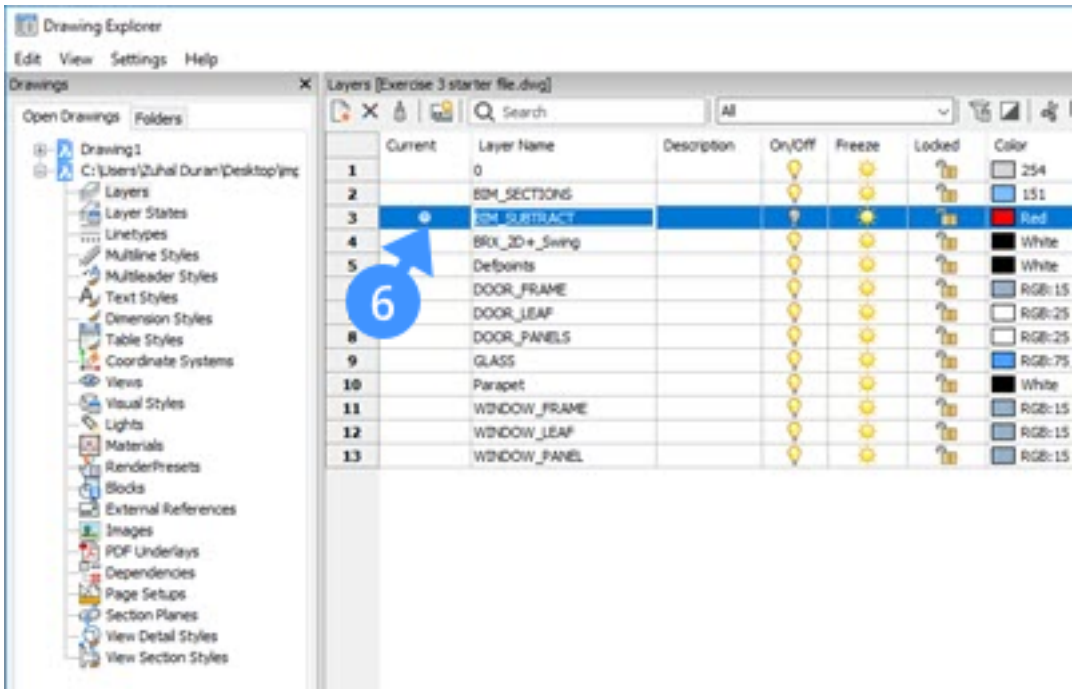
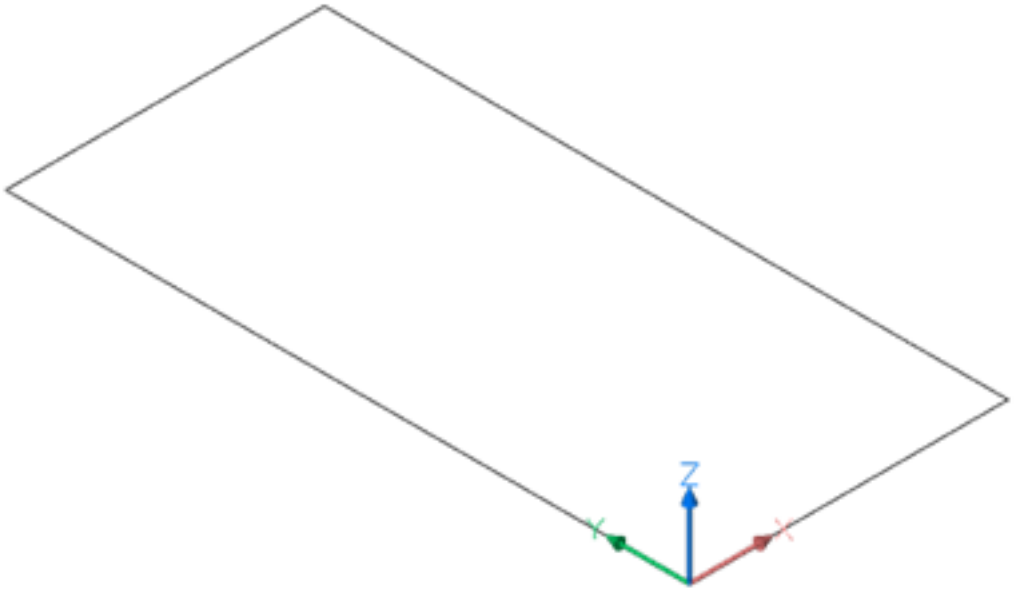


Figure 2.116



Step 2: Creating the subtractor solid The subtractor solid will be used to create an opening. **Note: BIM Subtract** Layer must be a current layer.

1. Use the outline of the solid which you have previously created (e.g. a rectangle). The outline lies in the first quadrant of the XY plane of the **World Coordinate System** (WCS). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/
   The lower left corner of the rectangle lies at the origin (0,0,0) of the WCS (7).

2. Select the Extrude tool and move the cursor downward (8) to extrude your solid outline in the negative Z direction. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/WindowSubtrac

3. Do one of the following to finish the subtractor solid creation.
   - Click a point
   - Type a value in the dynamic dimension fields, then press Enter.

Figure 2.117

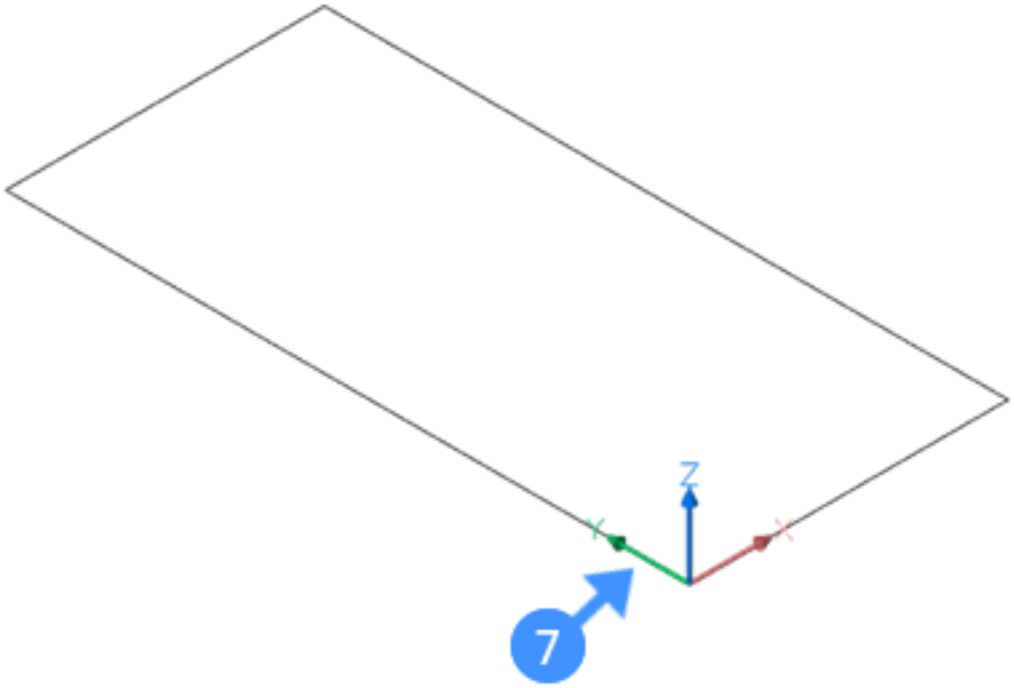
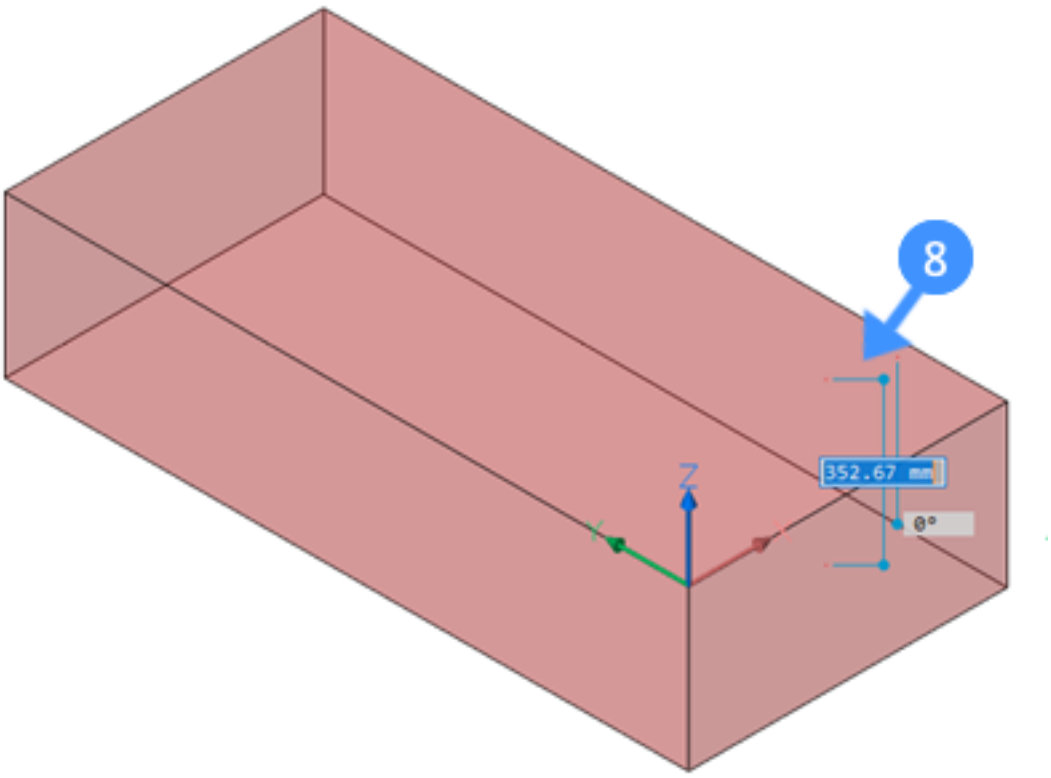


Figure 2.118



Step 3: Creating the fixed frame The outline of the fixed frame coincides or is parallel (in case of a rebate) with the outline of the subtractor solid.

1. Set the Window_Frame layer current.
2. Highlight the top face of the subtractor solid.
3. Next, extrude the top face of the subtractor solid in the negative Z direction (9) using the **Extrude** tool 'Create' (10) option by pressing the CTRL key once. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/
4. Type a value in the dynamic dimension field to specify the height of the fixed frame. **Note:** The height of the extrusion will be the thickness of the fixed frame.
5. Turn off 'BIM_SUBTRACT' layer by clicking the light bulb (11) next to the layer name. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMSubstractLayer2.png})
6. Use the Offset tool to create a parallel offset of the outline of the fixed frame (12). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/FrameOffset2.png}) **Note:** The distance of the offset equals the width of the frame.
7. Hover over the inside boundary of the parallel offset (13) and select Extrude tool from the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/FrameBoundary.png})

Figure 2.119

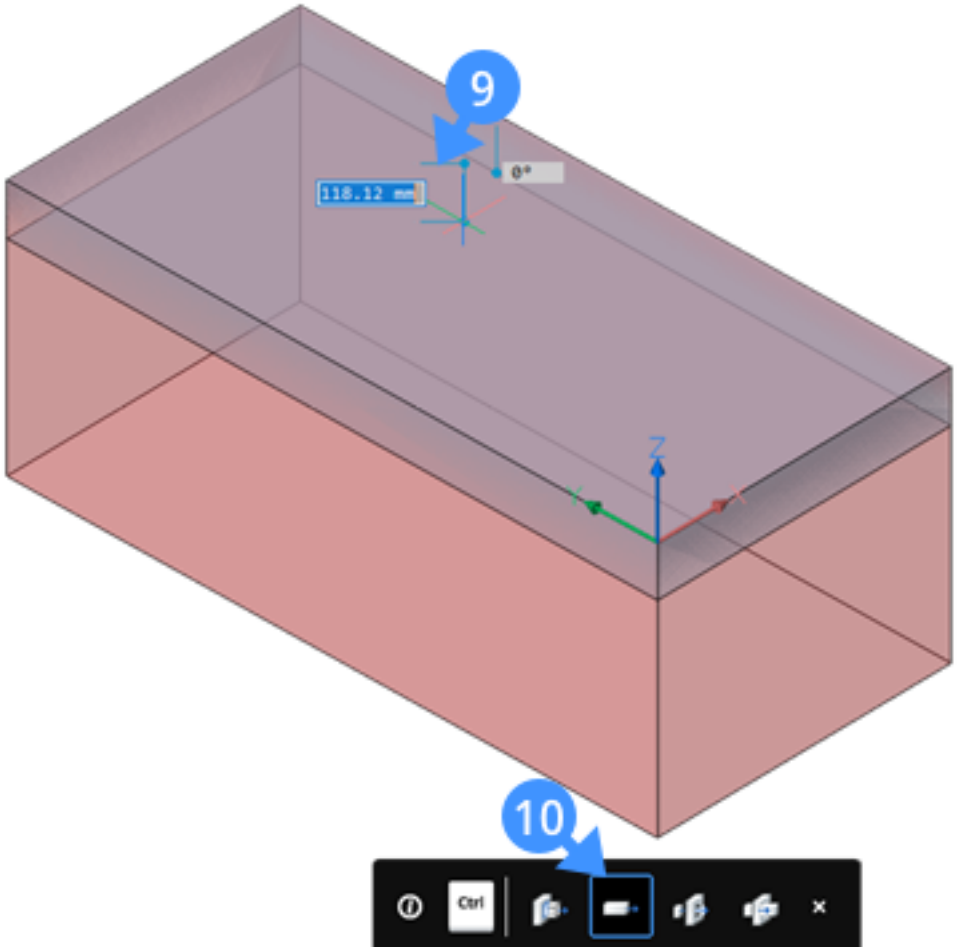


Figure 2.120

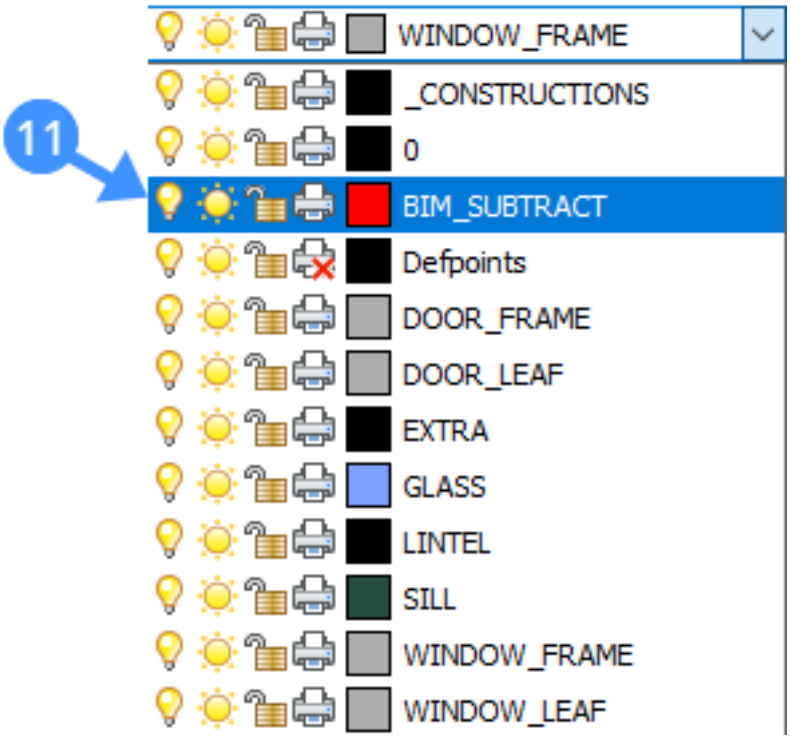
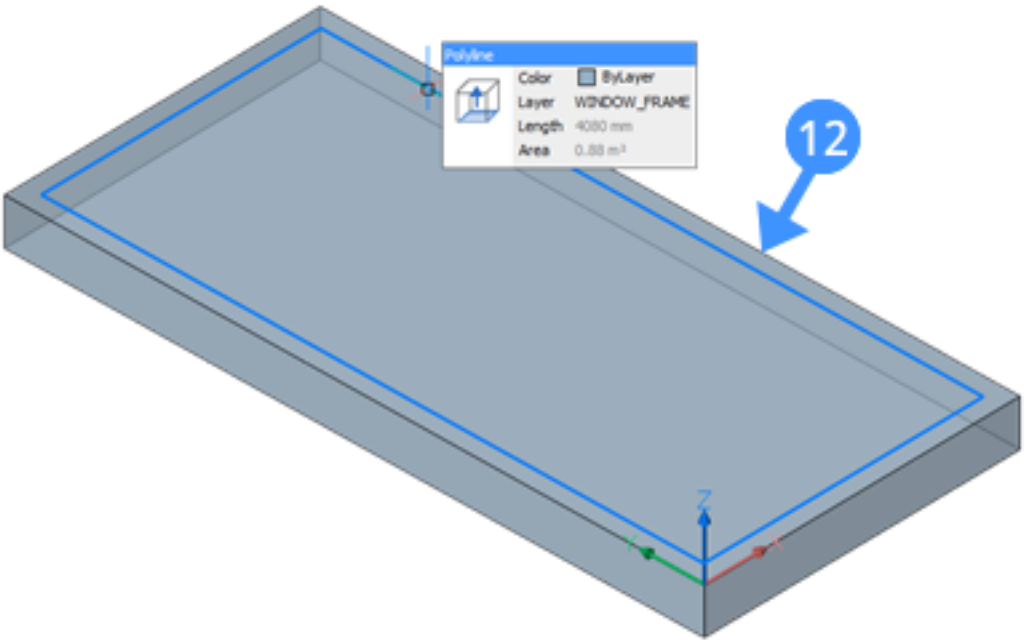


Figure 2.121



8. When the extrude mode is on the Subtract option (14) move your cursor downward (15) to create an opening in the fixed frame. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/Fram

9. Do one of the following to finish the creation of the opening in the fixed frame.
   - Click a point
   - Type a value in the dynamic dimension fields, then press Enter.

Figure 2.122

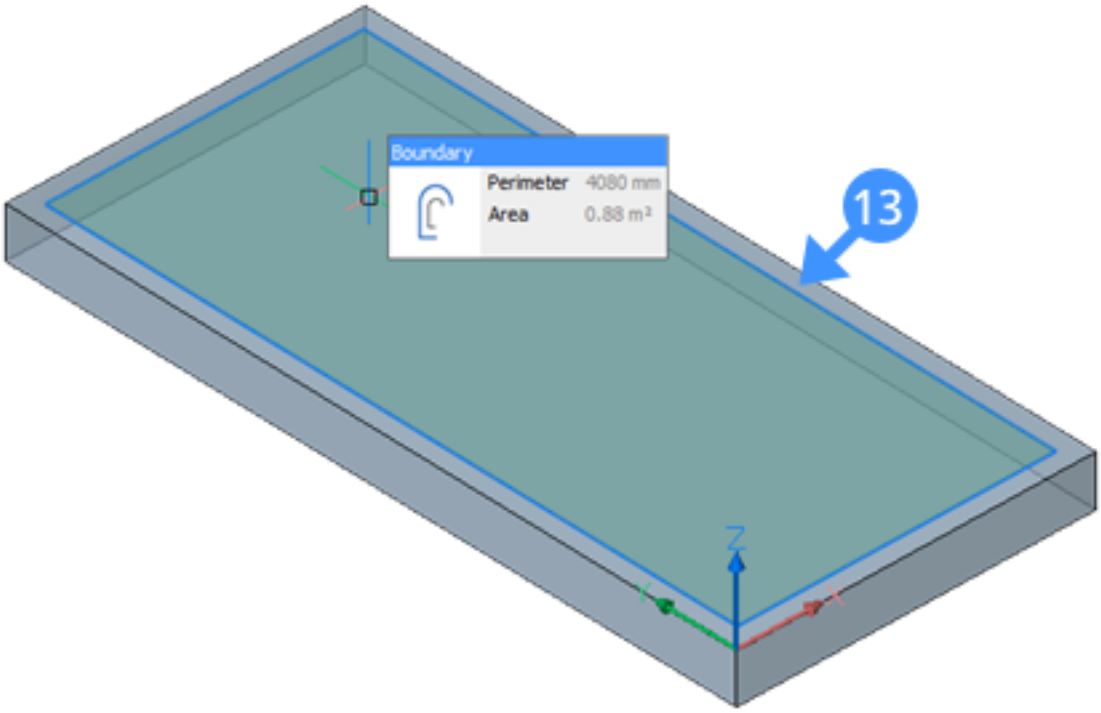
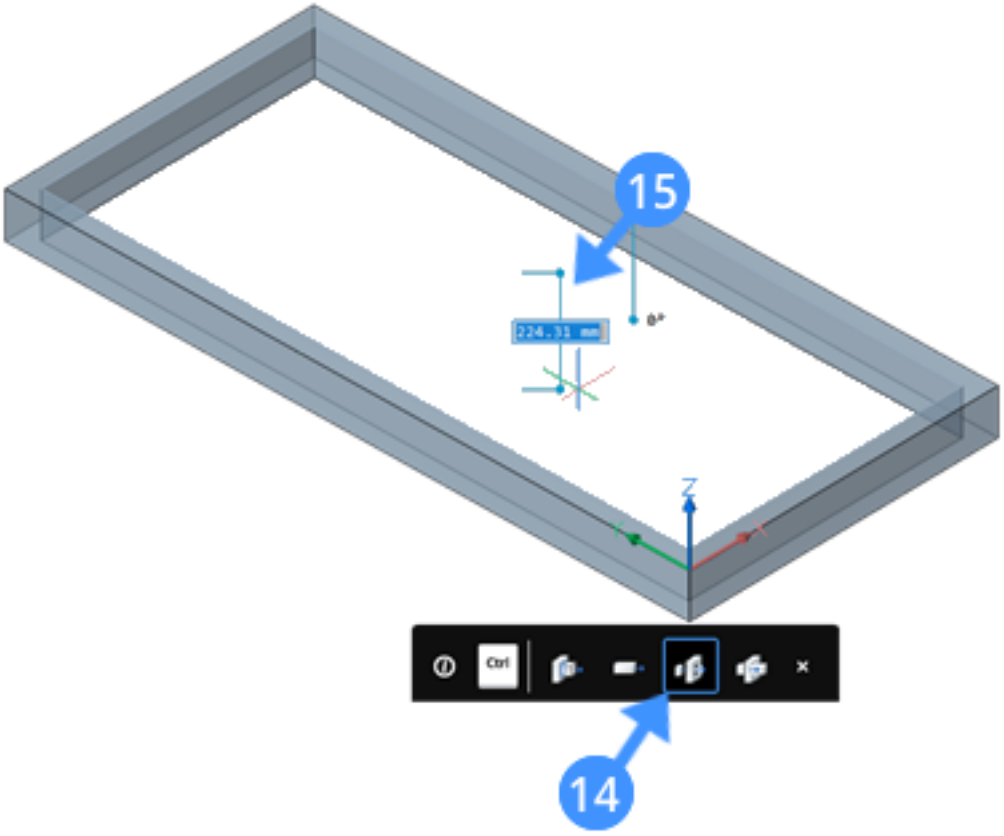


Figure 2.123



Step 4: Creating the glass panes

1. Set the Glass layer current.
2. Use the DMEXTRUDE command to extrude the inside boundary of the window frame. **Note:** The height of the extrusion equals the thickness of the glass pane. The following illustration shows the inside boundary of the window frame after using DMExtrude to create a window glass pane. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/GlassExtrude.png}) Note that to create a glass pane, the create option should be selected from the Tooltip of DMExtrude.
3. Move the glass pane solid down in the negative Z-direction if the outside face of the fixed frame does not coincide with the outside face of the solid (wall) the insert will be placed in. The following illustrations show the glass pane solid after moving it (17) down. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/

Optionally, before saving the drawing, classify the BIM as a window element. To do so, follow these steps:

1. Type BIMCLASSIFY in the command line.
2. Press 'I' for window and press enter to accept it.
3. Next, press 'D' for Drawing and press Enter key until the command is executed.

The drawing is classified as a window. When you insert this block in your drawing, it will automatically be classified as a window.

5. Optionally, when no selection is active, you see a set of BIM properties on the right-hand side of your screen. Here you can assign information about the thermal transmittance coefficient of the window materials, fire rating according to the national fire safety classification, and so on. The following illustration shows the BIM properties of the window in the properties panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19

5. Save the drawing.

Figure 2.124

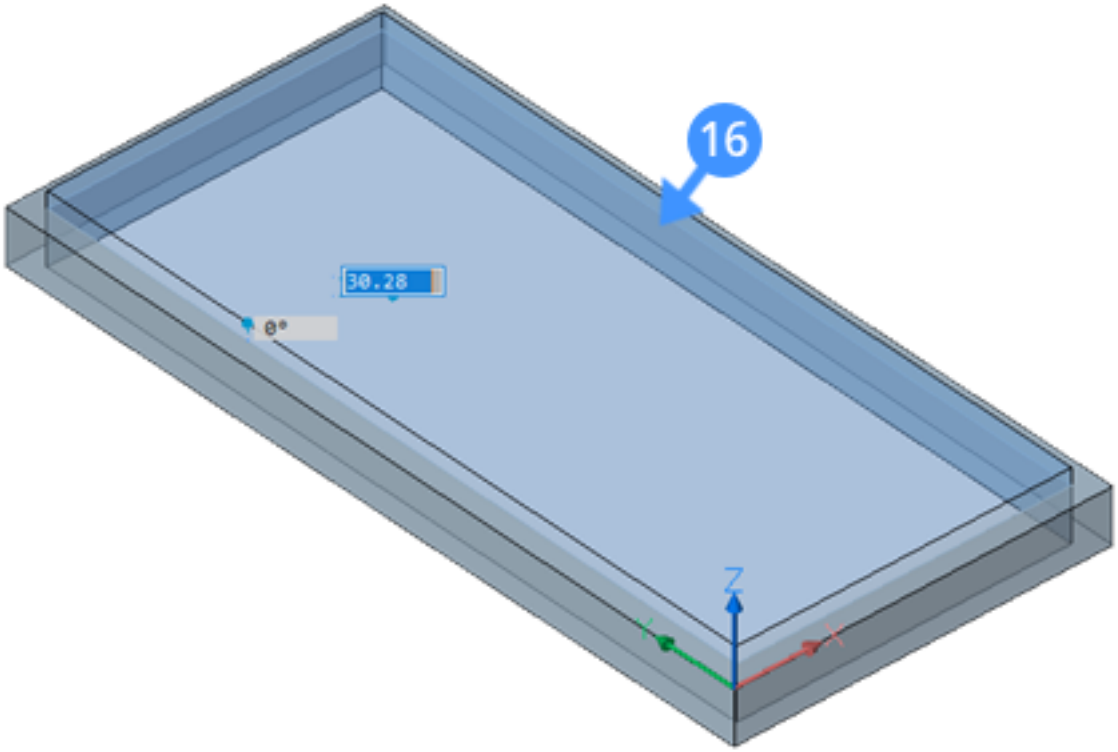


Figure 2.125

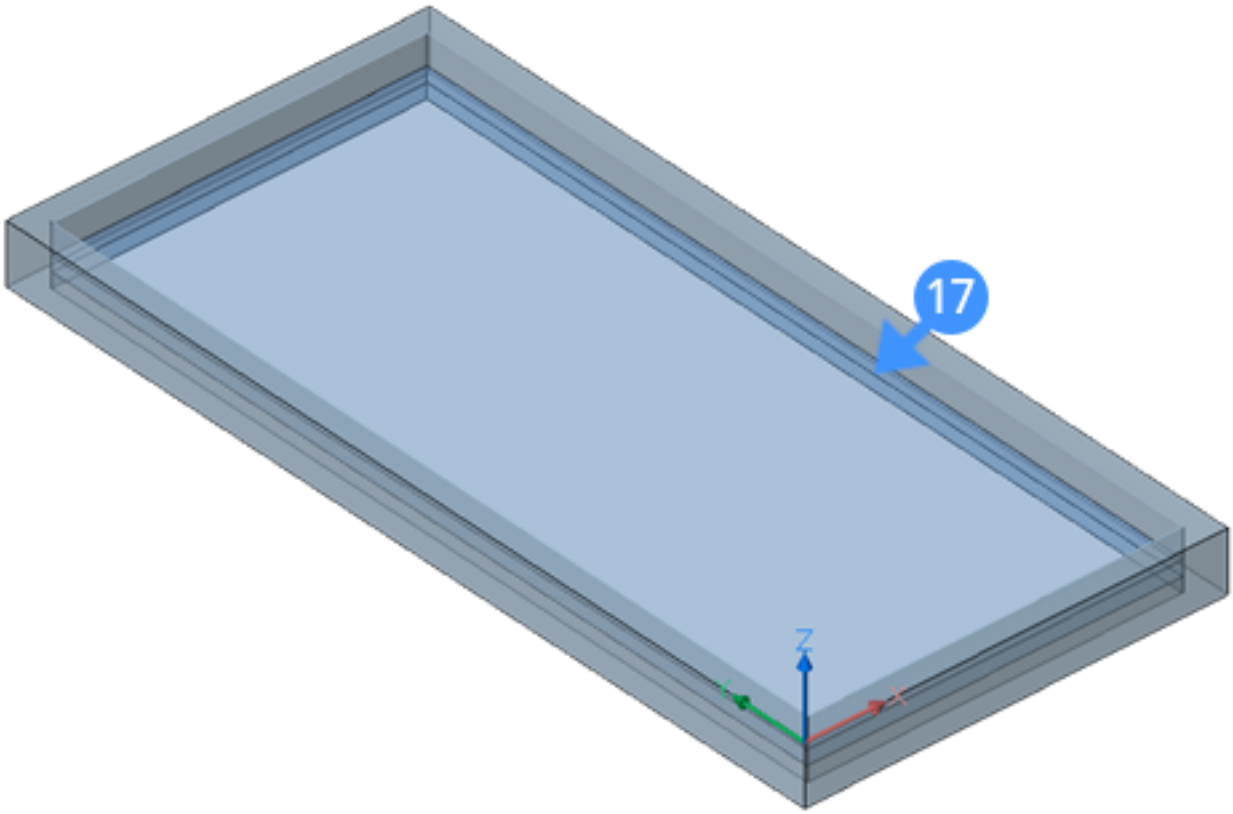


Figure 2.126

Window Common	
Reference	
Fire rating	
Acoustic rating	
Security rating	
Is external	Off
Infiltration	0 m³/s
Thermal transmittance	0 W/m²·K
Glazing area fraction	0
Smoke stop	Off

Adding 2D entities

Optionally, windows and doors drawings can contain 2D symbols. These 2D symbols will either replace the actual 3D Solid section of the window or door or be added to the 3D Solid section, depending on their layer:

- Symbols on a layer with prefix**BRX_2D_**will be used to replace the geometry resulting from the actual section,
- Symbols on a layer with prefix**BRX_2D+_**will be added to the section result. Only those 2D entities in a plane parallel to the section plane are.

2.4.6 Create Opening

Command: BimWindowCreate (6.7.36) **Overview:**Creates an opening without a window geometry. It uses the same command as BimWindowCreate. The command gives the possibility to create an opening at the location of the selected base profile. The base profile can be a boundary or a closed polyline.

Openingcreate

 command bar:**bimwindowcreate** menu bar:**Bim | Create Opening** toolbar:**Bim | Create Opening** ribbon: **Model| Component |Create Opening** Quadcursor menu:**Model | Create Opening**

Procedure: Creating an opening in an existing mode

1. Draw a rectangle on the face of a wall.
2. While boundary detection is on hover over inside the polygon(1) **Or** select the closed polyline for this rectangle shape.

3. Select the Create Opening tool from the Quad (2). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19
   The opening is created (3) based on the closed contour. In the Structure Browser, the opening is added as a Block Reference. The block reference is classified as an opening (4). (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc
   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/OpeningStructureBro
   If you select the opening from the structure browser, you will see a set of properties related to this opening element (5) in the properties panel. **Note:** If your closed contour has a rectangle shape, the opening element will include Width and Height parameters (6). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/M
   Note: Opening elements contain subtractor solids. To display the subtractor solid, open the layers panel and thaw the BIM_Subtract layer. **Note:** The associative tags can be generated using the bimtag (6.7.33) command for the opening element in a sectioned drawing. The following animated gif demonstrates how to create an opening based on a closed contour. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Mod

Figure 2.127

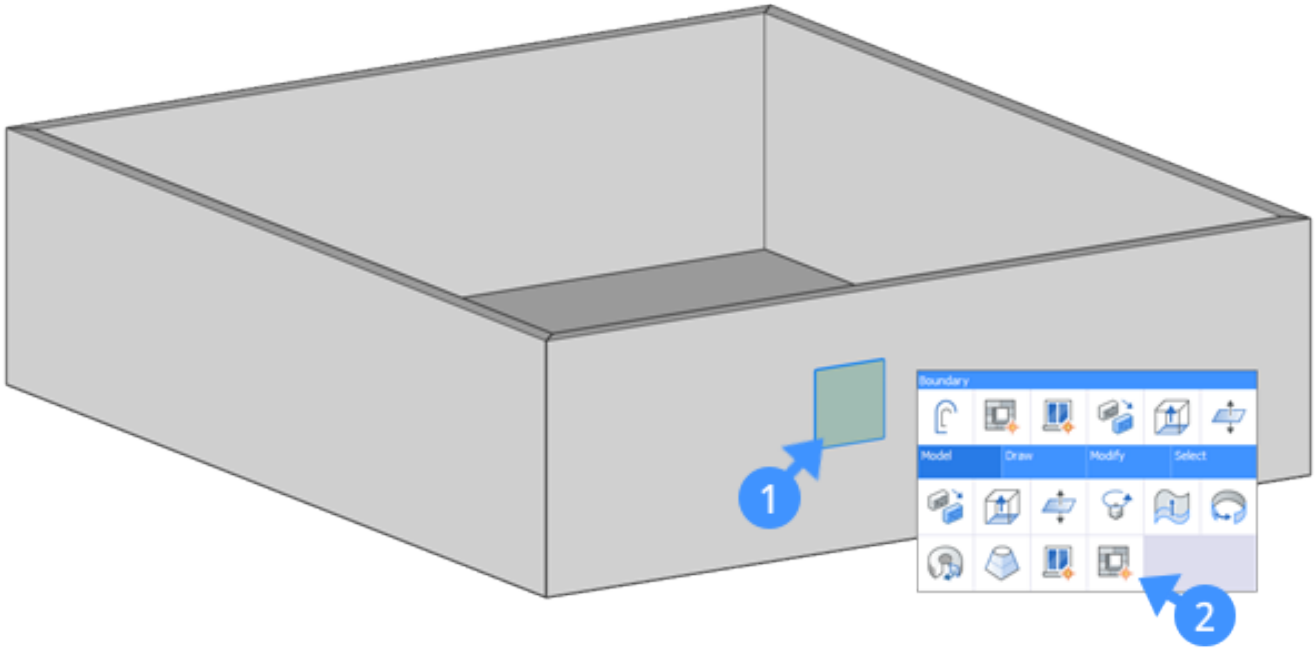


Figure 2.128

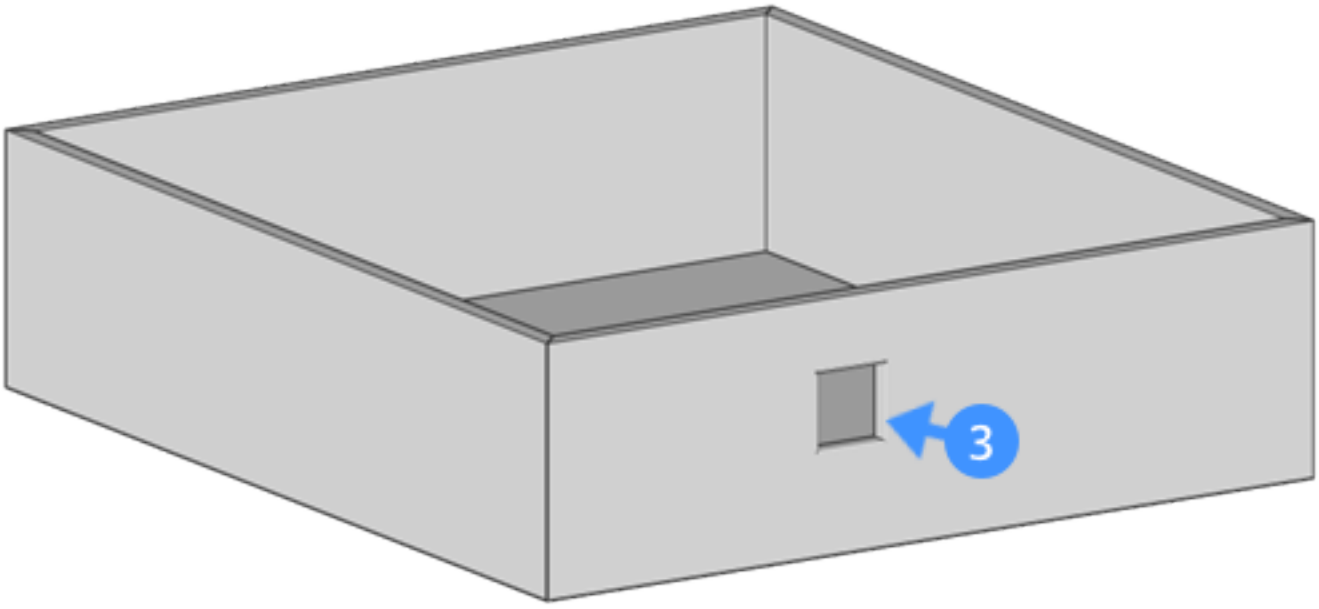


Figure 2.129

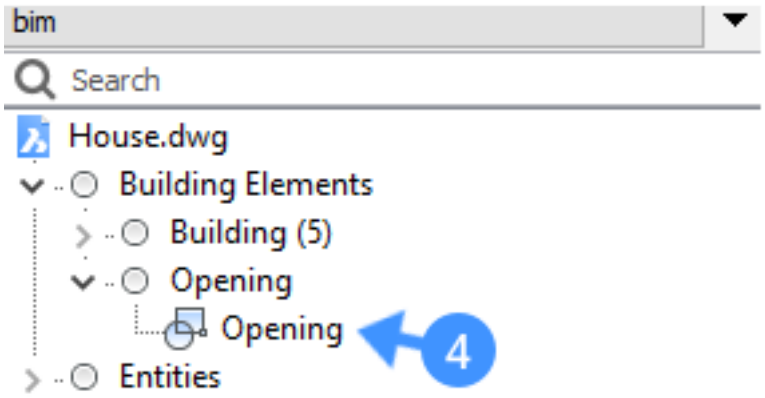


Figure 2.130



Figure 2.131

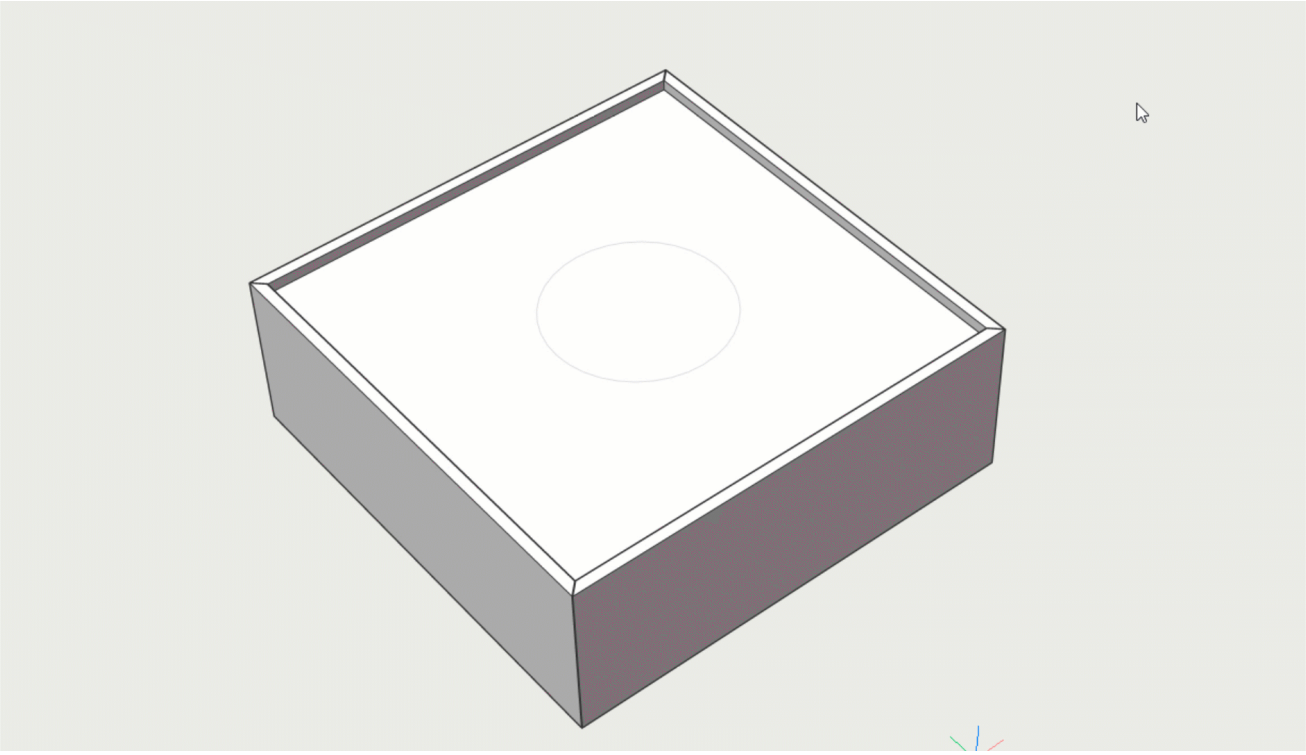
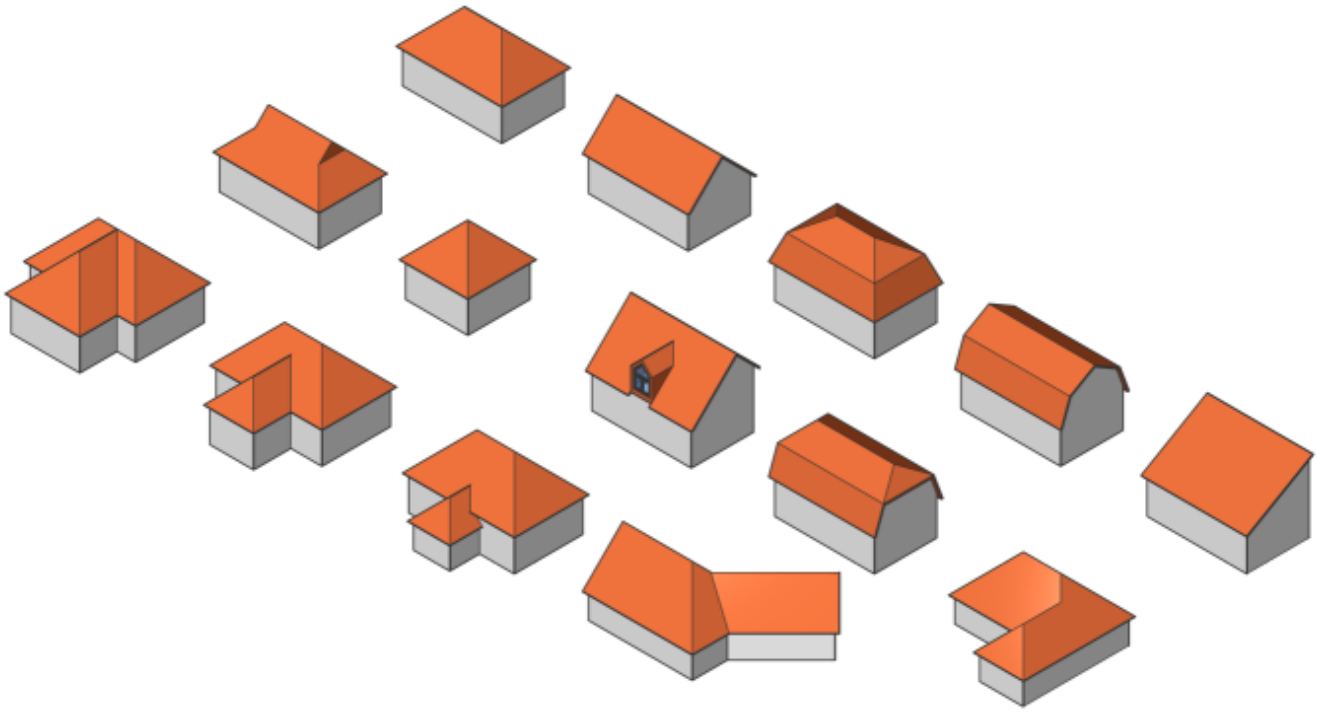


Figure 2.132



2.4.7 Creating Roofs

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/Roofs.png}) In BricsCAD BIM roofs are created using default commands and modeling techniques. The roof creation workflow consists of the following steps:

- Creating the roof footprint.
- Extrude the footprint and create a hip roof. (DmExtrude (6.12.13)command)
- Adjust the slope of the roof faces if necessary. (Manipulator (6.21.2) widget)
- Defining the thickness of the roof solid. (Body/Shelll option of the SolidEdit (6.27.39)command)
- Create additional roof components.

Tutorial: How to create a pitched roof Click here (2.3.2) to watch

Creating the roof footprint

You can create the roof footprint:

- From the exterior walls boundary.
- Creating a closed polyline by clicking points or other 2D procedures.

To create the roof outline from the exterior walls boundary.

1. Isolate (6.17.21) the exterior walls*.
2. Set the coordinate system to the top face of the exterior walls:
   - Type **ucs** and press Enter. You are prompted: **Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World] <World>:**
   - Snap to a top corner of an exterior wall, then press Enter.
3. Set the **Enable boundary detection** ON. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Mo
4. Move the cursor to a void area inside the exterior walls. The inside boundary highlights..
5. Choose **Offset** in the **Draw** tab of the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/
6. Do one of the following:
   - Type a value in the dynamic distance field.
   - Click a point

* Isolating entities is easily done using the Structure Panel. (6.27.50)

Figure 2.133

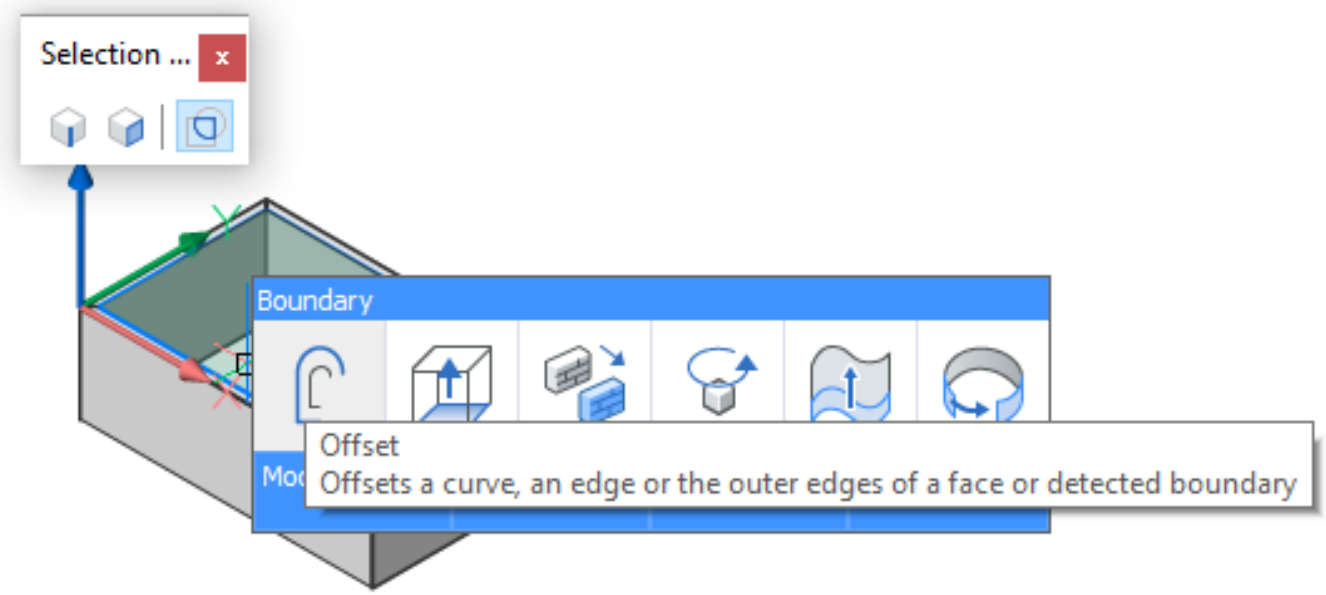
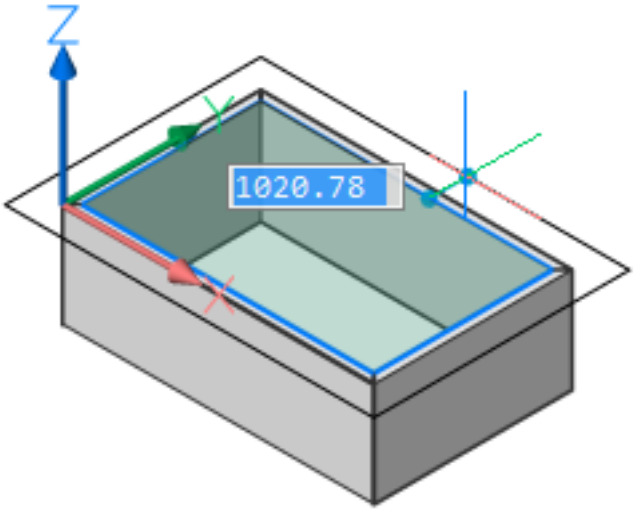


Figure 2.134



Creating a hip roof

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofHip.png})

1. Create the roof footprint.
2. Move the cursor over the roof outline and choose **Extrude** in the **Model** tab of the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/HipRoofStep1.png})
3. Move the cursor vertically, then hit the TAB key. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/E
4. Type a value in the **Tapar Angle**field and hit the TAB key. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
5. Do one of the following:
   - Hit the TAB key and type another value in the **Taper Angle**field. The **Taper Angle** field is the complement of the inclination angle of the roof.
   - Press Enter to create the roof.
6. Depending on the shape of the roof outline, the result is:
   - Pyramid roof: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/R
   - L-shaped hip roof: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechnique
   - T-shaped hip roof: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechnique

Figure 2.135

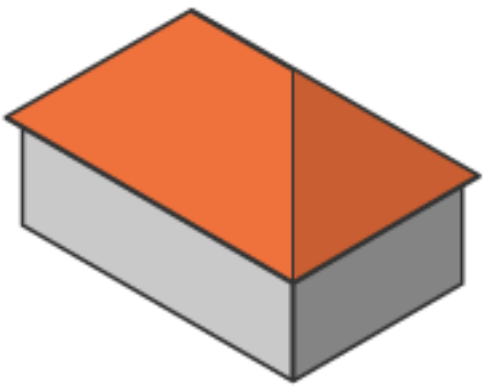


Figure 2.136

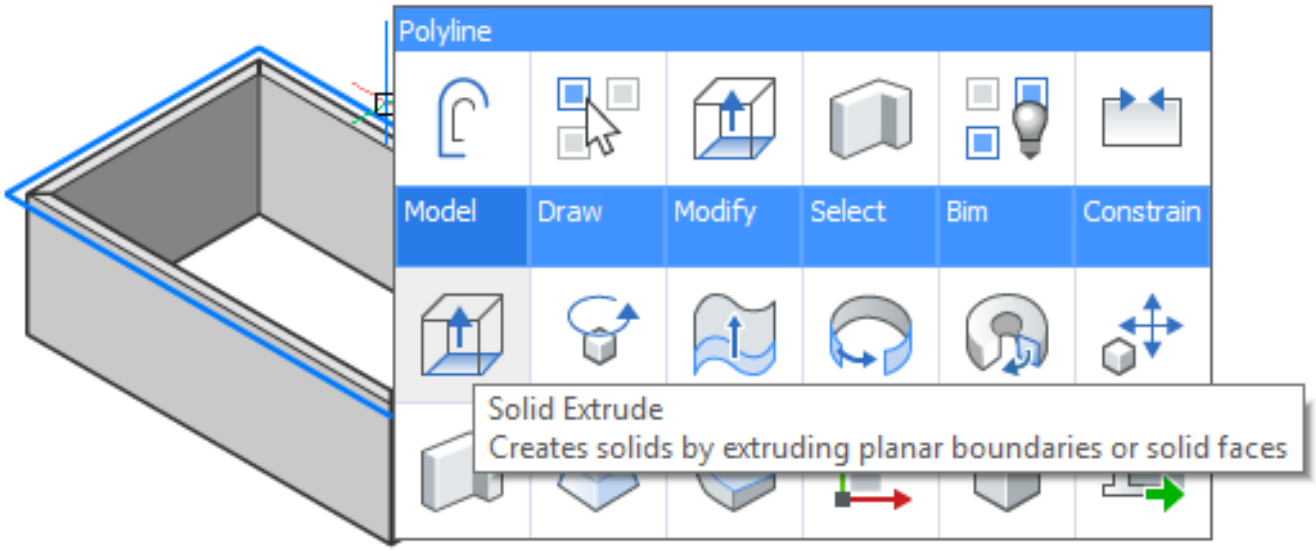


Figure 2.137

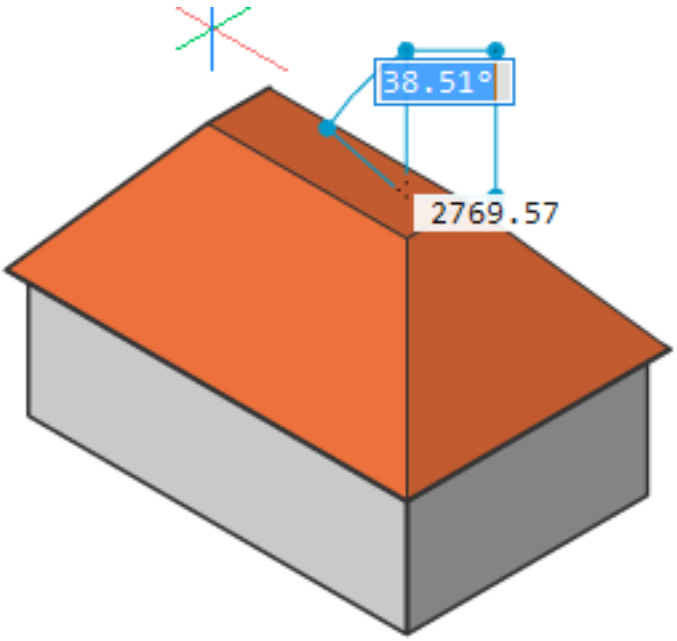


Figure 2.138

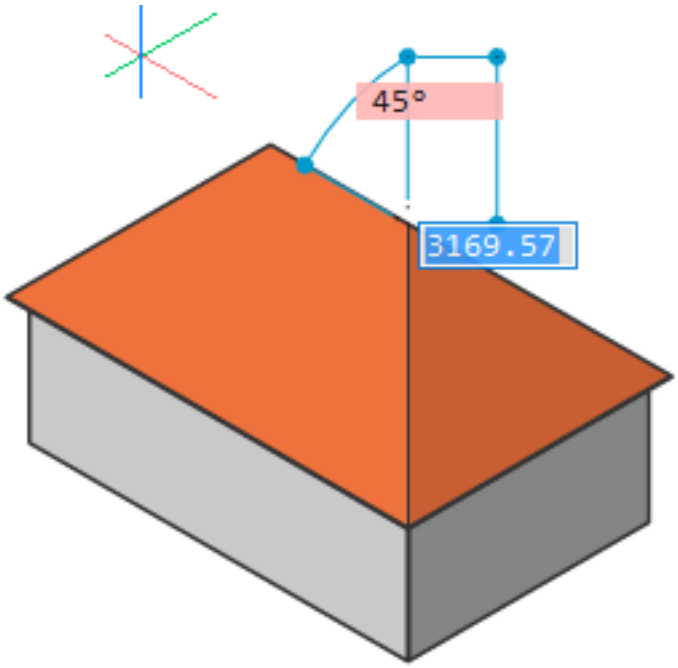


Figure 2.139

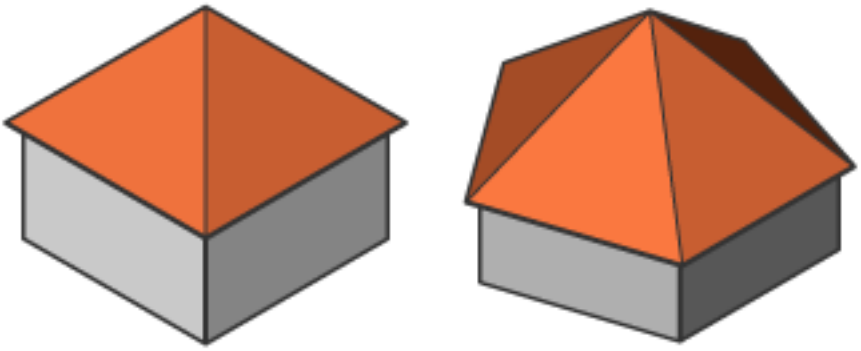
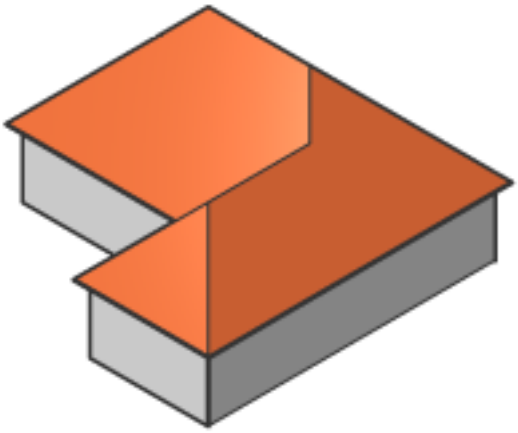


Figure 2.140



(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofHipT2.png}
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofHipT3.png}

- U-shaped hip roof: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechnique
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofHipU2.png}

- Complex hip roof: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechnique

7. To define the thickness of the roof solid we will use the **Body/Shell** option of the SolidEdit (6.27.39) command.


Do one of the following:

- Choose**Shell** in the **Solid Edit** |**3D Solid Editing**ribbon panel.
- Choose **Shell** in the **3D Solid Editing** toolbar.

- You are prompted:**Select 3d solid:** Select the roof solid.
- You are prompted:**Select face to remove or [Add/Undo/ALL]:** Select the bottom face of the roof solid.
- You are prompted: **Enter the shell offset distance:** Type a value to define the thickness of the roof solid.

Figure 2.141



Figure 2.142



Figure 2.143



Figure 2.144

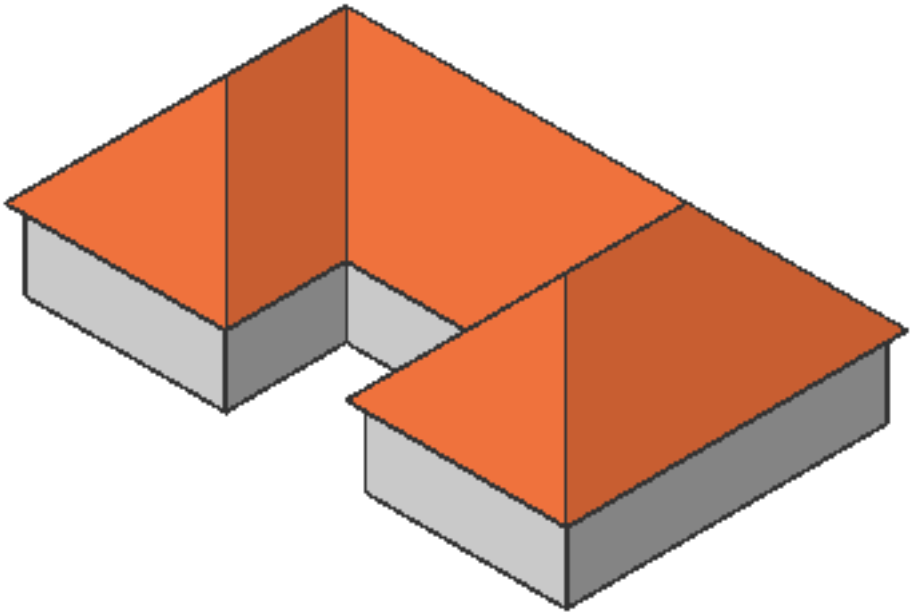


Figure 2.145

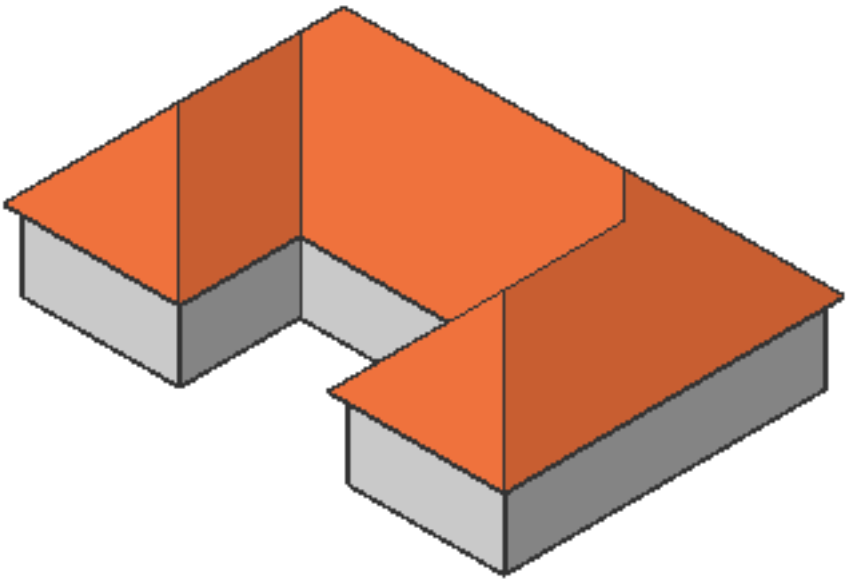
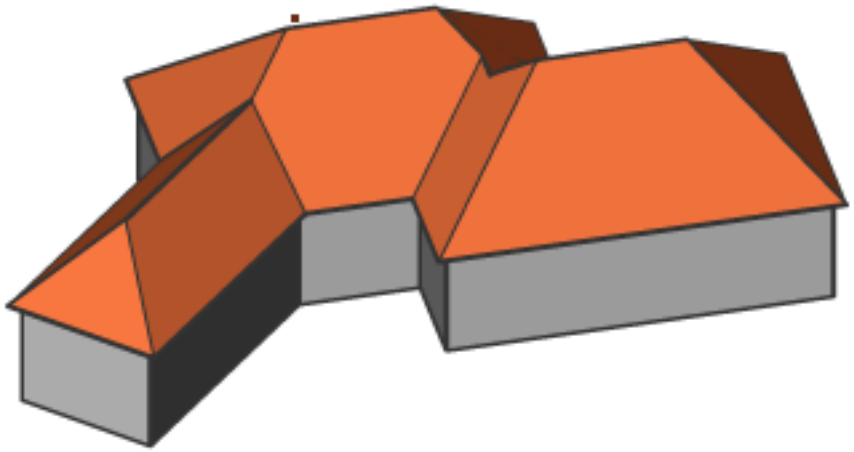


Figure 2.146



Creating a gable roof

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofGable.png})

1. Create a hip roof.
2. Make both end faces vertical:
   - Move over the end face then hold down the left mouse button about half a second; the Manipulater widget (1.23.10) displays (set the value of the MANIPULATOR system variable = 3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofRotateEnd1.png})
   - Place the cursor over vertical rotation arc. When the arc highlights, move the cursor. The selected face rotates dynamically. Click when the face is in a vertical position. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc
3. Execute step 7 of theCreate a hip roofprocedure. Also remove the vertical faces of the roof solid.

Figure 2.147

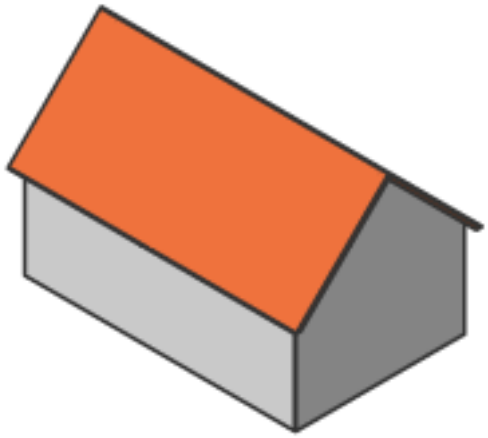


Figure 2.148

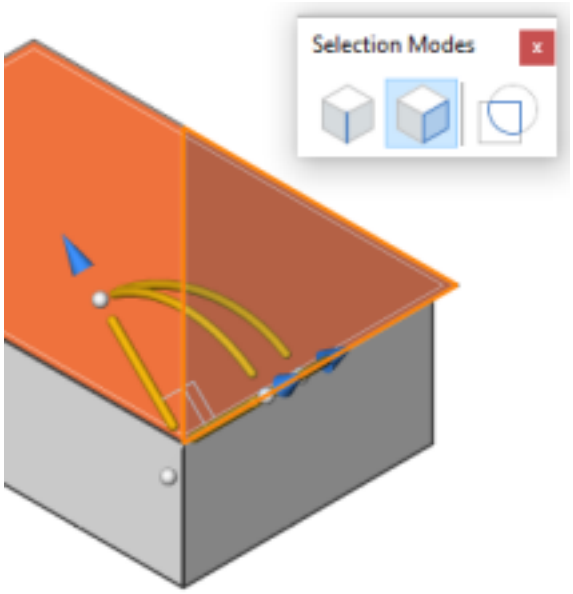
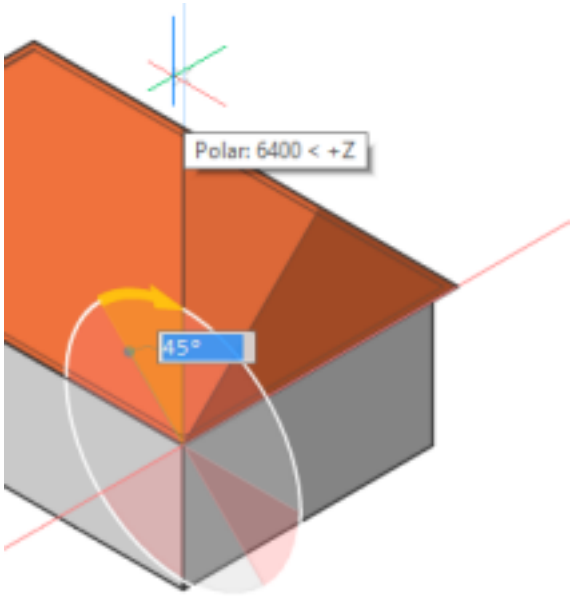


Figure 2.149



Creating a mansard roof

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofMansard.png})
A mansarde roof is composed of two solids, which we will create as two separate extrusions.

1. Create the roof footprint.
2. Move the cursor over the roof outline and choose **Extrude** in the **Model** tab of the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/HipRoofStep1.png})
3. Move the cursor vertically, then hit the TAB key.
4. Type a value in the **Taper Angle*** field then hit the TAB key. The **Taper Angle** field is the complement of the inclination angle of the roof. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniqu
5. Type a value in the extrusion **Height** field, then do one of the following:
   - Hit the TAB key and type another value in the **Taper Angle**field.
   - Press Enter to create the first solid.
6. Optionally hit the TAB key to go back and forth between the entry fields and press Enter when you are satisfied with the result.
7. Move the cursor of the top face of the first roof solid and choose **Extrude** in the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MansardeRoofStep2a.png})
8. Move the cursor vertically, then hit the TAB key.
9. Type a value in the **Taper Angle*** field then hit the TAB key. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
10. Do one of the following:
    - Hit the TAB key and type another value in the **Taper Angle**field.
    - Press Enter to create the first solid.
11. Select both solids and choose **Union** in the **Model** tab in the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom
12. Repeat step 7 of theCreate a hip roof procedure.

Figure 2.150

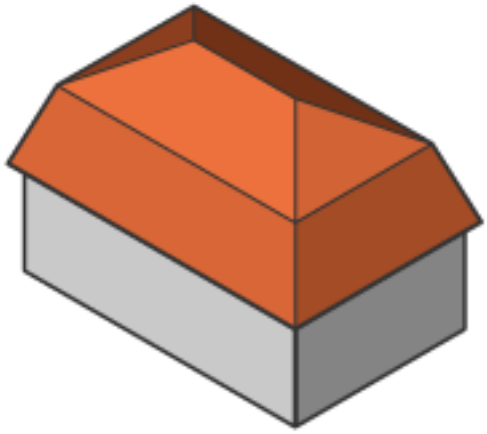


Figure 2.151

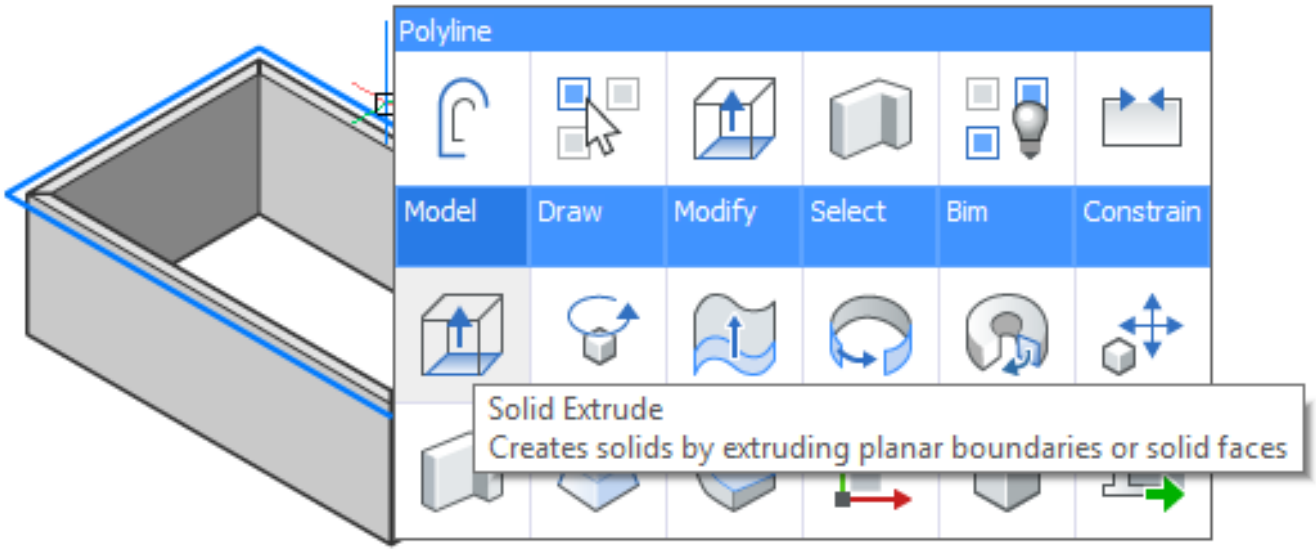


Figure 2.152

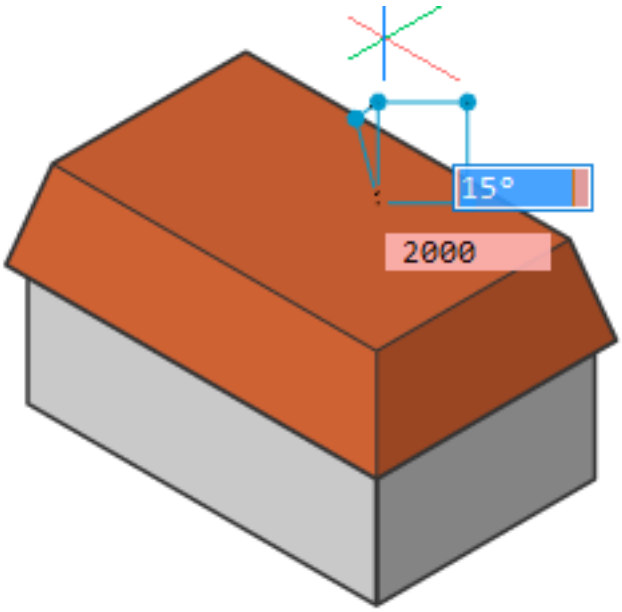


Figure 2.153

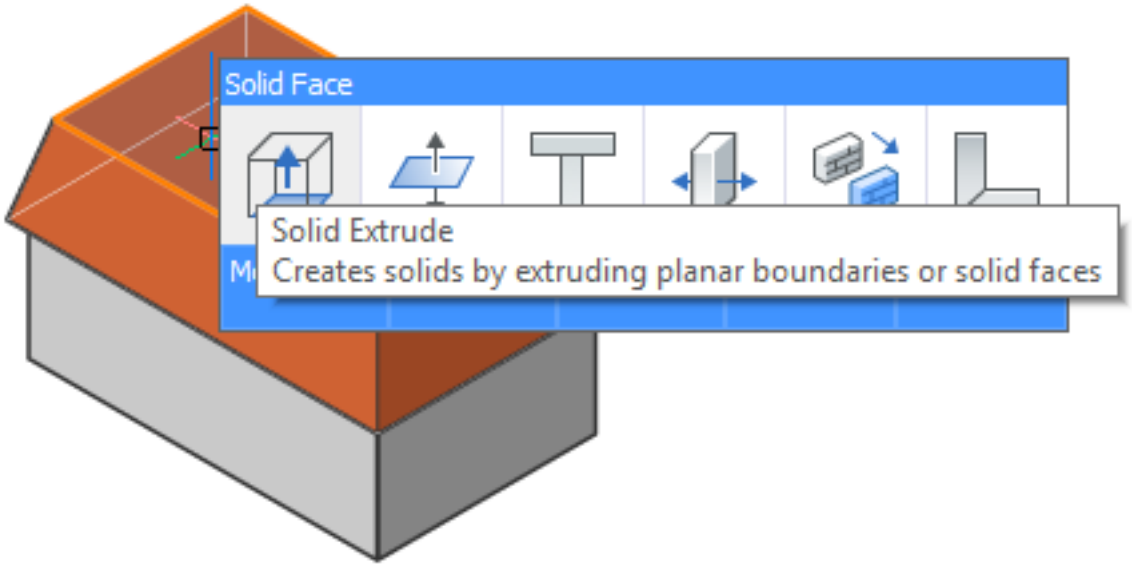


Figure 2.154

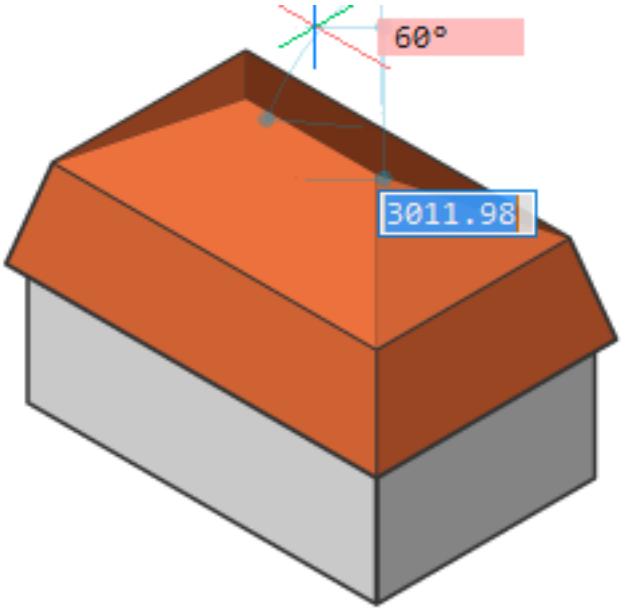
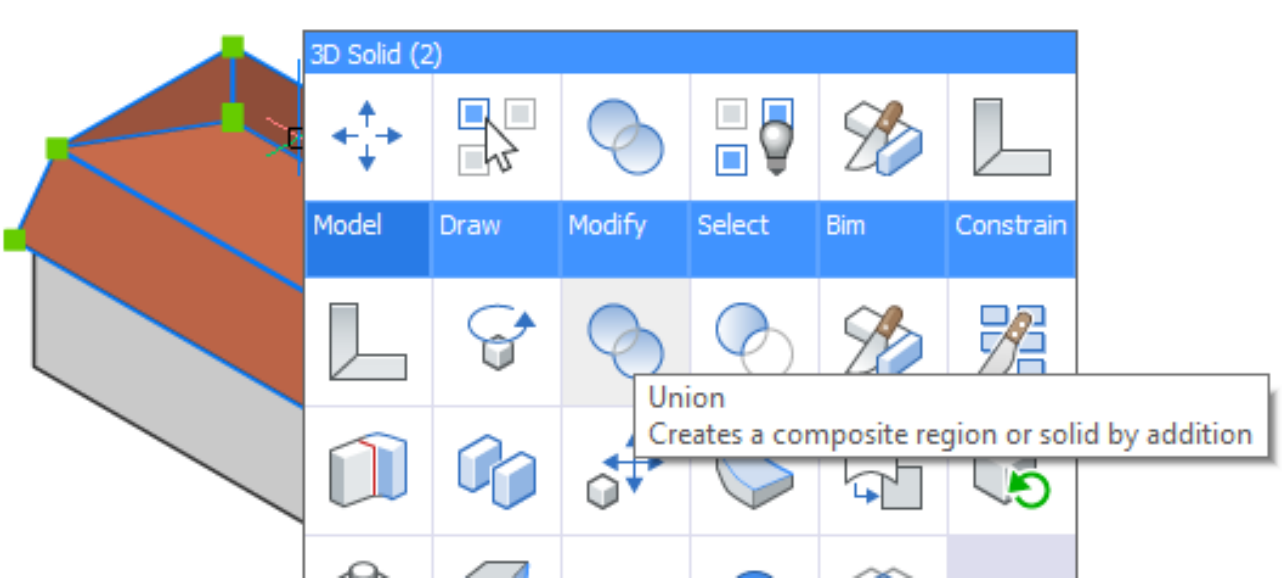


Figure 2.155

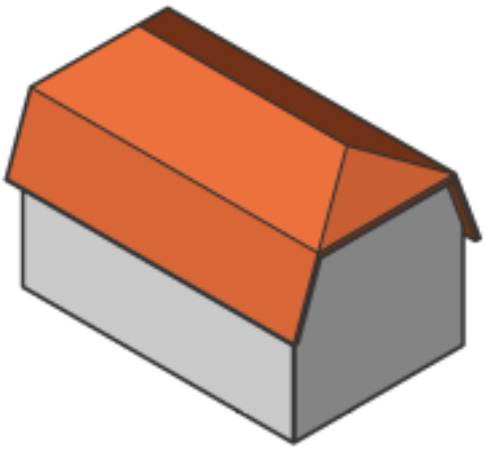


Creating a jerkinhead roof

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofJerkinhead.png}) A jerkinhead roof is composed of two solids, which we will create as two separate extrusions.

1. Execute steps 1 to 6 of the Creating a mansard roof procedure, to create the first solid.
2. Execute step 2 of the Creating a gable roof procedure to set the end faces vertical.
3. Execute steps 7 to 11 ofthe Creating a mansard roof procedure.
4. Execute step 7 of theCreate a hip roofprocedure. Also remove the vertical faces of the roof solid.

Figure 2.156

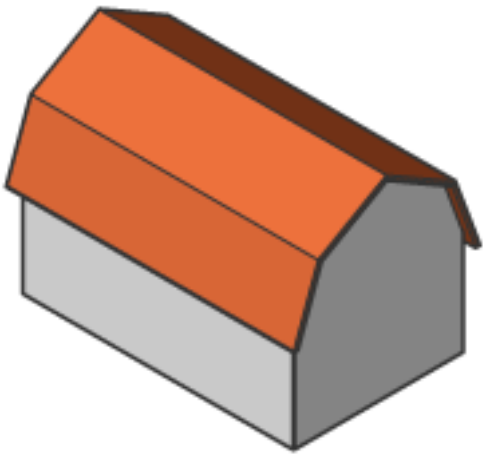


Creating a gambrel roof

A gambrel roof is composed of two solids, which we will create as two separate extrusions. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofGambrel.png})

1. Execute steps 1 to 6 of the Creating a mansard roof procedure, to create the first solid.
2. Execute step 2 of the Creating a gable roof procedure to set the end faces vertical.
3. Execute steps 7 to 10 ofthe Creating a mansard roof procedure to create the second solid.
4. Execute step 2 of the Creating a gable roof procedure to set the end faces vertical.
5. Execute step 11 ofthe Creating a mansard roof procedure.
6. Execute step 7 of theCreate a hip roofprocedure. Also remove the vertical faces of the roof solid.

Figure 2.157



Creating a dutch gable roof

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofDutchGable.png}) A dutch gable roof is based on a hip roof.

1. Create a hip roof.
2. Set the **Select Edges** option of the **Selection Modes** system variable ON.
3. Move the cursor over the bottom edge of the roof face, then choose **Offset** in the **Draw** tab in the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/DutchGableStep1.png})
4. Move the cursor and type a value in the dynamic distance field or click a point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/DutchGableStep2.png})
5. Do one of the following: 
   - Choose **Imprint** in the **3D Solid Editing** toolbar
   - Choose Imprint in the **Solid Edit** | **Solid Editing** ribbon panel.
6. You are prompted:**Select 3d solid, surface or region:** Select the roof solid.
7. You are prompted:**Select an entity to imprint:** Select the offset roof edge.
8. You are prompted:**Delete the source object[Yes/No] <No>:** Choose Yes. A triangular face is created
9. Execute step 2 of the Creating a gable roof procedure to set the triangular face vertical. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/DutchGableStep3.png})

Figure 2.158

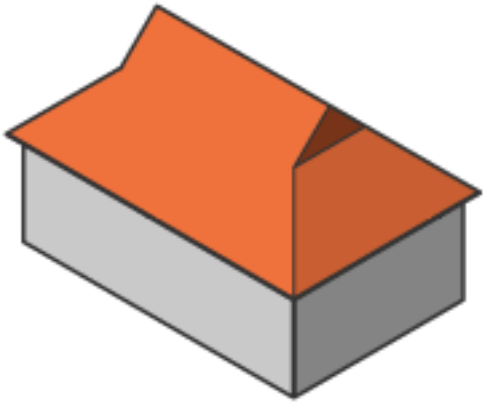


Figure 2.159

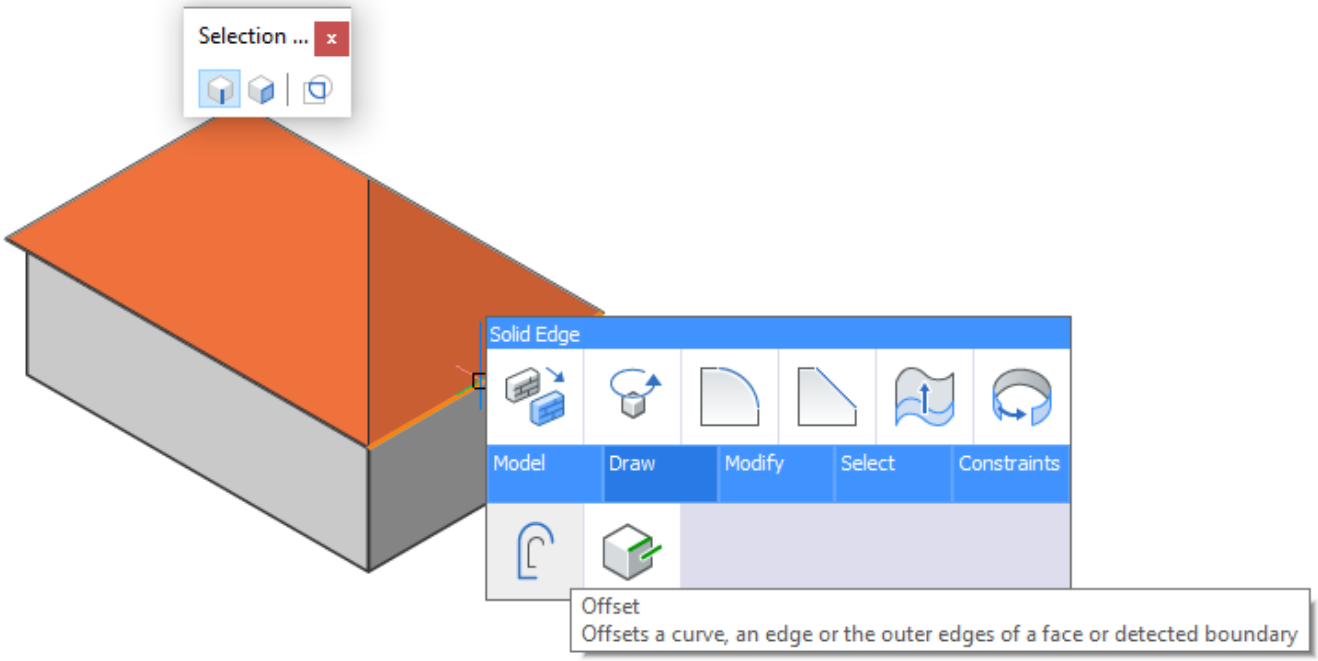
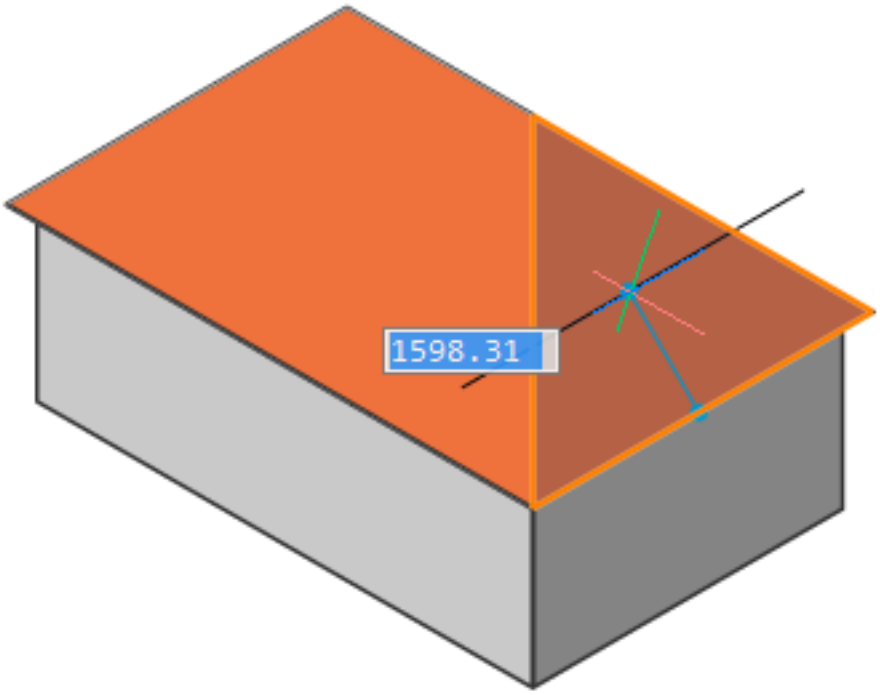


Figure 2.160



(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/DutchGableStep4.pn

10. Execute step 7 of the of theCreate a hip roofprocedure to set the thickness of the roof solid.

Figure 2.161

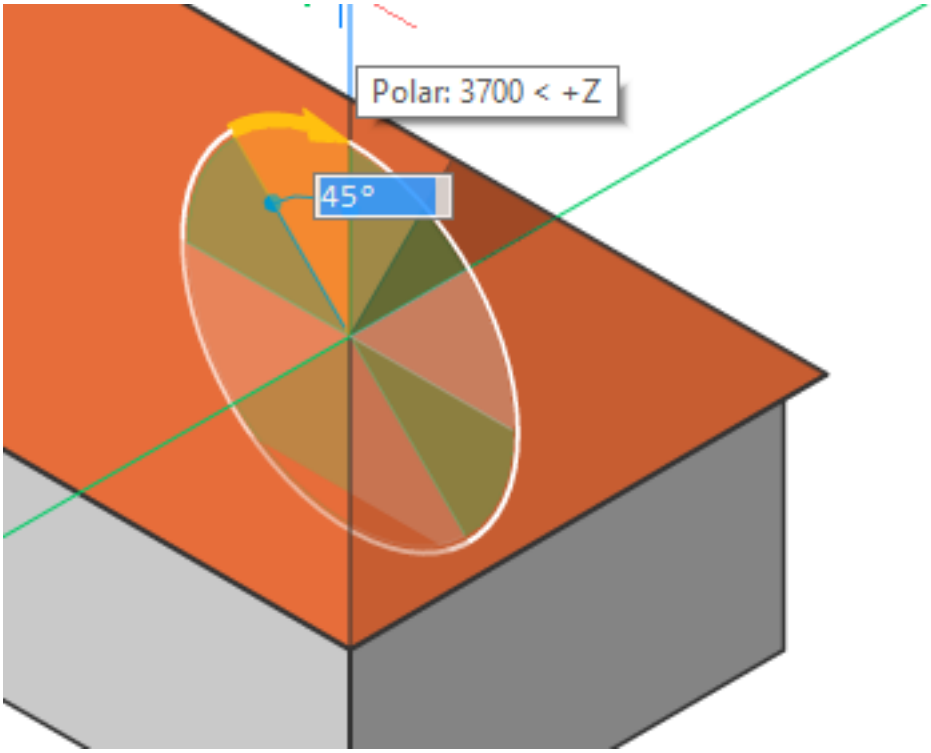
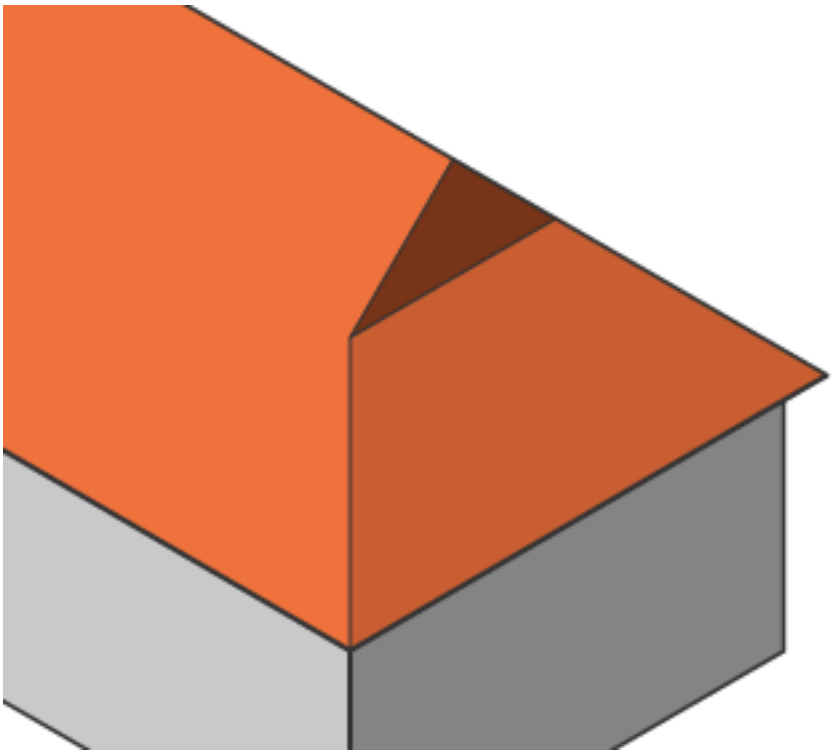


Figure 2.162

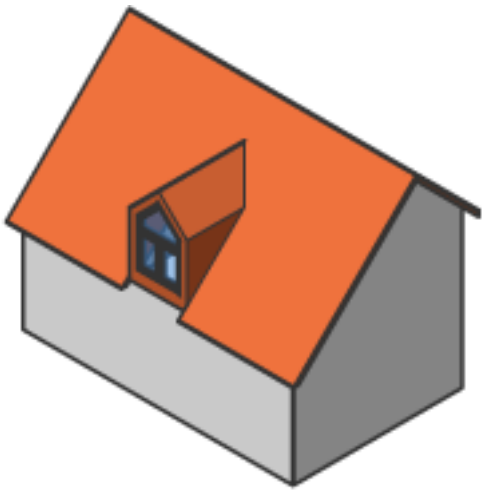


Creating dormers

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofDormer.png})
The procedure to create a cormer consists of the following steps:

- Create a box solid at the bottom edge of the roof plane and connect the box to the roof.
- Define the position and dimensions of the dormer.
- Create the dormer roof.

Figure 2.163



Create a box solid

1. Right click, then choose **Box** in the **Model** tab of the Quad. You are prompted: **Set corner of box or [Center] <0.00,0.00,0.00>:**
2. Click a point at the bottom edge of the roof plan. You are prompted: **Set opposite corner or [Cube/Length of side]:**
3. Click a point, then define the height of the cube: either click a point or type value. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofDormer1.png})
4. Select the back face of the cube: move the cursor over the cube at the location of the back face, then hit the Tab key until the face highlights (Selection Modes: Select Faces = ON.
5. Choose Connect with nearestin the Model tab in the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofDormer4.png})
6. Select the front face of the box, choose **PushPull**in the **Model** tab in the Quad and move the face to the roof plane.
7. Use**PushPull**on the front, top and side faces of the dormer solid to define the dimensions and the position of the dormer on the roof:
   - Select a side face, then hit the TAB key to define the width. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h
   - Select both side faces, then choose **Add rigid set** in the **Contraints** tab in the Quad to fix the width. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RoofDormerWidthRigi
   - Drag one of the side faces to position the dormer. Do one of the following:
     - Type a distance in the dynamic entry field.
     - Click a point.
     - Move the cursor over a face which is parallel to the face being dragged and hit the TAB key. The distance field now shows the distance to this face.
   - Drag the top face to define the height of the dormer.
   - Drag the front face to set the distance from the roof eave. Move the cursor over the exterior face of the wall beneath the dormer, then hit the TAB key. The distance field now shows the distance to this face.
8. Modify the shape of the dormer:
   - Add a hip roof.
   - Add a hip roof, then rotate the front face vertical to create a gable roof.
   - Add a vault.
   - Rotate top and side faces.
   - Starting from a gable roof move the front face of the dormer backwards until the front face is triangular.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e Hip	(\vref {fig:C:/Dev/Task- Flat	(\vref {fig:C:/Dev/Task- Vault
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e Special	(\vref {fig:C:/Dev/Task- Triangle	(\vref {fig:C:/Dev/Task- Eyebrow

Figure 2.164

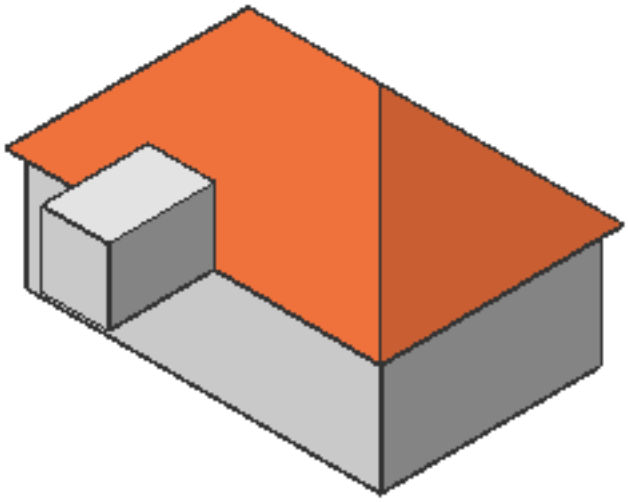


Figure 2.165

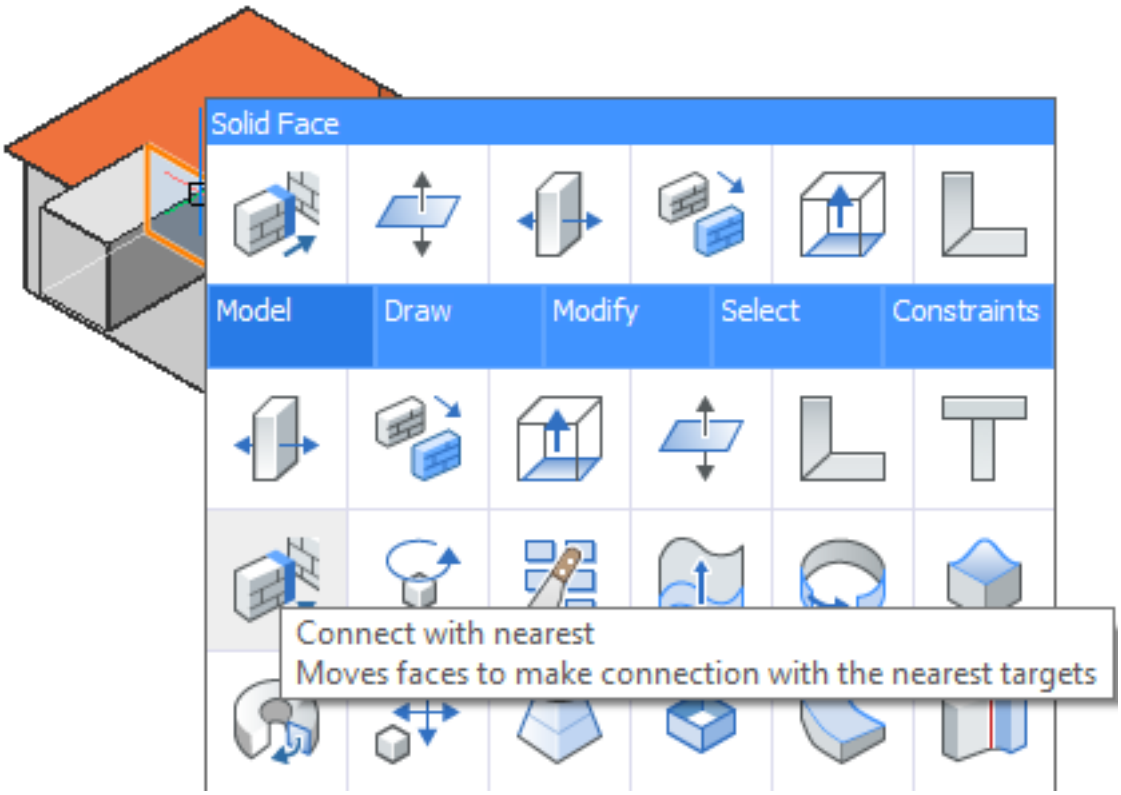


Figure 2.166

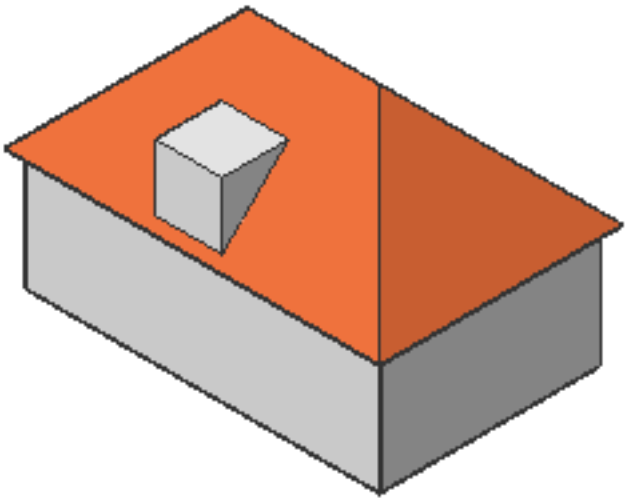


Figure 2.167

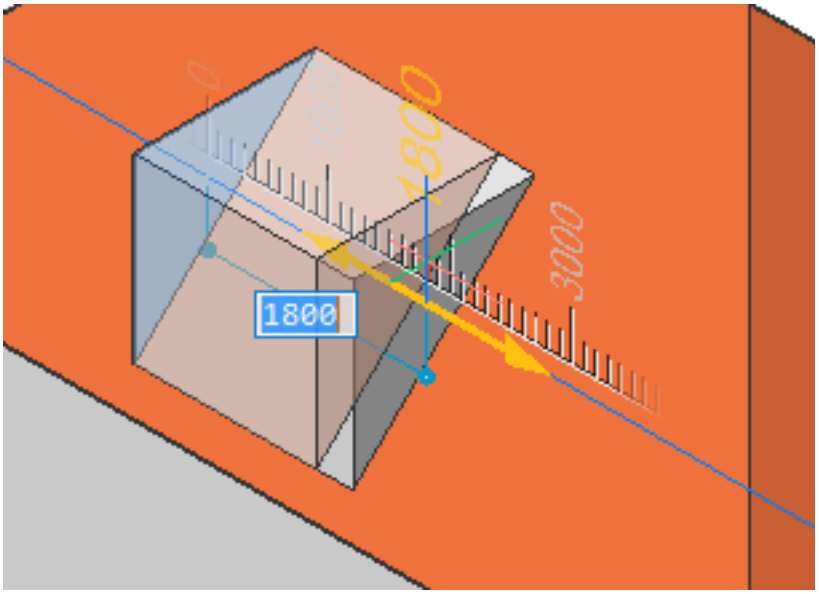


Figure 2.168

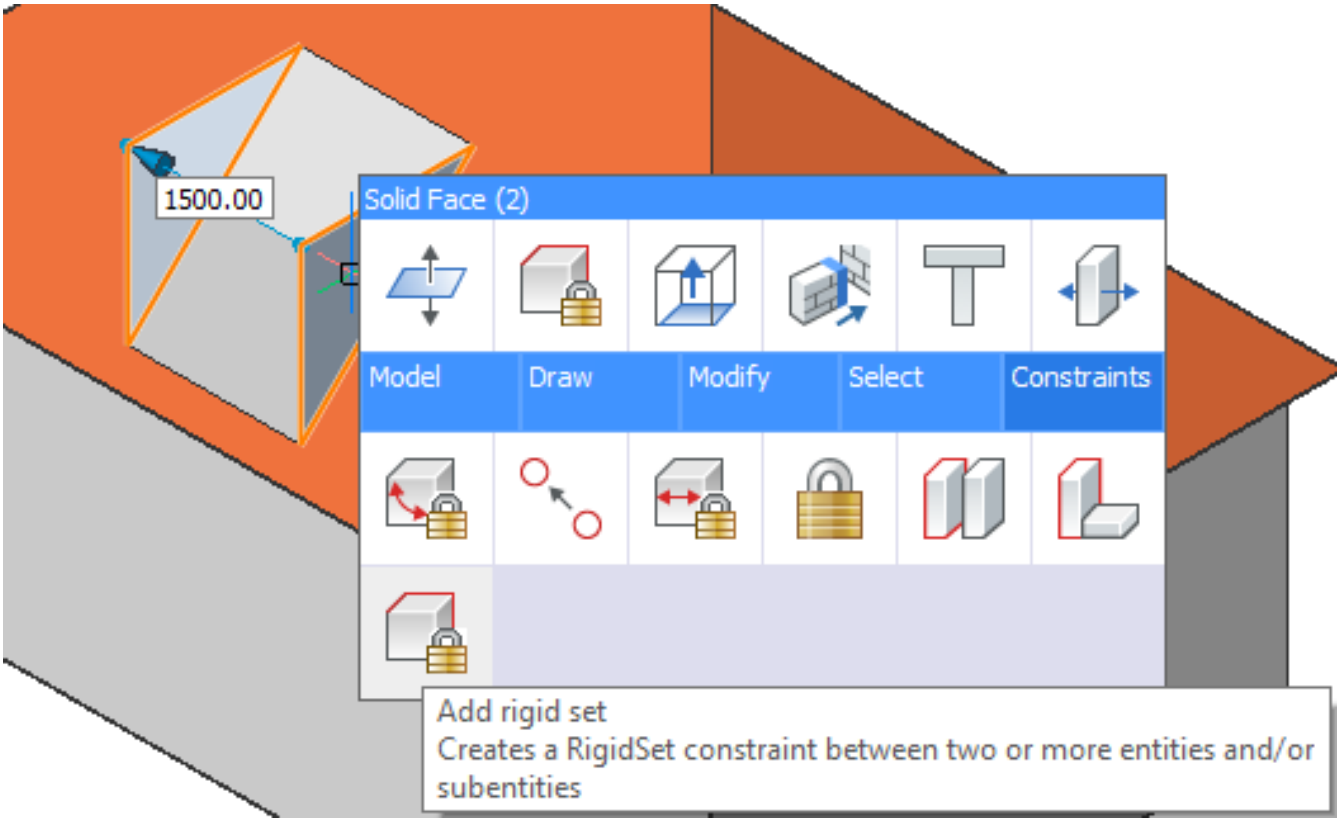


Figure 2.169

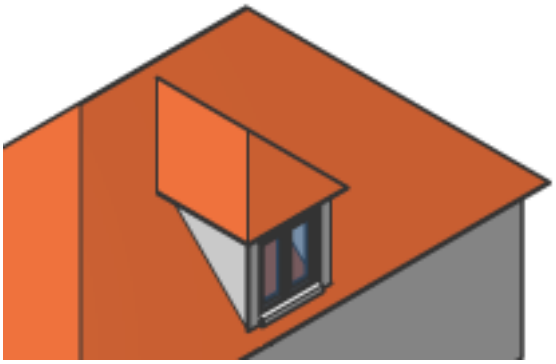


Figure 2.170

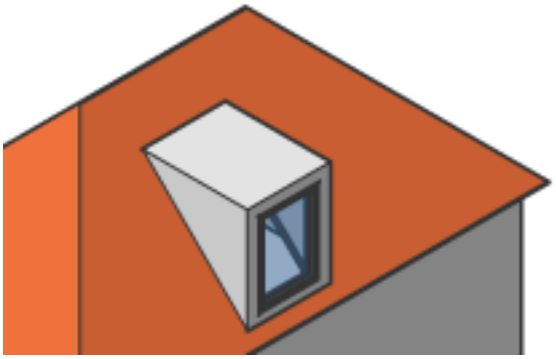


Figure 2.171

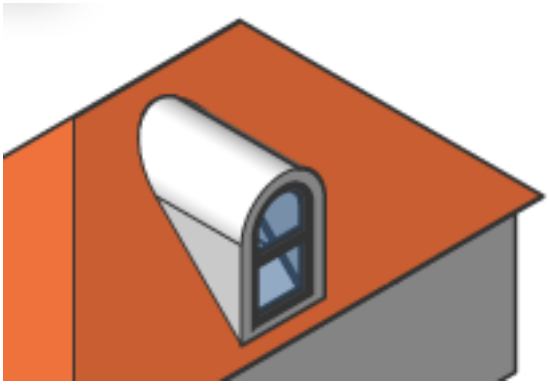


Figure 2.172

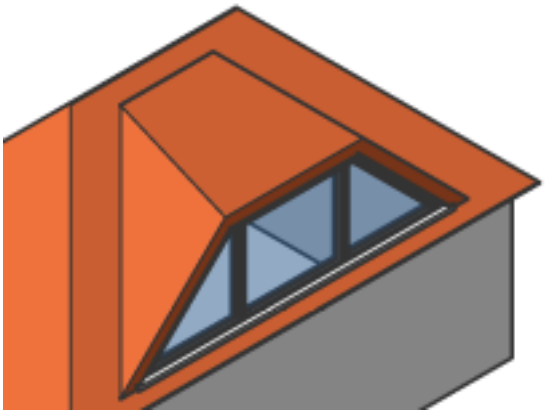


Figure 2.173

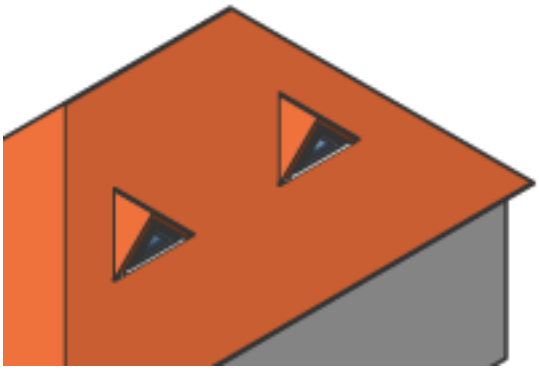
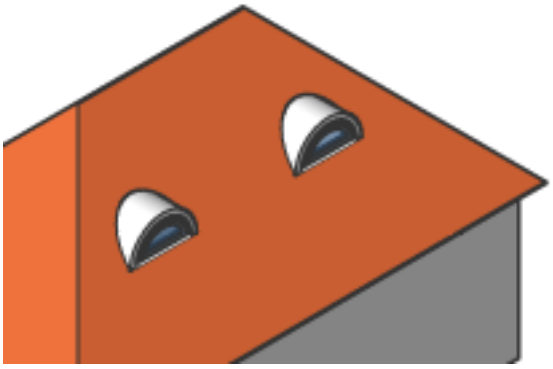


Figure 2.174



2.4.8 Creating Slabs

Commands: DmExtrude (6.12.13),Polysolid (6.24.44) TheDmExtrudecommand allows to extrude a closed 2D entity such as polylines, circles, ellipses and extrude a detected boundary, enclosed by walls and/or 2D entities. TheP-olySolidcommand allows to create pitched roofs.

Creating floor slabs between walls

Make sure that:

- TheEnable Boundary Detectionoption of theSelectionmodes (1.23.5)system variable is ON.
- The bottom faces of the surrounding walls are in the XY-plane of the current coordinate system. If neces-sarymove the origin of the coordinate system (1.15.18).
- Dynamic dimensions (1.15.9)are ON.

1. Move the cursor inside the surrounding walls.
2. When the boundary highlights, chooseExtrudein theModelcommand group in the Quad. The slab is extruded dynamically.
3. Do one of the following:

- Type the thickness of the slab in the dynamic dimension field.
- Pick a point.

Creating a pitched roof slab

Make sure that:

- Dynamic UCS (1.15.18)is ON.
- Dynamic dimensions (1.15.9)(DYN) is ON; it is recommended to have theTracking dynamic dimensionsoption of theDYNMODEsystem variable set.
- Entity Snap Tracking (1.15.17)(STRACK) is ON.
- Endpoint (6.13.7)entity snapis ON.

1. Launch thePolysolid (6.24.44)command. **You are prompted:** Start point or [Height/Width/Justification/Entity/Separate solids/Dynamic] <Entity>:
2. Move the cursor over the outside face of a wall which is perpendicular to the roof slab you want to create.
3. When the face highlights, hit the Shift key to lock the dynamic UCS.
4. Do one of the following: To create the roof at the top edge of the wall, snap to the exterior top corner of the wall. To create a roof overhang (eave):
   - Move the cursor over the exterior top corner of the wall to require a snap tracking point.
   - When a small red cross indicates the tracking point is acquired, move the cursor to the outside and type the overhang distance in the dynamic entry field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS
5. Hit the TAB key to activate the Angle field, then type the pitch angle and hit the TAB key again to jump to the Dis-tance field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/CreateRo
6. Specify the height of the roof slab, then press Enter twice: the first Enter creates the footprint of the roof slab, the second Enter interrupts the creation of a second solid. The roof slab is extruded dynamically.
7. Specify a point to define the length of the extrusion. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
8. Repeat the previous steps to create more roof slabs.
9. Use ConnectWithNearest to connect the walls to the roof slabs.

Figure 2.175

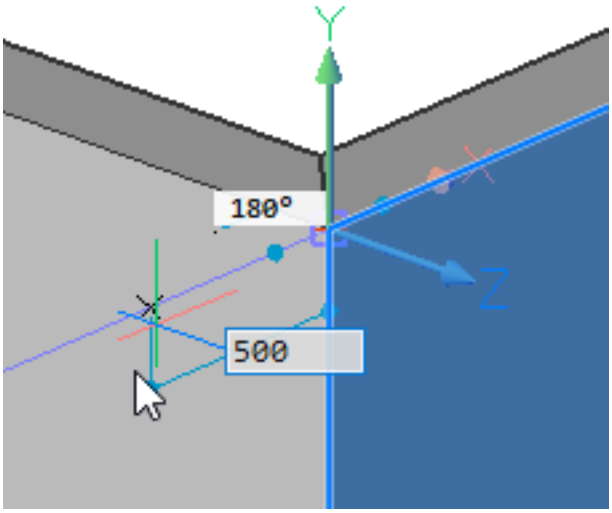


Figure 2.176

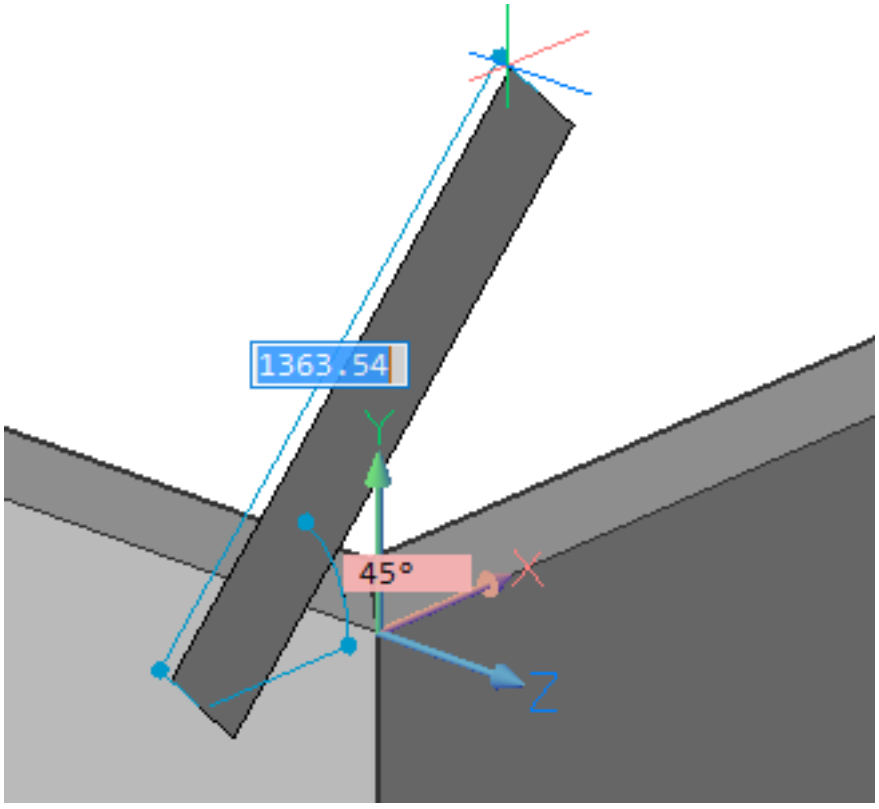
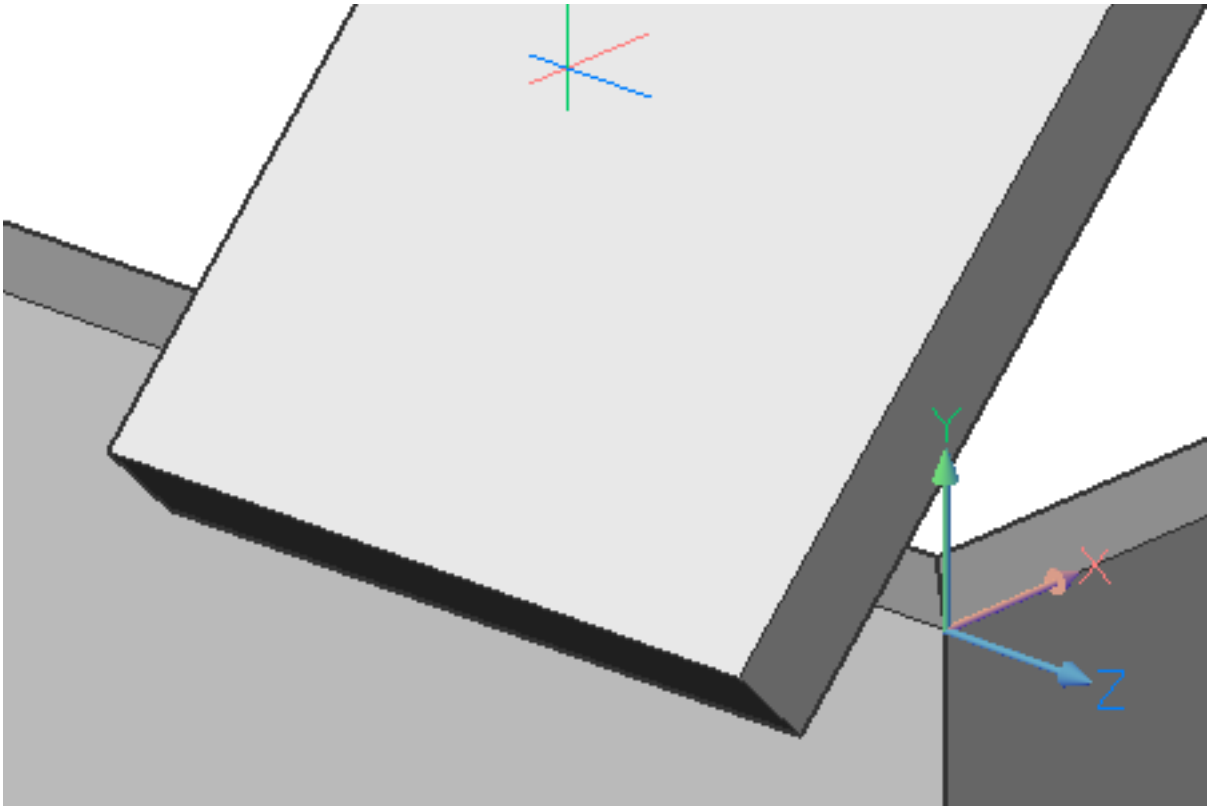


Figure 2.177



2.4.9 Creating Walls

Overview: In this lesson, you learn how to create and modify a basic wall element. **Lesson Objectives** After completing this lesson, you will be able to:

- Use the Polysolid tool to create a basic wall.
- Create walls using an existing 2D plan.
- Create a connection between the two walls.
- Connect walls to other elements.

Tutorial: How to create walls Click here (2.3.3) to watch

About: Polysolid Tool

The Polysolid tool lets you create wall solids by selecting a start point, an endpoint and providing a height. Note that in BricsCAD BIM, we start by creating the geometry first, and only later adding materials or compositions. That way we don't have to worry about details when we're modeling our preliminary design. When drawing walls it is recommended to useDynamic Dimensions (1.15.9)to specify the length of the wall andPolar Tracking (1.15.16)to control the direction.

Access: Polysolid Tool



Menubar: **Model tab > 3D Solids menu >Polysolid** Toolbar: **3DSolids > Polysolid** Quad: **Model tab > Polysolid** Keyboard: **POLYSOLID, PSO** Ribbon: **Model tab > Create 3D menu > Polysolid**

Procedure: Creating a simple wall

The following are general steps to create a wall element using the Polysolid tool.

1. Launch the Polysolid tool.
2. Select a start point, the footprint (1) of the wall displays automatically.

Move the cursor in the desired direction, the current length value displays in the length dynamic entry field (2). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360020244994/mceclip0.png})

3. Select an endpoint or enter a value in the "dynamic input field" and press enter. The footprint (3) of the adjacent wall displays automatically. You can keep creating adjoining wall segments by clicking the next points or entering values in the dynamic input field.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360020245134/2018-12-1812-32-06.png})

Figure 2.178

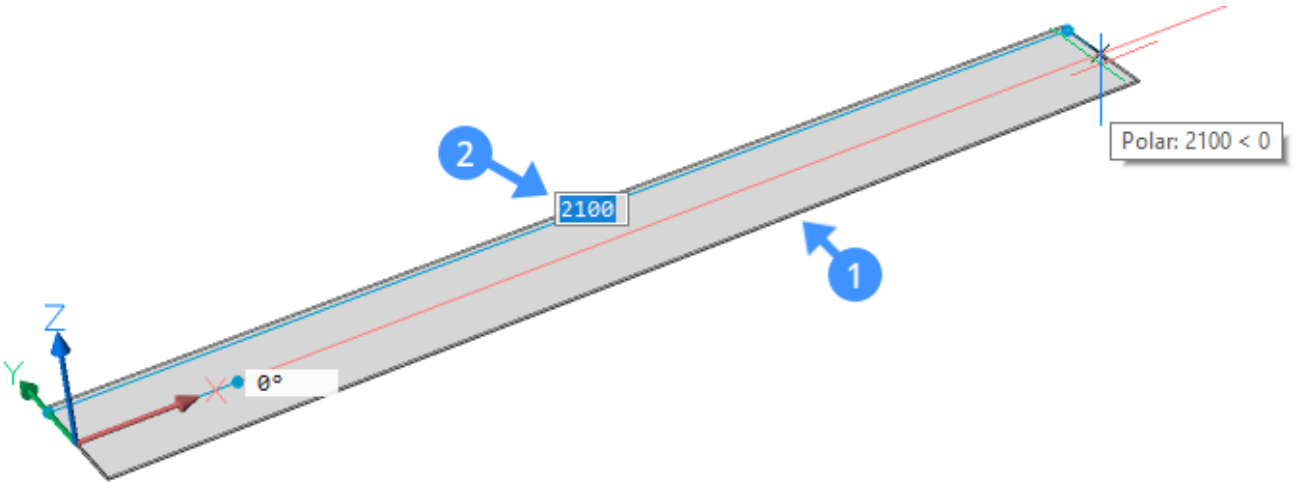


Figure 2.179

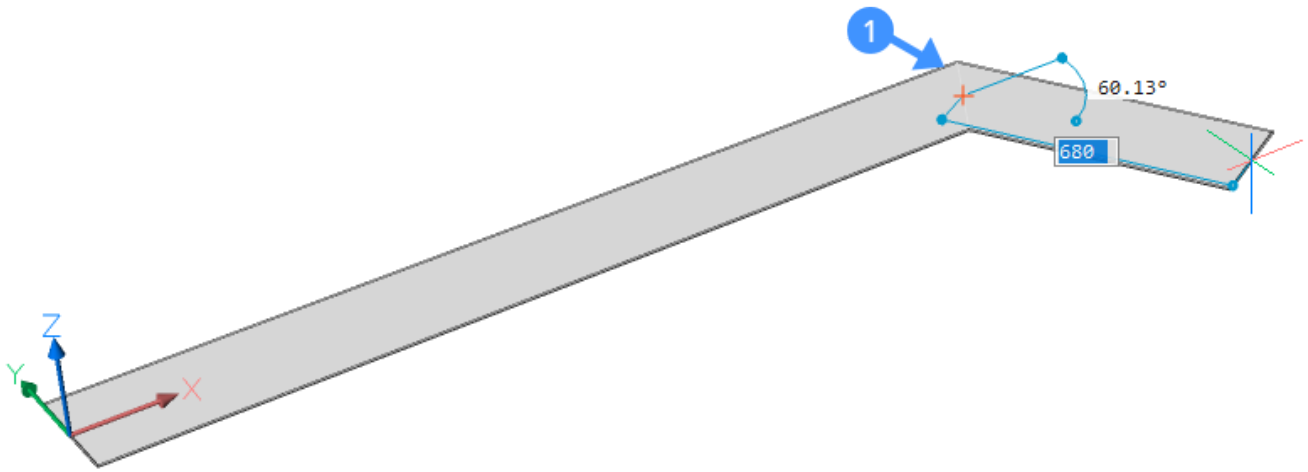
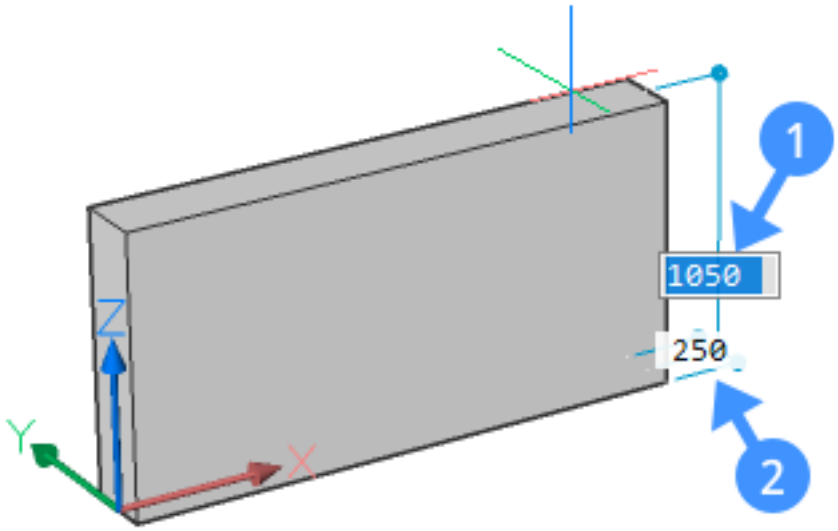


Figure 2.180



4. Press enter once more or right-click. The height of the wall displays dynamically in the height field (4). Herein, you can change the default height of the wall. To change the width of the wall use the width field (5) by pressing TAB once. You can switch between the two dynamic input fields by pressing TAB.

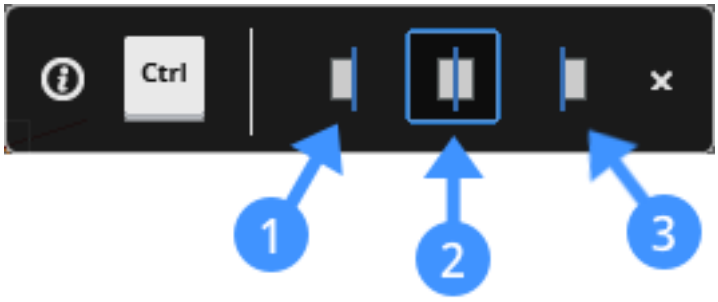
(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360020245214/2018-12-1812-35-01.png})

5. To change the justification of the wall hit the CTRL key while the 'Polysolid justification Hotkey Assistant widget' appears at the bottom of the screen.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009348654/mceclip5.png}) Note that the width and height of the wall that was used previously will be the new default values of the next Polysolid.

Tutorial: How to create wall layouts Click here (2.3.4) to watch

Figure 2.181



Procedure: Using anexisting 2D plan to create walls

Using an existing 2D plan view enables you to create a wall element in 3D.

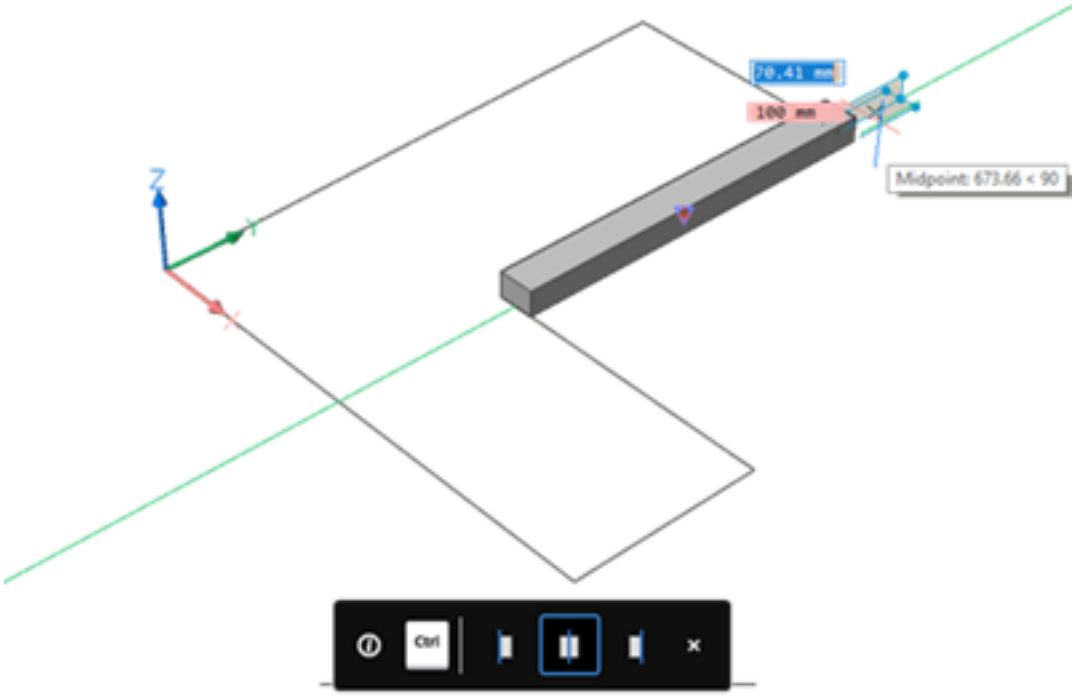
1. Click the “Polysolid” tool from the quad and type “E” on the command line or select “Entity” in the prompt menu. Using the “Entity” option in Polysolid enables you to create 3D geometry from a 2D layout.
2. Choose a 2D linear entity which will be the Polysolid base.

Lines, open and closed polylines, arcs, circles, ellipses, elliptical arcs, and splines are accepted as a polysolids base.

3. Move the solid up or down and left-click to set the height of the wall, enter a value in the dynamic input field, or right-click to accept the default value.
4. The current width of the wall appears in the **Width Field**. Press the Tab key to change the default value of the width.
5. The **Hotkey Assistant** appears to remind the possible options to set the justification. Press CTRL key to cycle through the wall justification options.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009401833/mceclip6.png})

Figure 2.182



About: BimConnect Tools

The BimConnect tools enable you to connect one wall to another element.

Access: LConnect Tool

Menubar: **BIM tab > BimConnect > LConnect** Quad: **Model tab > LConnect** Keyboard: **LCONNECT** Ribbon: **Model tab > Modify 3D menu > Connect > LConnect**

Tutorial: How to connect walls Click here (2.3.5) to watch

Procedure: Connecting two walls

1. Select two wall elements and apply the LConnect tool. When two walls intersect, a miter connection between two walls will be automatically created by default. The following illustration shows the wall elements before (9) and after (10) applying the LConnect tool. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009349194 (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009402193/mceclip9.png})

2. The layout of the wall connections can be changed by using the 'LConnect HotKeyAssistant widget'.

This widget appears at the bottom of the screen with the available options after you apply the LConnect tool to the selected walls and HKA is turned on in the status-bar. The wall connection can be made tobisector L-connection (11), L parallel type (12) or L parallel type 2 (13) accordingly by pressing the CTRL key. The selected connection type is indicated by a blue frame. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360021158693/201 12-1812-57-20.png}) When the bisector L-connection is selected, the walls are connected to an end face in the other wall in an angle 45^0. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009349774/mceclip13.
Bisector L-connection (miter) When the connection is made between the two walls selecting the L parallel type 1 or type 2, the walls are connected to an end face in the other wall in an angle 90^0. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009402633/mceclip11.png})

Figure 2.183

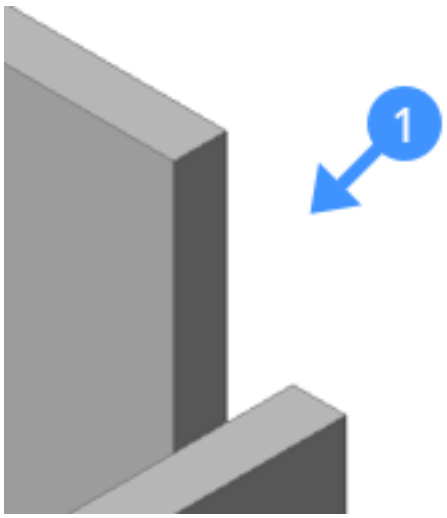


Figure 2.184

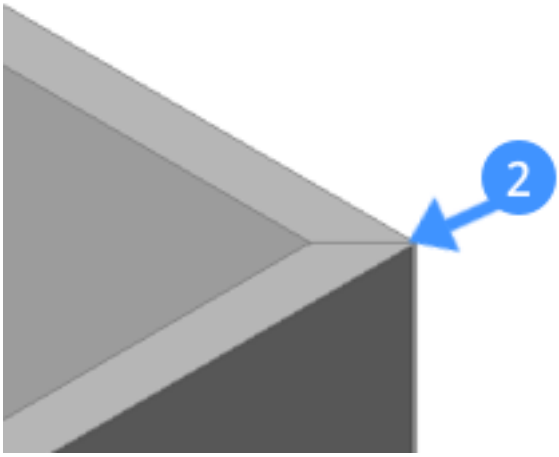


Figure 2.185

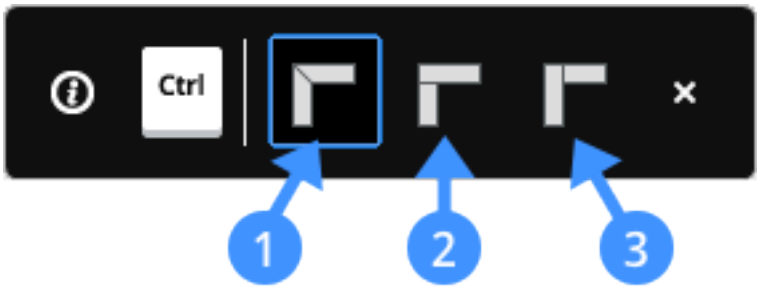
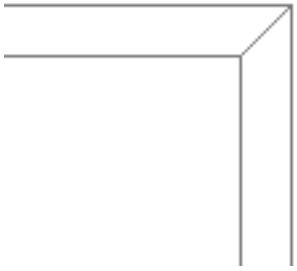


Figure 2.186



(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360009349734/mceclip12.png})
Parallel L-connection 1 Parallel L-connection 2 Press Enter key to accept the current connection type or press CTRL key to cycle through the possible connection options.

Figure 2.187



Figure 2.188



Connecting walls to other elements The connect with nearest tool allows you to connect a minor face of a wall to the major faces of other elements, regardless what the orientation of the faces is. This tool can be used to connect the top or bottom face of a wall to a roof or a floor slab or to connect the side faces of a slab to walls. **Note:** Before using the connect with nearest tool, make sure the Select Faces option is enabled in the selection mode settings.

Access: Connect with nearest tool

 Quad: **Model tab > Connect with nearest**

Procedure: Creating T- connections between two walls

Before using the connect with nearest tool, make the Select Faces option is enabled in the selection mode settings.

1. In the drawing area, highlight a face or faces that need to be connected to another wall (14).

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360020251814/mceclip6.png})

2. Use Connect with nearest tool from the quad.

The connect with nearest tool automatically makes a T-connection (15) between the two walls by moving the face of the first wall to the nearest face of the other wall. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/
About: Select Aligned Faces Tool The connect with nearest tool can also be used to connect the multiple faces of a wall to the nearest face of another element. To start with selecting multiple faces of the building element, use **Select aligned faces** tool. This tool enables you to highlight all faces automatically which are coplanar with the highlighted face.

Figure 2.189

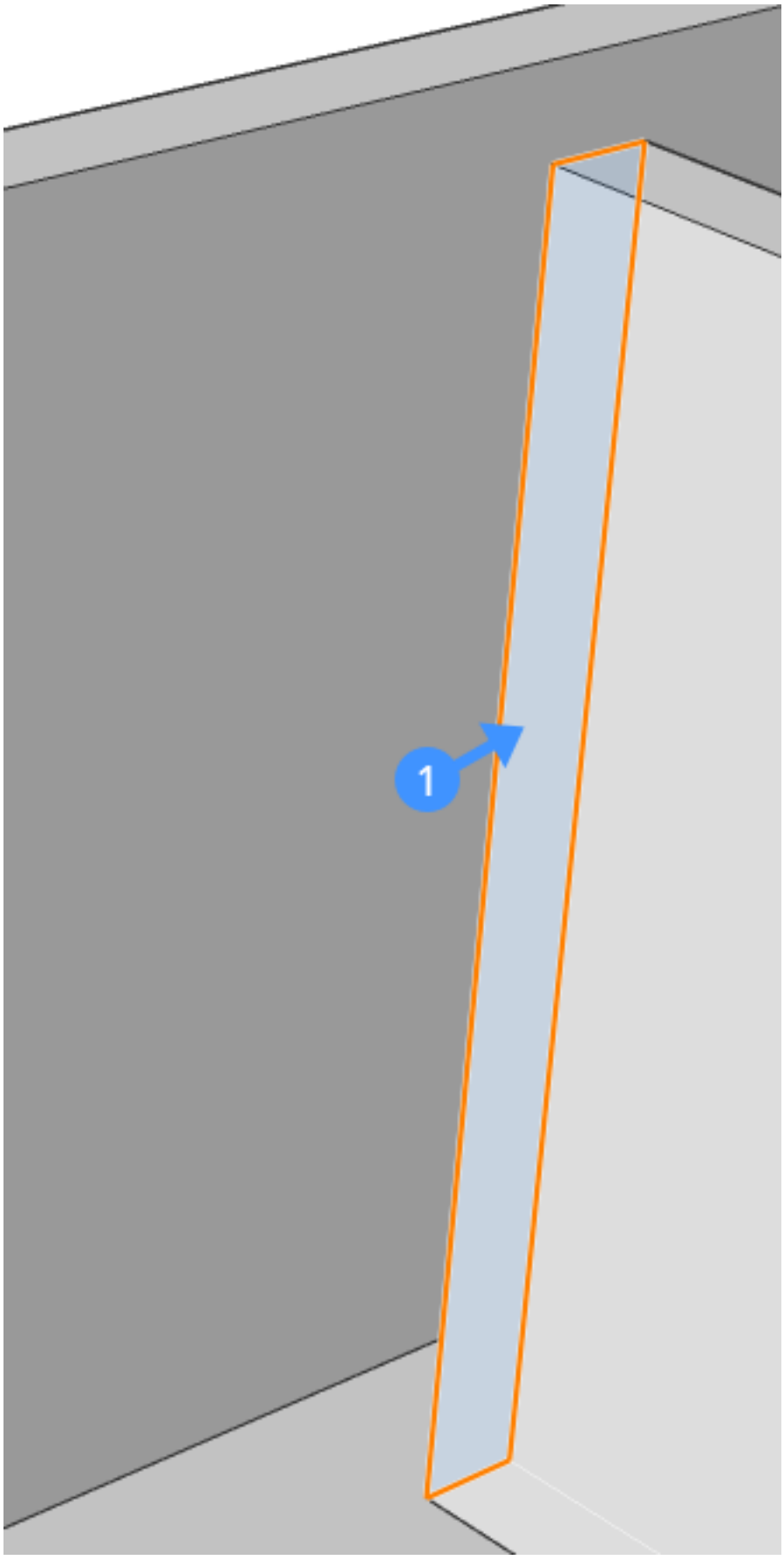
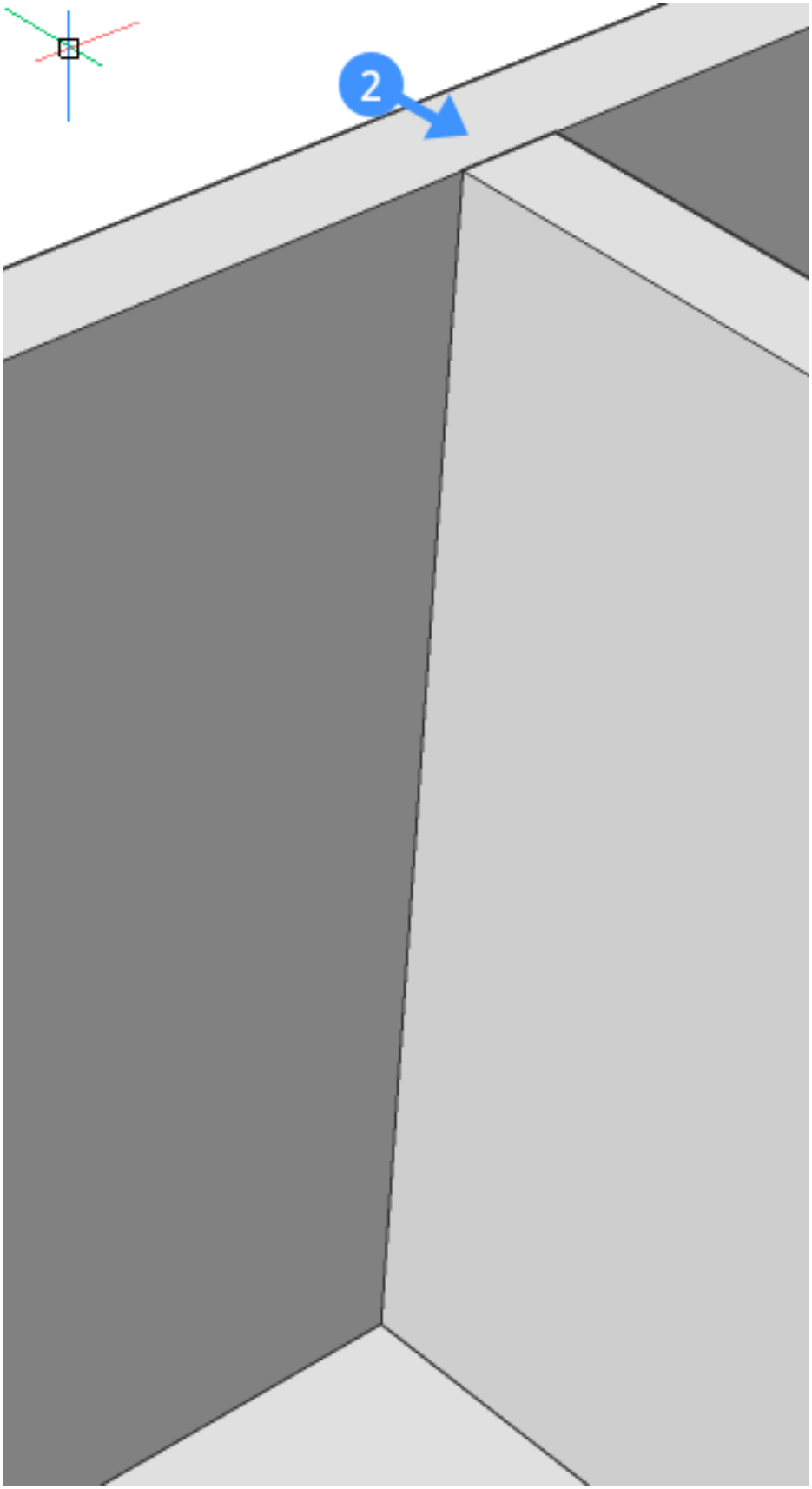


Figure 2.190



Access: Select Aligned Faces Tool

 Quad: **Select tab > Select aligned faces** Keyboard: **SELECTALIGNEDFACES**

Procedure: Connecting multiple faces of the wall to the roof slab

The following steps show how to connect multiple faces of the wall to the roof slab.

1. In the drawing area, highlight the top face of one of the walls (16) and then click the **Select aligned faces** tool from the quad. All aligned faces to this highlighted face will be selected (17).

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360021163953/mceclip8.png}) (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360021164013/mceclip9.png})

2. Launch the connect with nearest tool. The top faces of the wall are attached to the roof (18).

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360020252414/mceclip10.png})

Figure 2.191

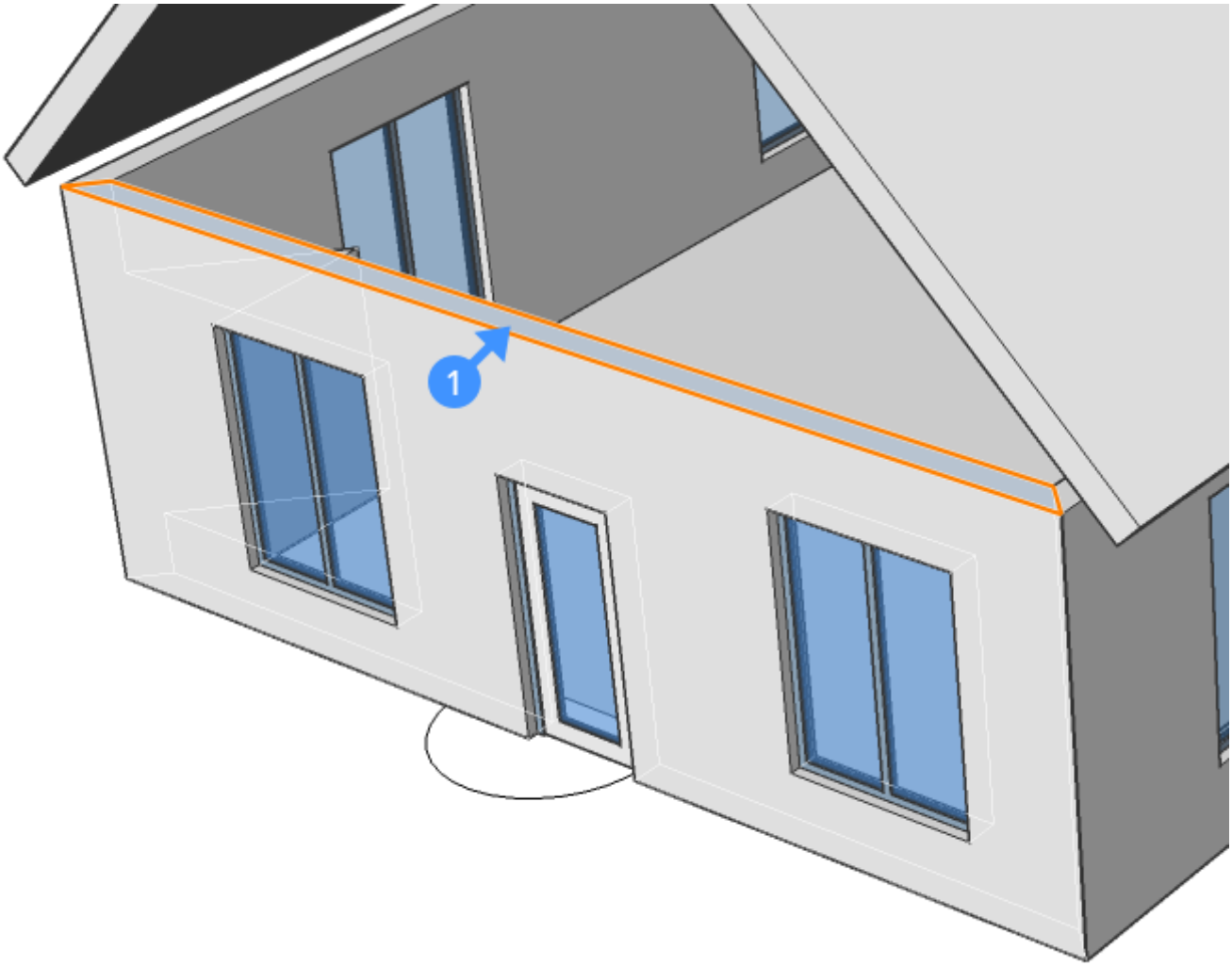


Figure 2.192

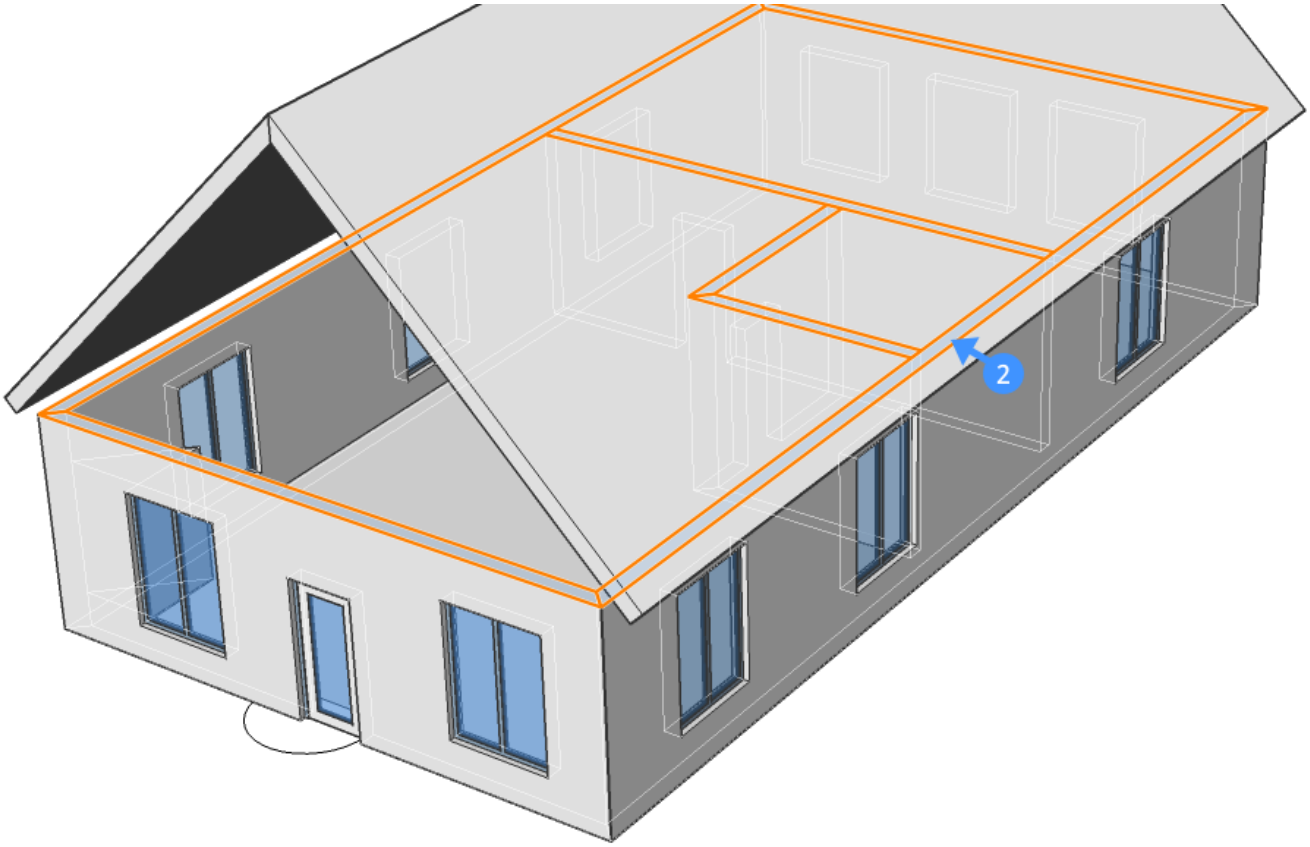


Figure 2.193

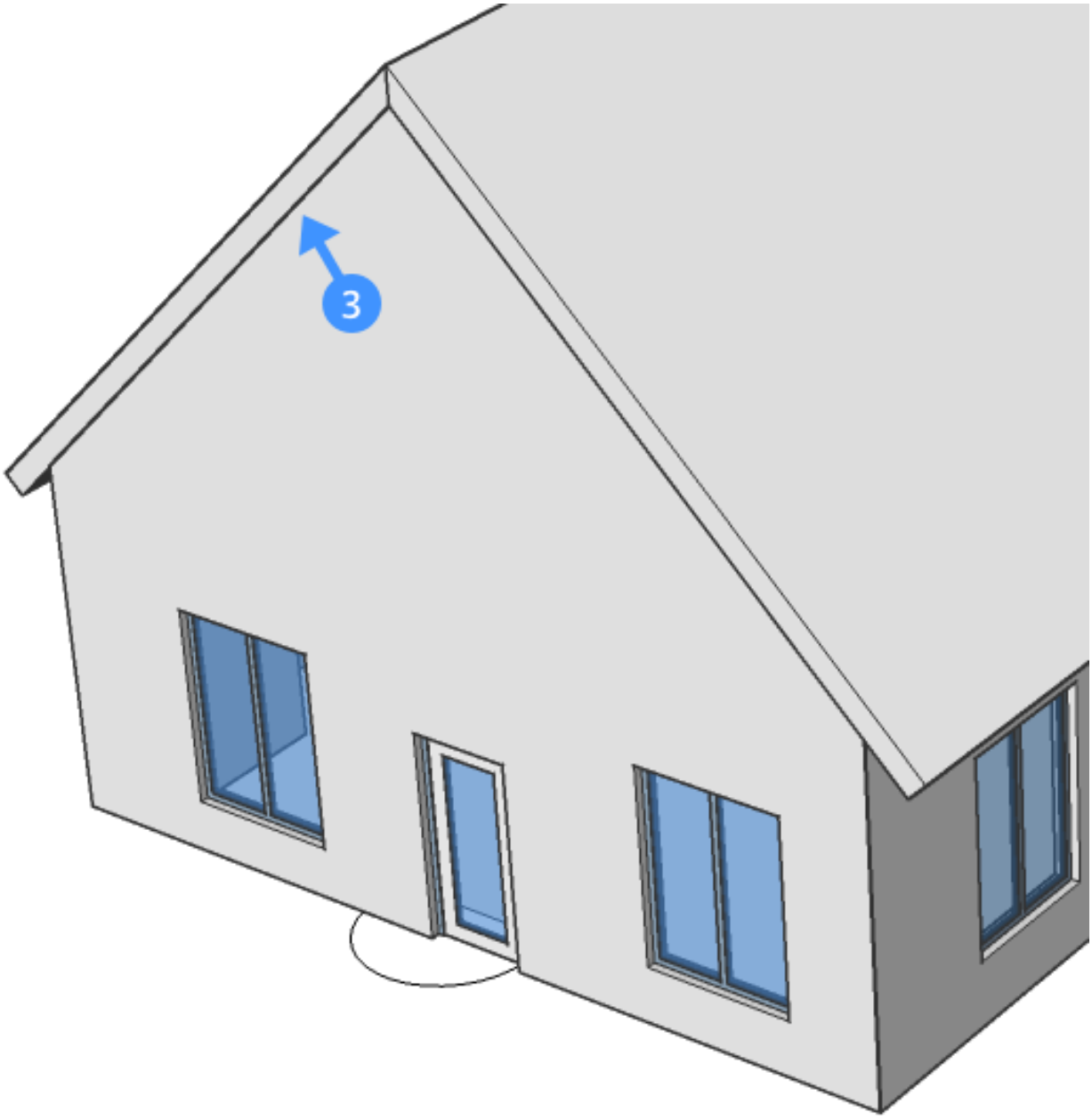
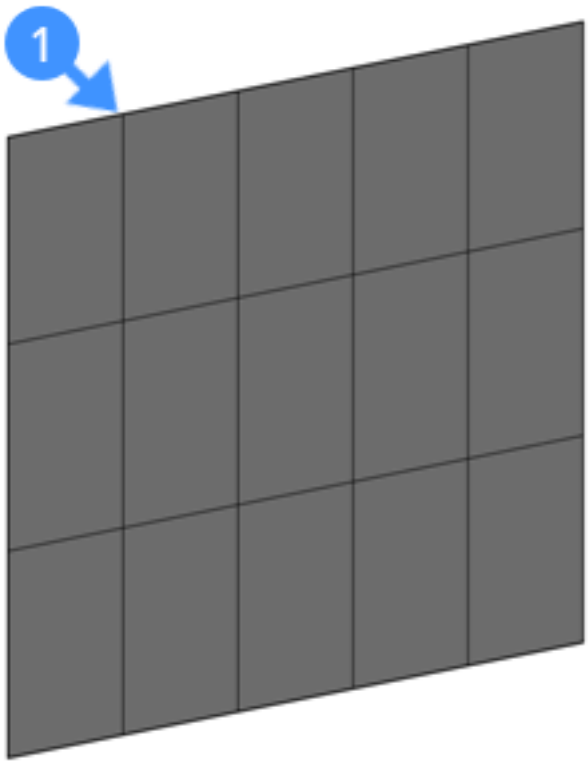


Figure 2.194



2.4.10 Curtain Wall

Command: BimCurtainWall (6.7.7) **Overview:** In BricsCAD, curtain walls are created using the **Curtain Wall** tool.

- Curtain walls can be created from any planar and curved surface.
- Window Frames are placed on the BIM Grid when creating the Curtain Wall.
- Changing the vertical (U line) and horizontal (V line) panels allow you to acquire the desired number of panels on the planar or curved surface.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017281654/GRID-arrow.png}) (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360018003533/glazing.png}) **Figure 1, the curtain wall grid (1) and glazing (2).** The curtain wall system consists of panels of glass. This is a panel infill for a curtain wall. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017281834/CURTAINWALL2.p **Figure 3, the curtain wall frames that coincide with the grid vertical (U) panels (1), the frames that coincide with the grid horizontal (V) panels (2), and in-fills of glass (3) are shown. By default, the straight connection (4) is made between the beams.**

Tutorial: How to create Curtain Walls Click here (2.3.13) to watch

Figure 2.195

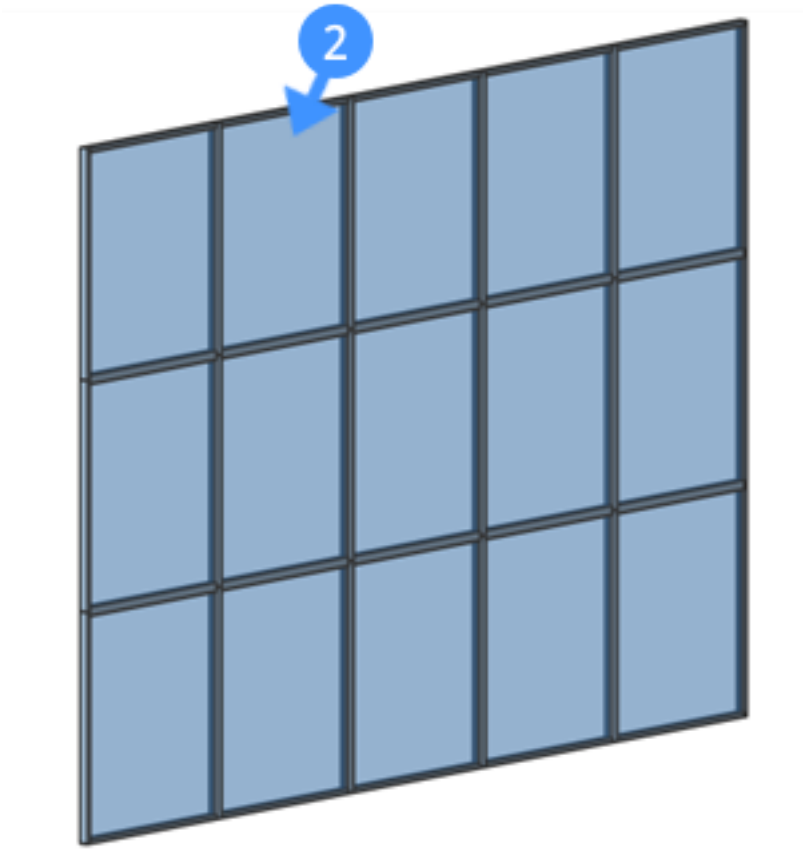
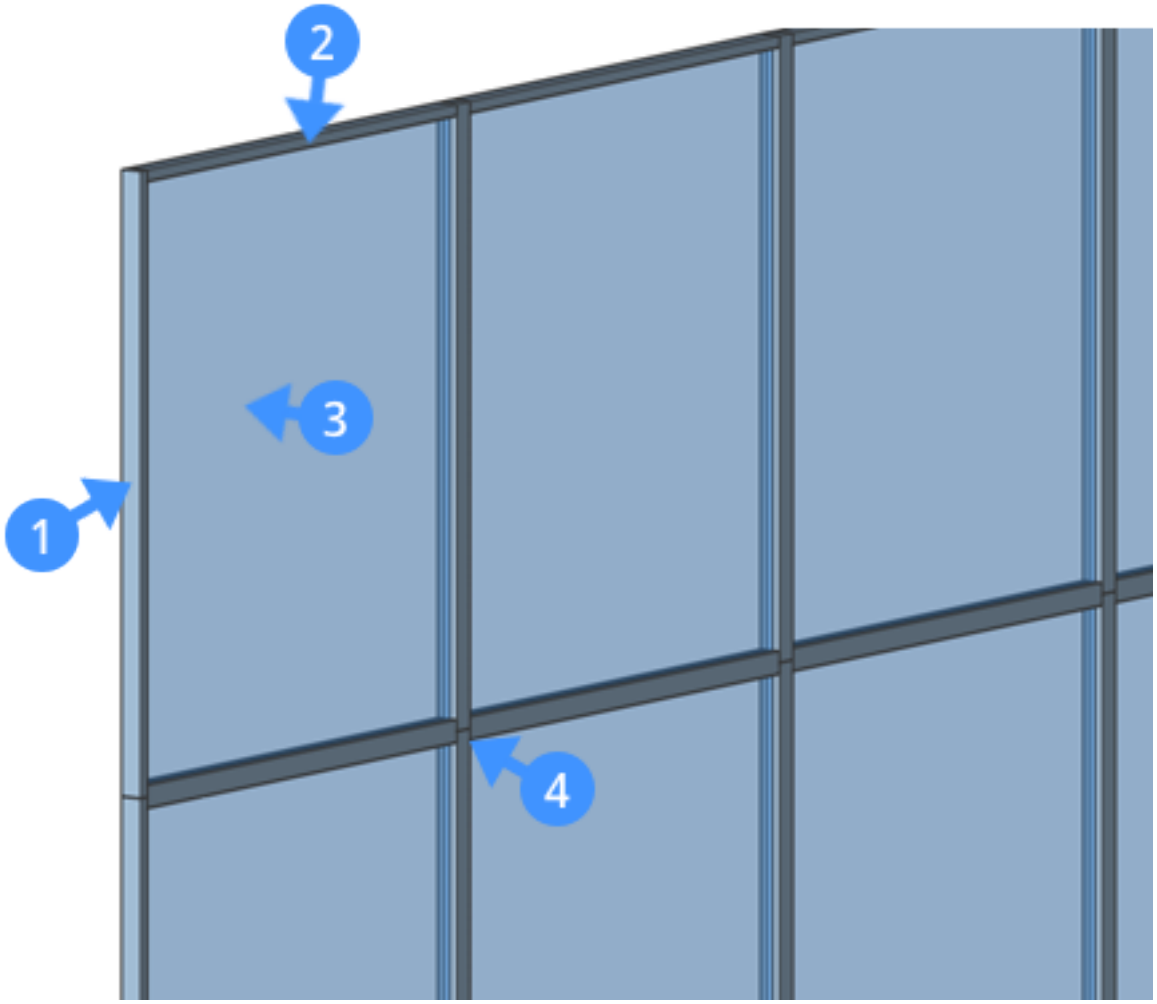


Figure 2.196



Creating a curtain wall

1. Launch the BIMCURTAINWALL command in the command line.

You are prompted:Select a face:

2. Select a face in the drawing area to create a curtain wall system.

The curtain wall grid is automatically generated on this face. **You are prompted:Create grid or [Lenght U panels//Length V panels/Number U panels/Number V panels]:**

3. Change the length of the U or V panels and number of U or V panels using the prompt menu and command line.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017282214/promptmenu.png}) **Or** if **Dynamic dimension (DYN)** is on, the length of the panels can be changed using the DYN fields (1). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360018004493/DYNCurtainwall.png})

4. Hit the **TAB** key to jump to the next dynamic dimension of the grid.

Or Press **Enter**to accept the current value. **Note:** The maximum deviation from planarity is displayed in the command line. **Optional** If the value of the max deviation from planarity is different to ”0”, the prompt menu (1) will appear and command line prompts a message (2). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/
(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017286214/planarizecommandline.png})
The prompt menu and command line allows you to planarize the grid cells. Click ”Yes” to planarize the grid cells. By default, the target deviation from planarity is set to ”0.00001”.

5. Press **Enter** to accept the default value.

You are prompted: Change parameters or accept [Accept/Width/Depth/Glass thickness/Smooth connections] <Accept>: The following image shows the prompt menu: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachm
Optional To change the frame width, select the Width option in the prompt menu or type ”W” in the command line. The default value of the frame width = 60.000000 **Optional** To change the frame depth, select the Depth option in the prompt menu or type ”D” in the command line. The default value of the frame depth = 60.000000 **Optional** To change the glass thickness, select the Glass thickness option in the prompt menu or type ”G” in the command line. The default value of the glass thickness = 20.000000 **Optional** To apply the smooth connection between the beams, select the Smooth connections option in the prompt menu or type ”S” in the command line.

6. The blue lines (1) on the face appear to show the side of the curtain wall to be created.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017291734/CURTAIN-WALLbluelines.png})

7. Use the widget (2) to flip the side of the curtain wall from outside to inside or vice-versa.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360018014113/flipfaceicon.png})

Figure 2.197

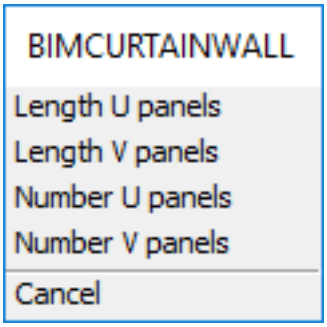


Figure 2.198

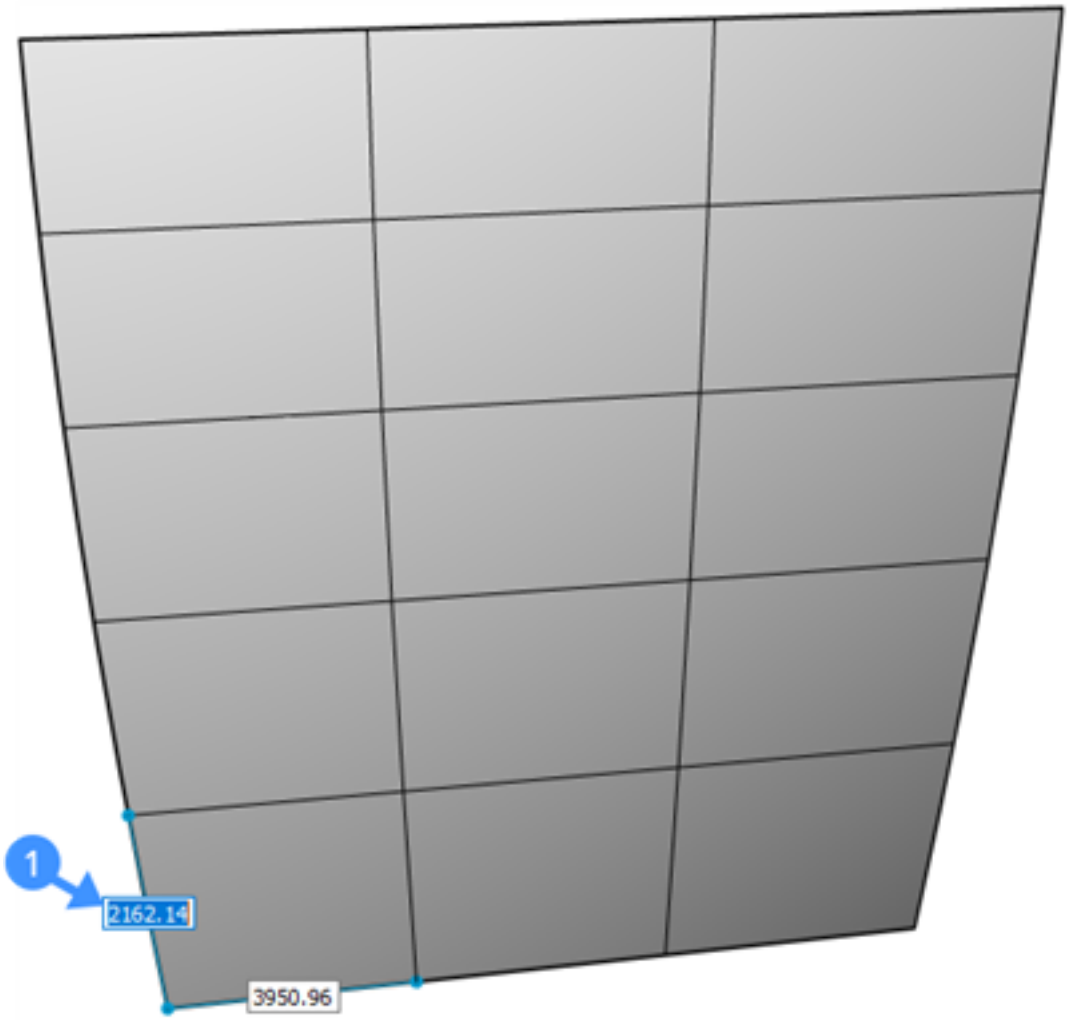


Figure 2.199

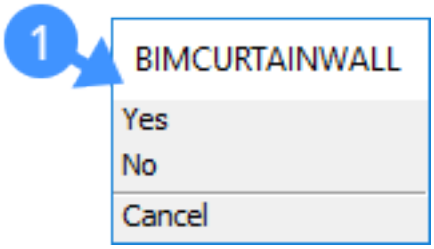


Figure 2.200

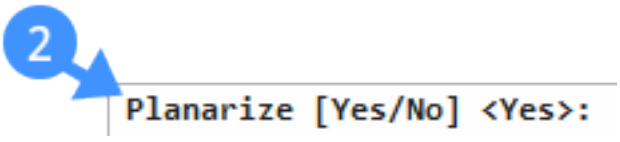


Figure 2.201

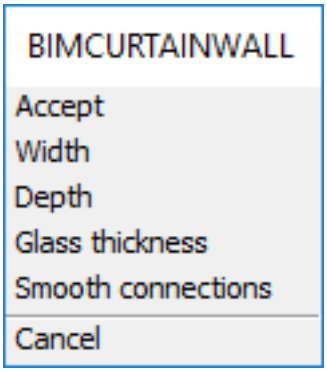
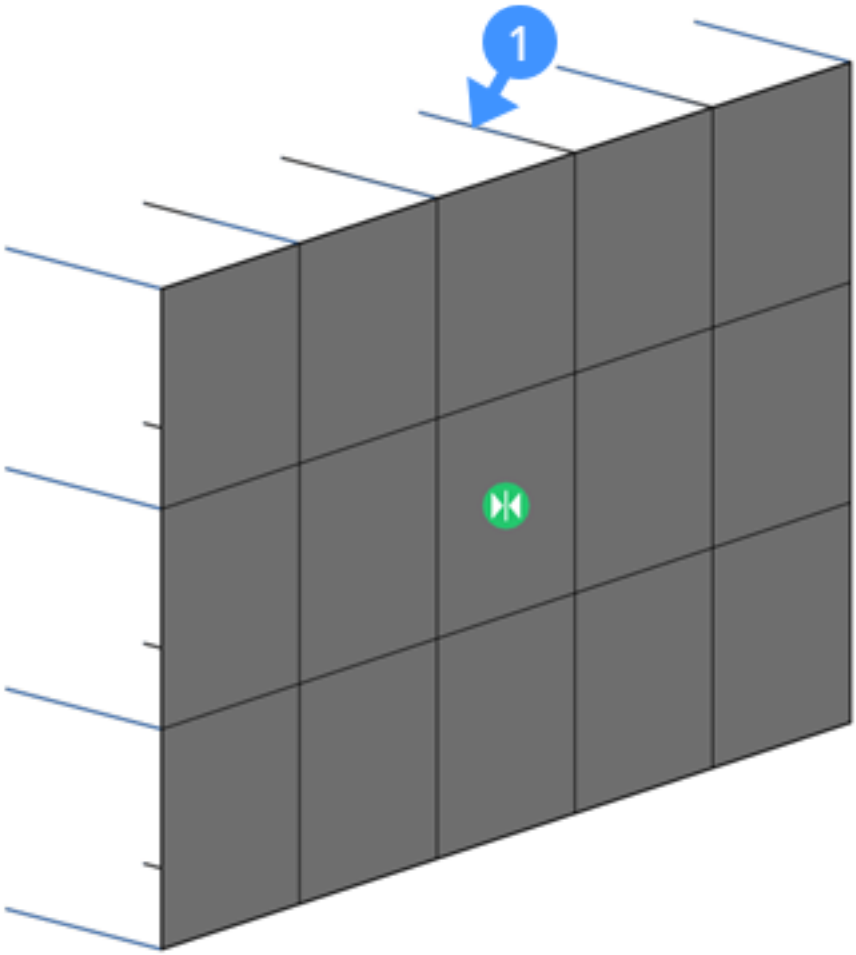
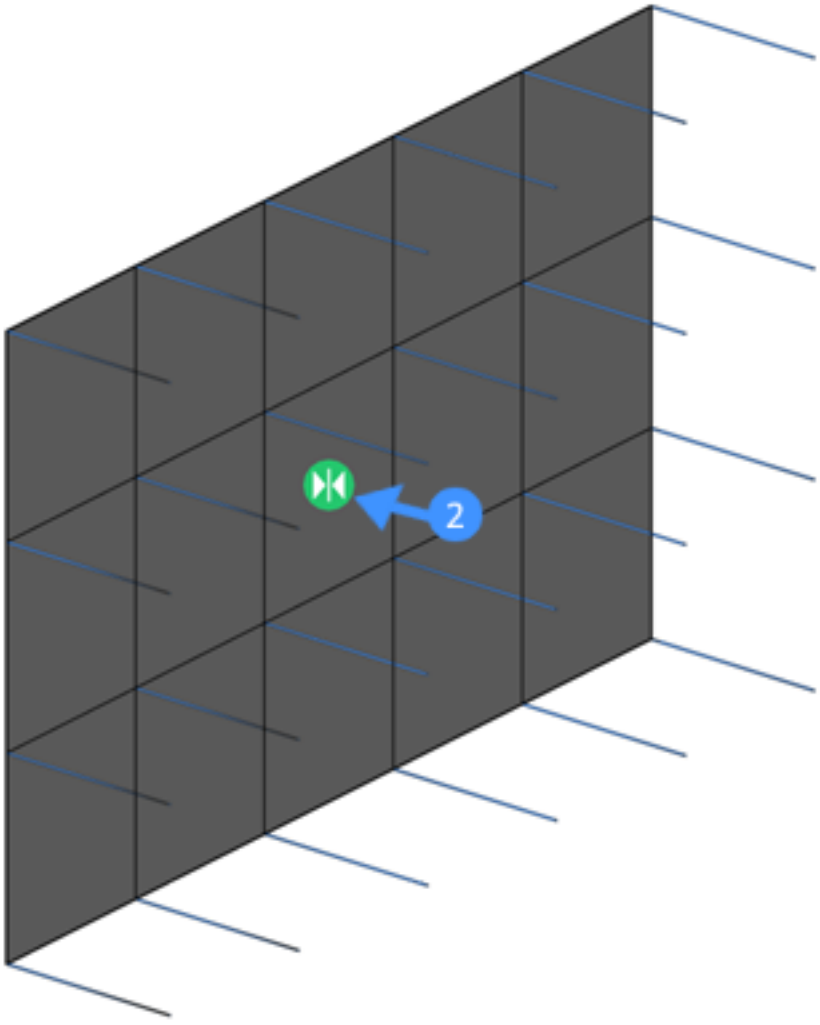


Figure 2.202



8. Press Enter to accept.

Figure 2.203



2.4.11 DmExtrude

Command: DMExtrude (6.13.21) **Overview:** In this lesson, you learn how to use the Extrude tool to create Solids and Surfaces. **Lesson Objectives** After completing this lesson, you will be able to:

- Create solids and surfaces from existing 2D entities or boundary.
- Understand the Extrude tool options: Auto, Create, Subtract, Unite, Both sides.

Tutorial: How to extrude Click here (2.3.7) to watch

About

The Extrude tool allows you to create 3D solid(s) by extruding closed 2D entities, faces of 3D solid(s), regions or closed boundaries. The Extrude tool enables you to cut the solid model partially by removing the volume from the main solid and it can also create new solid(s) by adding volume to the main solid model. Menubar: **Model tab> 3D Solids > Extrude** Toolbar: **Modify> Extrude** Quad: **Model tab > Extrude** Keyboard: **DMExtrude** Ribbon: **Model tab > Modify 3D> Extrude** The following illustration shows the selected entity, entities or boundary to be extruded. Closed entities (1) (2) are shown. A region object (3) is shown in the figure. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ClosedEntities2D.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ClosedEntities3D.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/RegionObject.png}) The following illustration shows a closed boundary in the XY-plane of the current coordinate system (4) and the closed boundary on the face of the solid (5) is highlighted while hovering over it. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ClosedBoundary3D.png}) The closed boundaries are recognized automatically in BricsCAD, unless the **Select detected boundaries**option on the **Selection modes** toolbar is inactive. **Note:** As a closed entity only closed polylines, circles, and ellipses are accepted.

Figure 2.204



Figure 2.205

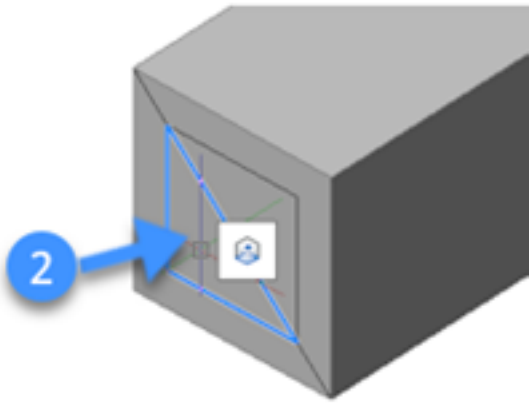


Figure 2.206

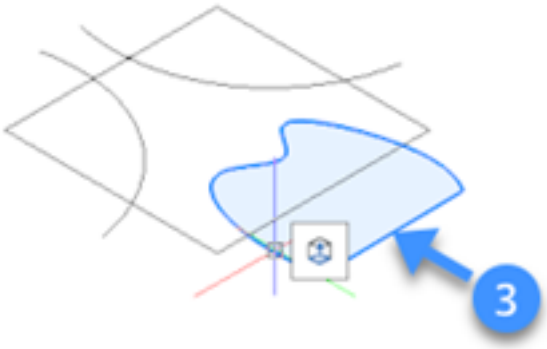


Figure 2.207

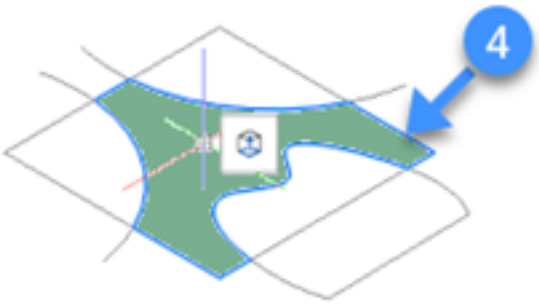
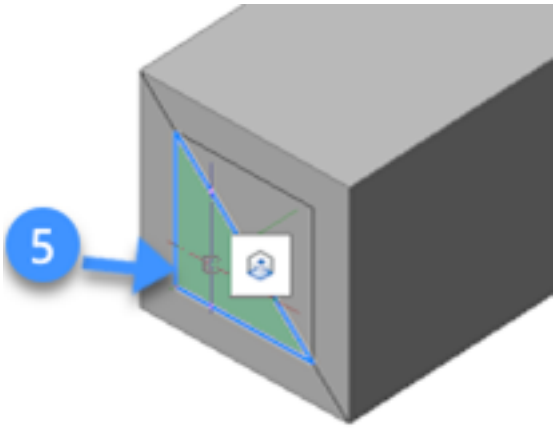


Figure 2.208



Procedure: Using DmExtrude in Solid Mode

The following are general steps to add and remove volume from the solid(s) using the Extrude tool.

1. Hover the cursor over an entity or sub-entity in the drawing area.
2. Select the Extrude tool in the Model tab from the Quad. **You are prompted:** Specify height of extrusion or set [Auto(subtract or create)/Create/Subtract/Unite/Both sides/Taper angle/Direction/Limit] <Auto>
3. The “DmExtrude Hotkey Assistant” appears at the bottom of the screen with a set of extrusion options. **Note:**The default mode is indicated by a blue frame. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V

- **Auto (6):** The result of the auto option depends on the extrusion direction of the 3D solid and the value of the DMEXTRUDEMODE setting value. **Note:** By default, the value of the DMExtrude mode is set to “8”. The following illustration shows the extruded entity from the main solid while the default value is set to “3”. When you highlight an entity (10) and extrude it to the **outward** (11), a new volume is added (12) to the main solid model. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ExtrudeOut1.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ExtrudeOut2.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ExtrudeOut3.png}) The following illustration shows the subtracted entity from the main solid. When you highlight an entity (13) and extrude it to the **inward** (14), a volume is subtracted (15) from the main solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ExtrudeIn1.png})

Figure 2.209

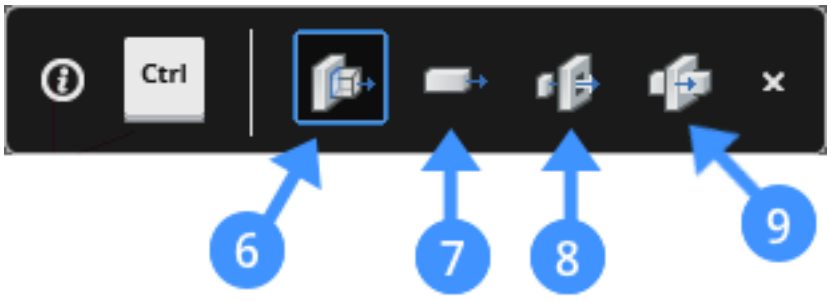


Figure 2.210

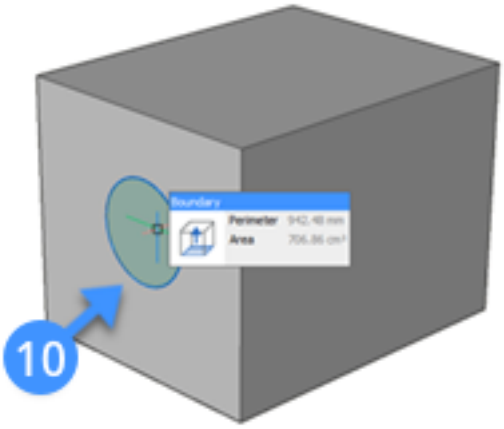


Figure 2.211

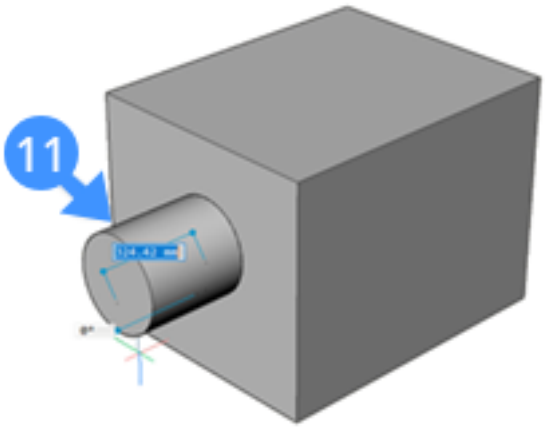
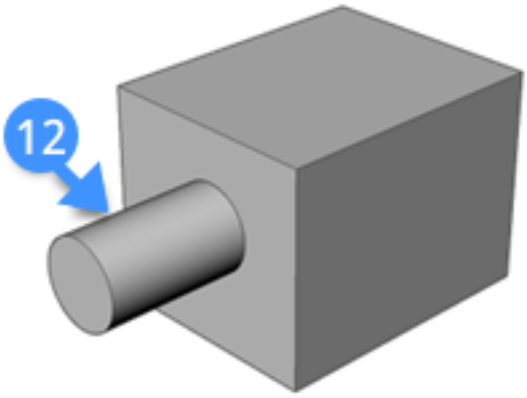


Figure 2.212



(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ExtrudeIn2.png})
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ExtrudeIn3.png})

- **Create (7):** Regardless of the extrusion direction, a new volume is created (16) from the existing solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ExtrudeCreate.png})

- **Subtract (8):** The 3D solid is subtracted from each interfering existing solid. The following illustration shows the subtracted solids (18,19) using the highlighted boundary (17). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ExtrudeSubtract2.pn

- **Unite (9):** The new volume is unified with each interfering existing solid. The following illustration shows the before and after when using the extrusion tool unite option. The solid surface is highlighted (20) and the distance between the left and right face is shown with the aligned line (21). The new volume is unified (22) with the first solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/Extrud
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ExtrudeUnite2.png})

- **Both sides:** New symmetric volumes are created (23), (24) in both directions from the main solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ExtrudeUnite3.png}) To create symmetric volumes from the existing solid, type “B” on the command line which corresponds to Both sides option of the Extrusion tool.

Select one of the available options to operate the extrusion function on your solid. Next, move the cursor in the desired direction to specify the extrusion direction. The selected entity or sub-entity is extruded dynamically. Define the height of extrusion by specifying a point in the drawing, entering a value in the dynamic dimension field or choose the limit option. Press enter or right-click to accept the extrusion from your existing solid.

Figure 2.213

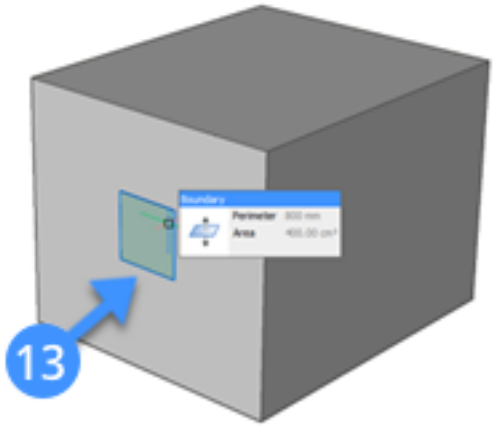


Figure 2.214

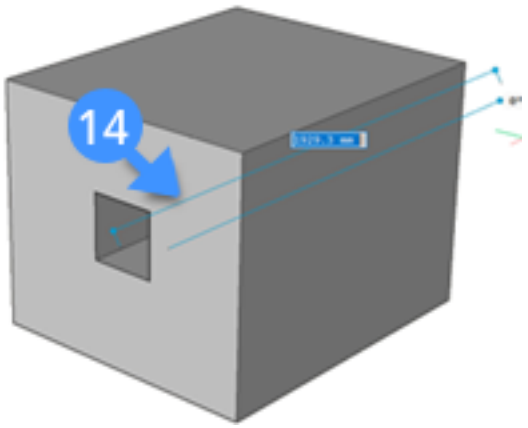


Figure 2.215

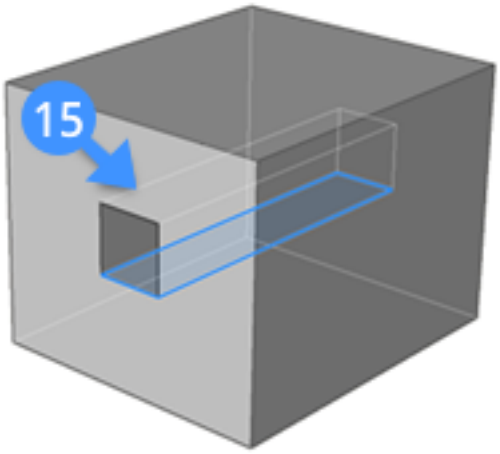


Figure 2.216

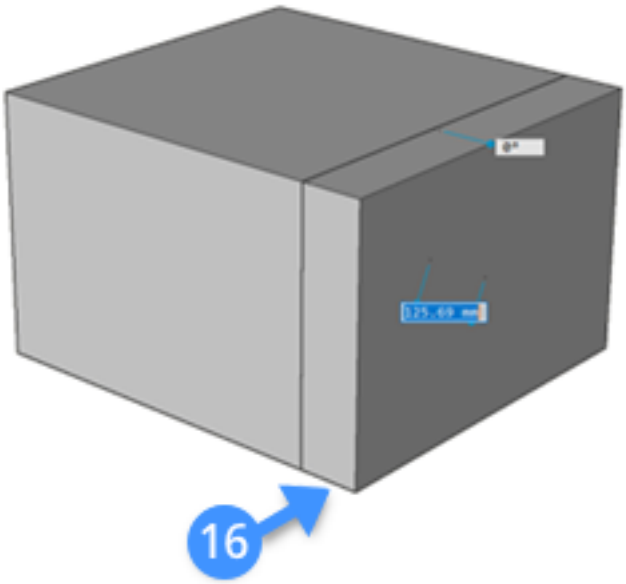


Figure 2.217

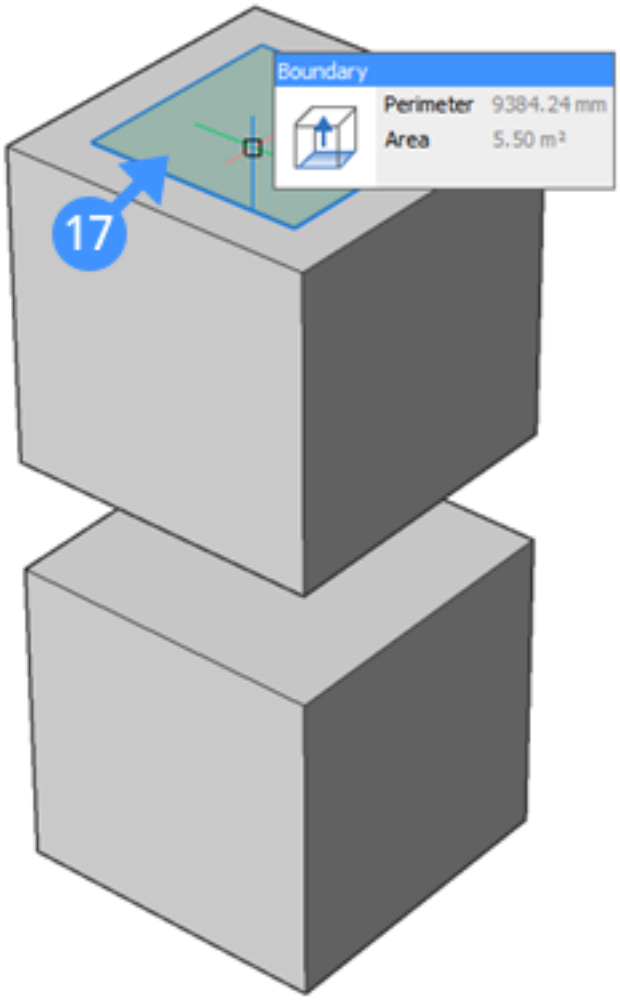


Figure 2.218

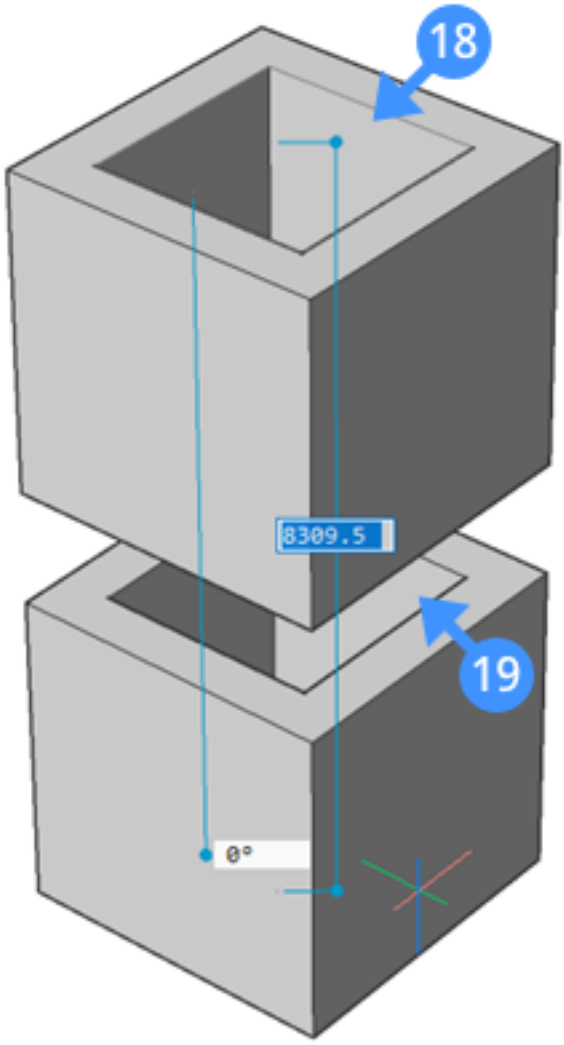


Figure 2.219

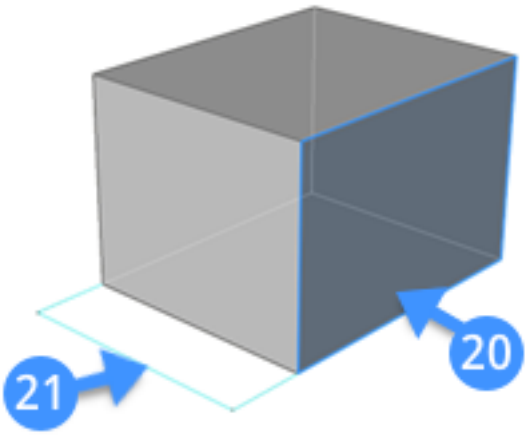


Figure 2.220

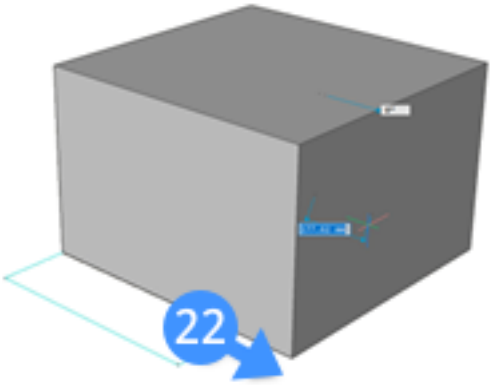
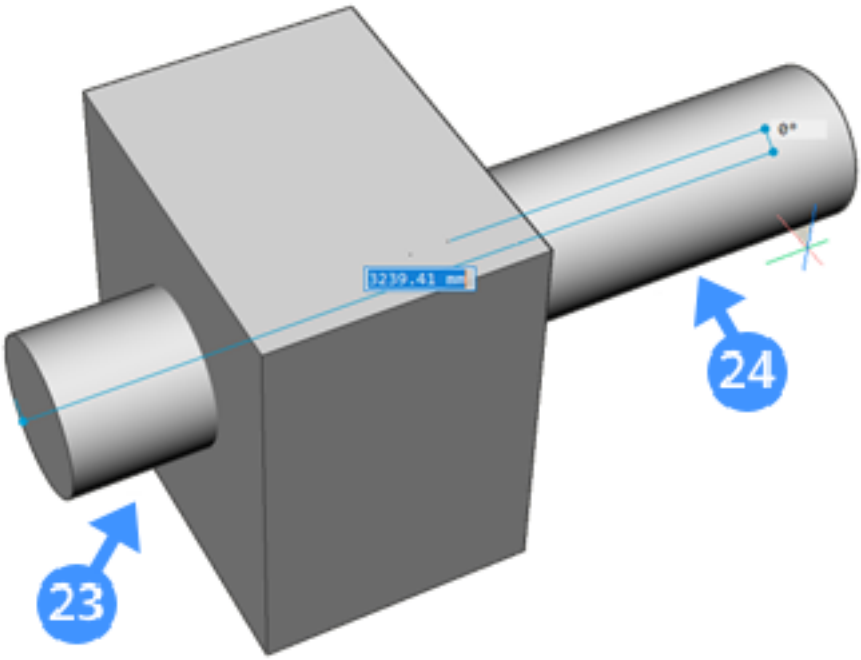


Figure 2.221



DmExtrude in Surface Mode

The extrude tool allows you to create 3D surface(s) by extruding 2D entities such as lines or polylines.  Toolbar: **Direct Modeling> Extrude Surface** Quad: **Model tab > Surface Extrude** Keyboard: **DMExtrude**

Procedure: Using the Extrude tool in Surface Mode

The following are general steps to create 3D surface(s) using the Extrude tool in the Surface mode.

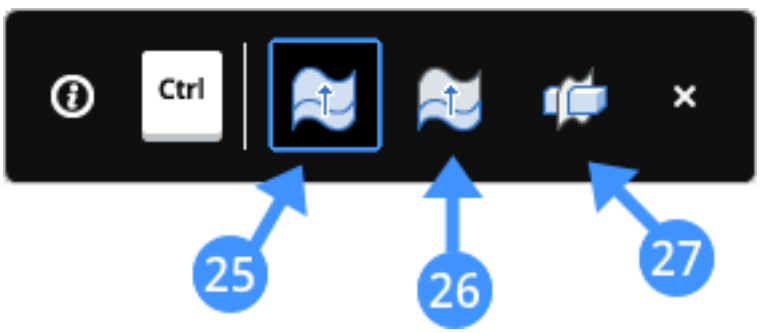
1. Move the cursor over an entity or sub-entity in the drawing area.
2. Select the Surface Extrude tool in the Model tab from the Quad. **You are prompted:** Specify height of extrusion or set [Auto(subtract or create)/Create/Subtract/Unite/Both sides/Taper angle/Direction/Limit] <Auto>
3. The "DM Extrude Surface Hotkey Assistant" appears at the bottom of the screen with a set of surface extrusion options. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/HKASurface
NOTE: The default mode is indicated by a blue frame.

Auto (25)	Select either a surface or the face of a solid element. When the surface's edge is used as a contour for extrusion, the created surface is stitched with**the base**surface. When the surface's face is used for extrusion, the surface results in not stitched withthe base surface.
Create (26)	Create new surfaces
Slice (27)	Cut through solids with the extruded surface. This option is much like the Slice command.

- Select one of the options and move the mouse to give the extrusion direction, entering a value in the dynamic dimension field.
- Press enter or right-click to accept the extrusion for the surface.

The following illustrations show the created surface (29) by extruding the selected 2D entity (28) in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SurfaceExtrude1

Figure 2.222



(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SurfaceExtrude2.png

Figure 2.223

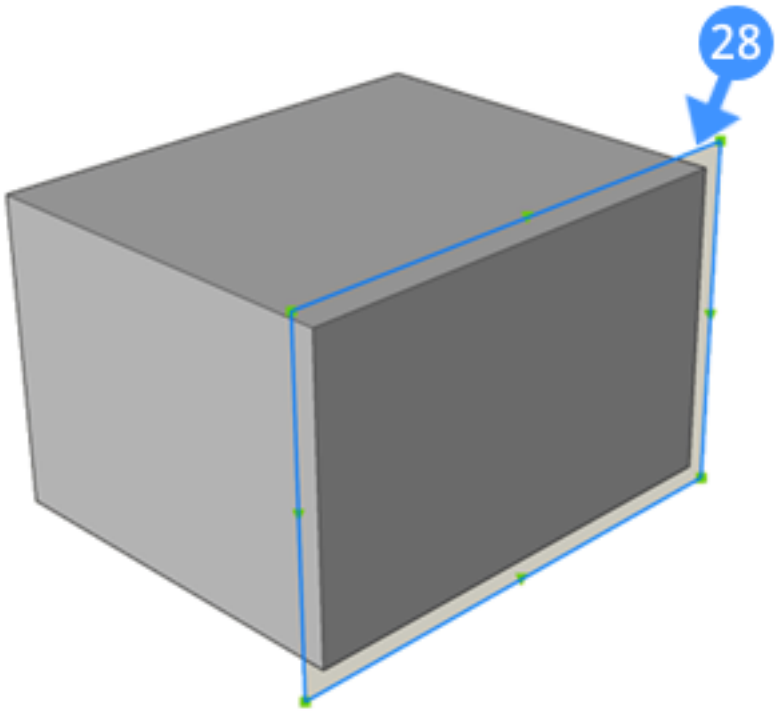
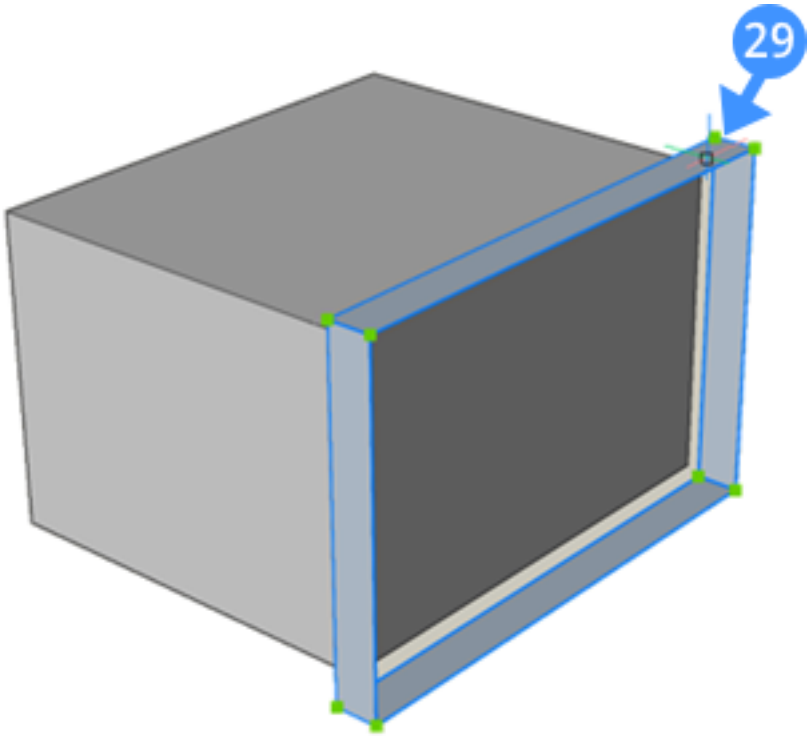


Figure 2.224



2.4.12 DmPushPull

Command: DmPushPull (6.12.20) **Overview:** In this lesson, you learn how to make use of the Push/pull tool in your drawings. **Lesson Objectives** After completing this lesson, you will be able to:

- Add a volume to a solid.
- Remove a volume from a solid.

Tutorial: How to push or pull solids Click here (2.3.10) to watch

About

The Push/pull tool allows you to add and remove volume from solids by moving highlighted faces. **Note:** Push/pull tool applies to Planar, cylindrical, spherical, conical and toroidal faces of a solid.  Menubar:**Model tab >Direct Modeling > Push/Pull** Toolbar: **Modify > Push/Pull** Quad: **Model tab > Push/Pull** Keyboard:**dmpushpull** Ribbon: **Model tab > Modify 3D > Push/Pull**

Procedure:Using Push/Pull tool to add volume to the solid

The following are general steps to add volume from the solid (s) using the Push/Pull tool.

1. Highlight any face(s) of the solid(s) in the drawing.
2. Select the Push/pull tool () in the Model tab from the Quad.
3. Move the cursor forward to pull the highlighted face. The following illustration shows the highlighted face behavior before and after using the Push/pull tool. On the left, the face moves dynamically (1) and on the right, extra volume is added (2) to the main solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/PushPullAddVolume
4. Define the height of pulling by specifying a point in the drawing or entering a value in the dynamic dimension field.
5. Press enter or right-click to accept changes.

Figure 2.225

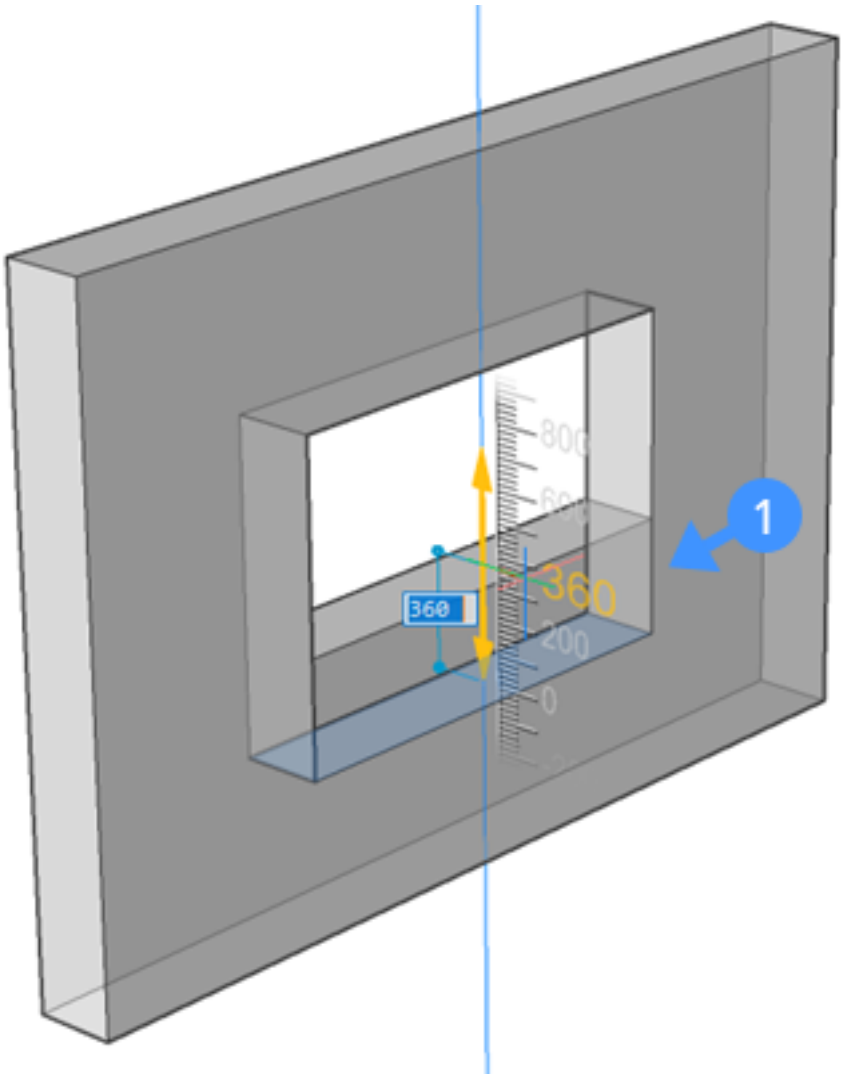
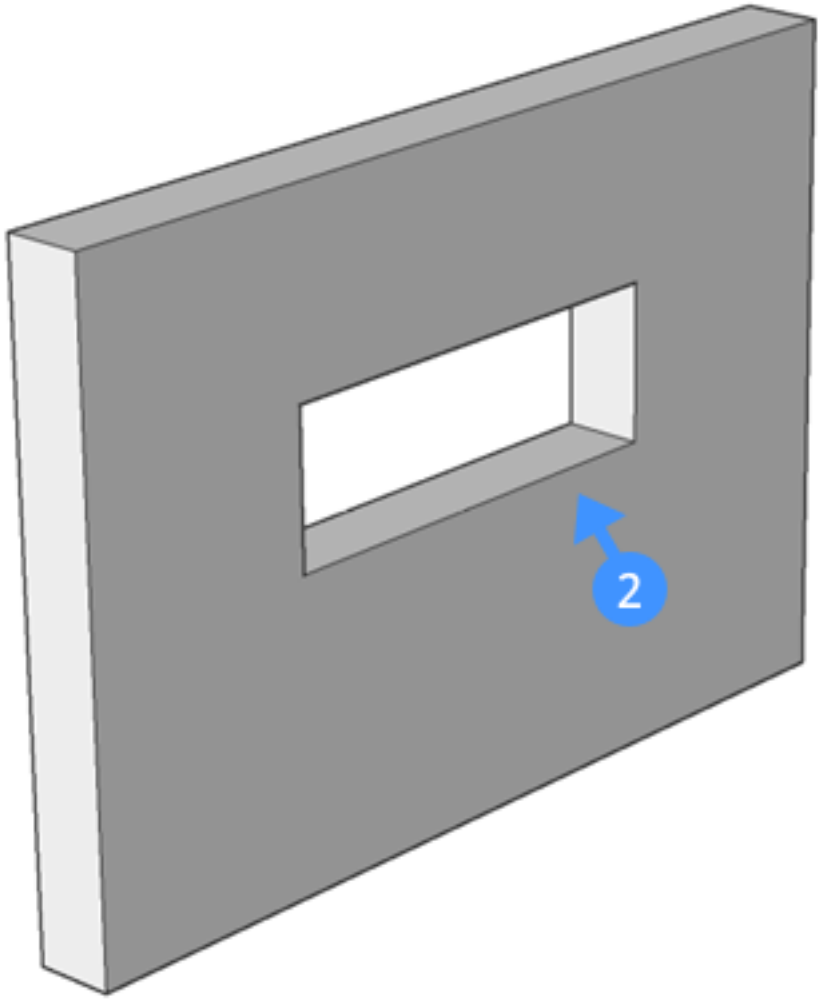


Figure 2.226



Procedure:Using Push/Pull tool to remove volume from the solid

The following are general steps to remove volume from the solid (s) using the Push/pull tool.

1. Highlight any face(s) that you want to move.
2. Select the Push/pull toolin the Model tab from the Quad.
3. Move the cursor backward to push the highlighted face. The following illustration shows the highlighted face behavior before and after using the Push/pull tool. On the left, the face moves dynamically (3) while pushing the face down and on the right, the solid is removed (4) from the main solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/PushPullRemoveVolume.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/PushPullRemoveVolume2.
4. Define the height of pushing by specifying a point in the drawing or entering a value in the dynamic dimension field.
5. Press enter or right-click to accept changes.

Figure 2.227

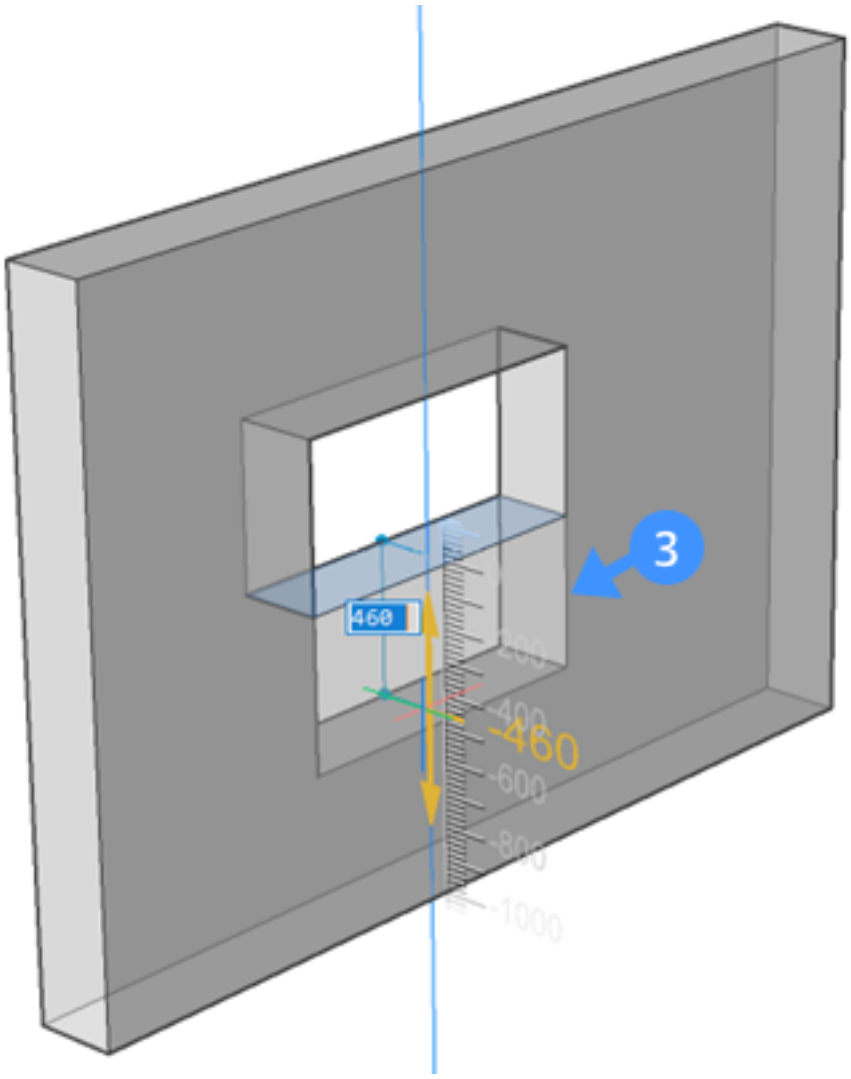
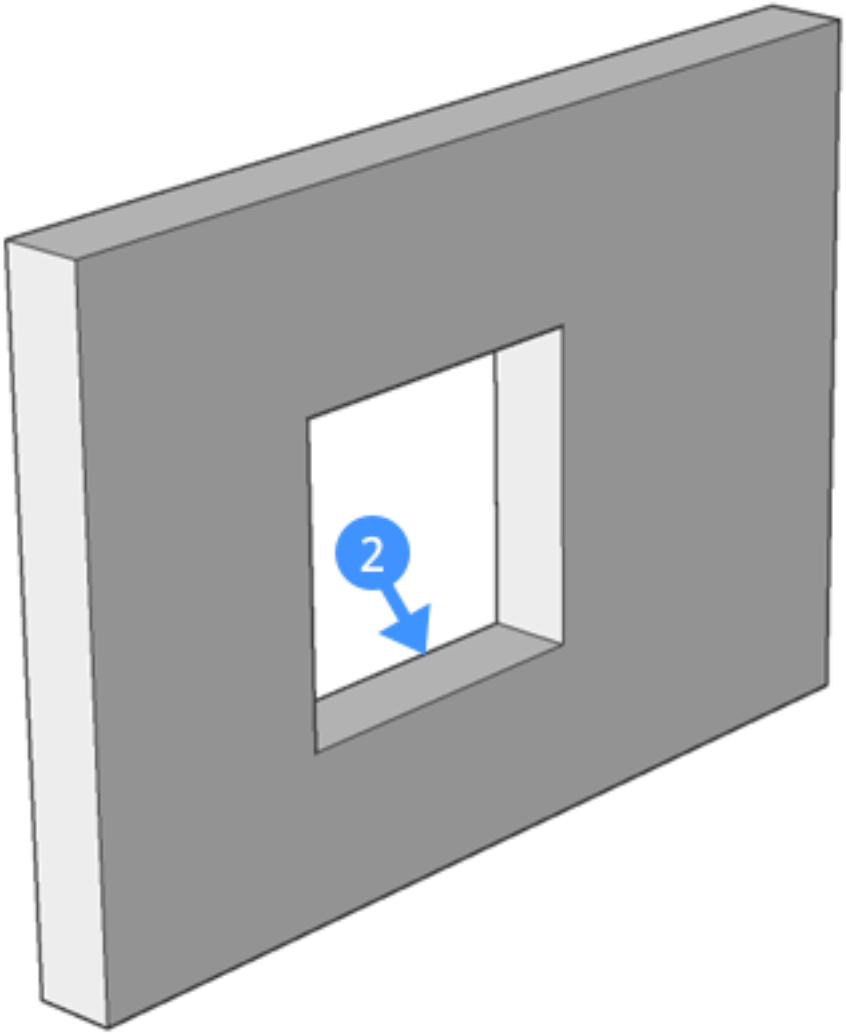


Figure 2.228



2.4.13 Editing Grids

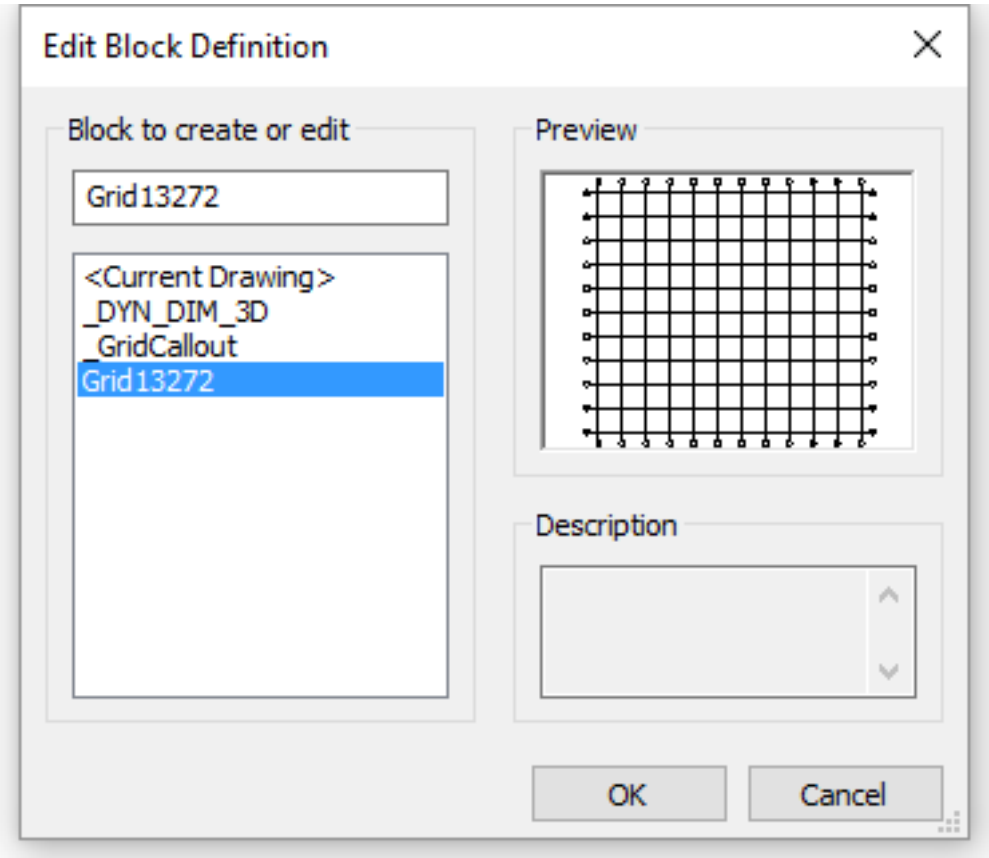
Command: BimGrid (6.7.12), BEdit (6.6.7), Number (6.22.9)

Editing a rectangular grid

The BEdit (6.6.7) (Block Edit) command allows to edit a rectangular grid. Do the following:

1. Double click the grid. The **Edit Block Definition** dialog displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h
2. Select the grid, then double click or click the OK button. A Block Edit session is started.
3. Edit the grid axes. You can move, rotate, copy, stretch, grip-edit or delete grid axes.
4. Renumber grid axes if needed.
5. Close the Edit Block session: Type **bclose** at the command prompt, then choose the **Save** option. or click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

Figure 2.229



Renumbering the grid labels

Renumbering grid labels is needed:

- If you want a different numbering style.
- When grid axes are added (copied) or deleted.

Do the following:

1. Double click the grid to start a Block Edit session (see step 1 and 2 in the procedure above).
2. Select all vertical axes.
3. Start the Number (6.22.9) command and choose the appropriate number style. Make sure to overwrite the existing numbers.
4. Repeat the previous steps for the horizontal axes.
5. Close the Edit Block session: Type **bclose** at the command prompt, then choose the **Save** option. or click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

2.4.14 MEP Modeling

Commands: BimApplyProfile (6.7.2), BimLinearSolid (6.7.15), BimFlowConnect (6.7.11), BimDrag (6.7.8) **Overview:** In this lesson you learn how to create HVAC and piping segments, and how to manipulate these segments using different tools. **Lesson objectives** After completing this lesson, you will be able to:

- Insert profiles into your BIM model.
- Apply a profile to an existing line or solid in your model.
- Connect different flow segments to each other.
- Alter the positioning of segments without losing their connection.

Tutorial: How to use MEP modeling tools Click here (2.3.28) to watch

About BIM Profiles panel

The BIM Profiles panel in BricsCAD makes it possible to insert a variety of preset profiles into your model. It covers three main domains: mechanical, engineering and plumbing profiles (also known as MEP). **Note:** If the BIM Profiles panel is not displayed on the right hand side of your screen, right-click on the toolpanel on the right hand side, then check BIM Profiles. You can access the BIM profiles panel by clicking the I-beam on the right hand side of your screen. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingBIM
You can filter based on domain (1) or country standard (2), or you can search using the searchbar (3). The menu (4) in the right corner allows you to access the BIM Profiles dialog box. Clicking the menu icon displays a context menu with a couple of options. Now the BIM Profiles context menu expands. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/
Select **Open profile dialog...** (5)in the menu to open the **Profiles** dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys

6. Filters	Filters the profiles list by: • **Type**: All, HVAC, Piping or Structural Steel • **Standard**: Localized standards • **Shape**: Profile shape (L, U, circular, ...) • **Name**: Select a character string. Only profiles of which the name contains the selected string are listed.
7. Tools	: Save the currently edited profile : Crate a new profile : Delete the selected profile : Opens the Project Info (6.7.19) dialog box.
8. Custom fields	Set the type, standard, name, and size of a profile
9. Preselect shape	Select a shape from the drop-down list
10. Profile properties	Shows the properties of the selected profile

Figure 2.230

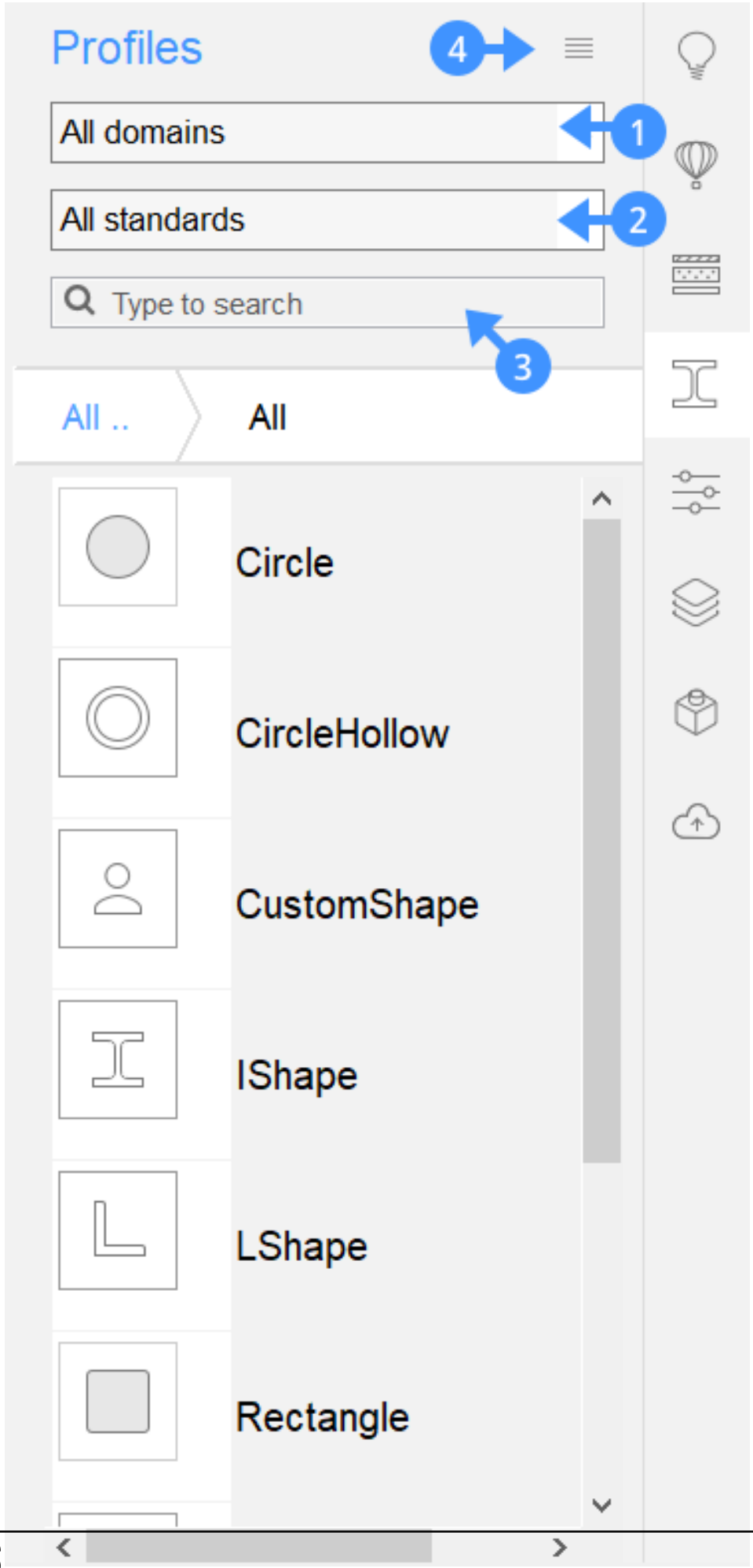

Profiles
4
All domains
1
All standards
2
Type to search
3
All ..
All
Circle
CircleHollow
CustomShape
IShape
LShape
Rectangle

Figure 2.231

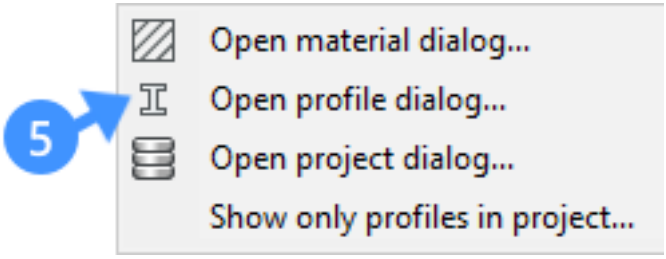
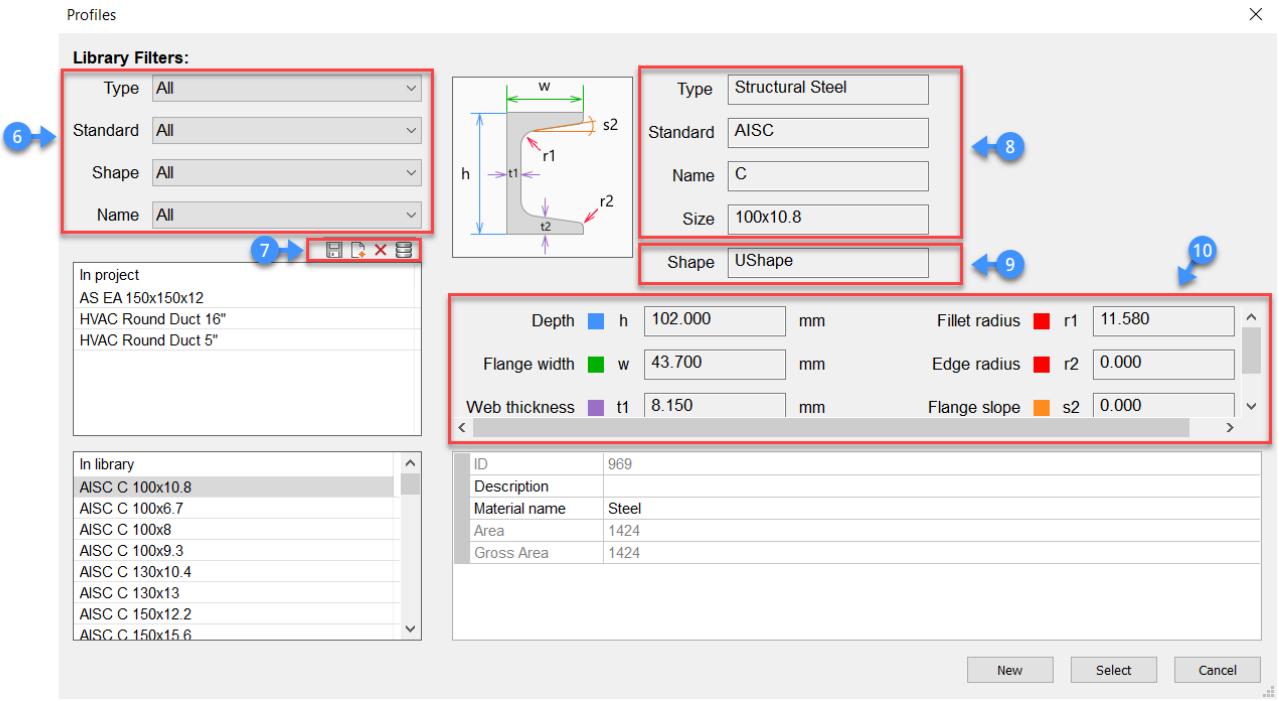
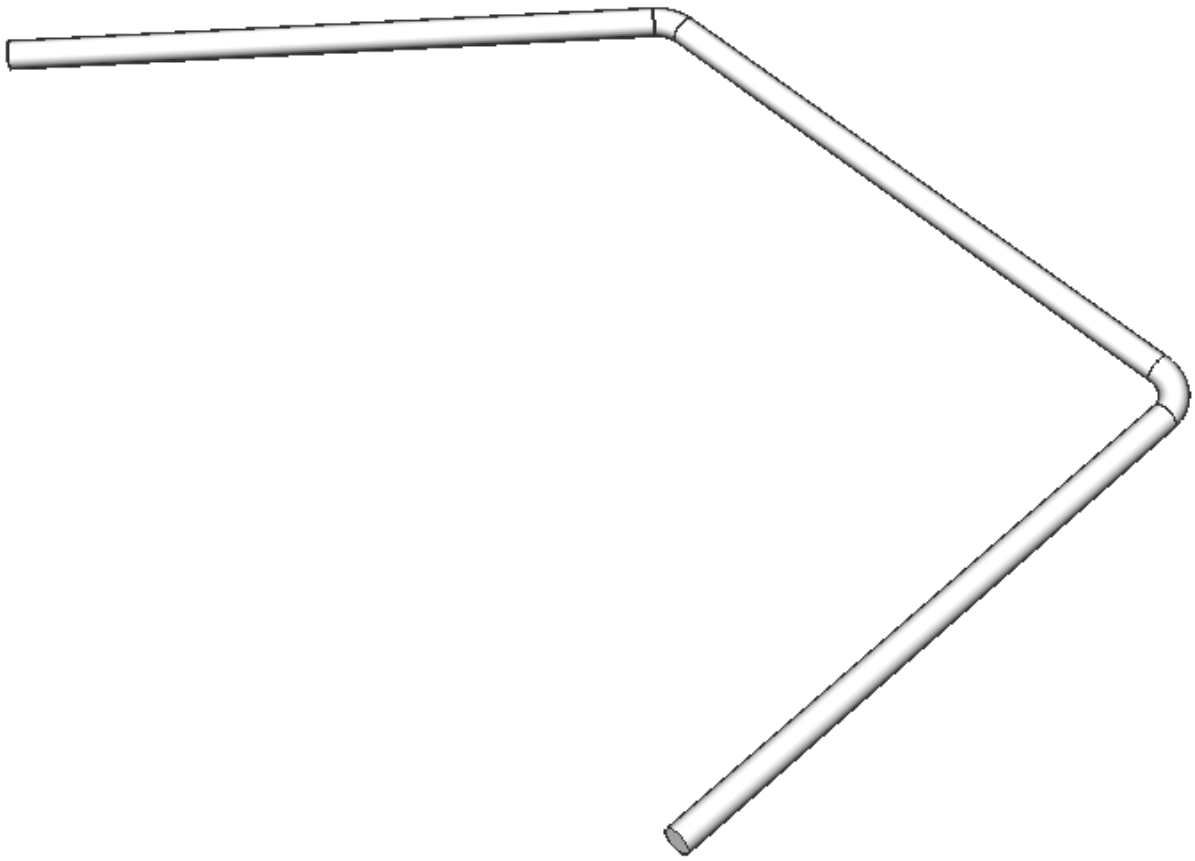


Figure 2.232



Using the BIM Profiles panel Drag and drop the desired profile from the BIM Profiles Panel to the model space. This command works the same as if you were drawing lines. You can define the start point and the next point(s) by clicking in the model space or using the dynamic dimension field. After reaching your final point, press Enter to end the command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModeli
Note: The software automatically generates the connections between the different segments, except if you chose a Structural Steel profile.

Figure 2.233

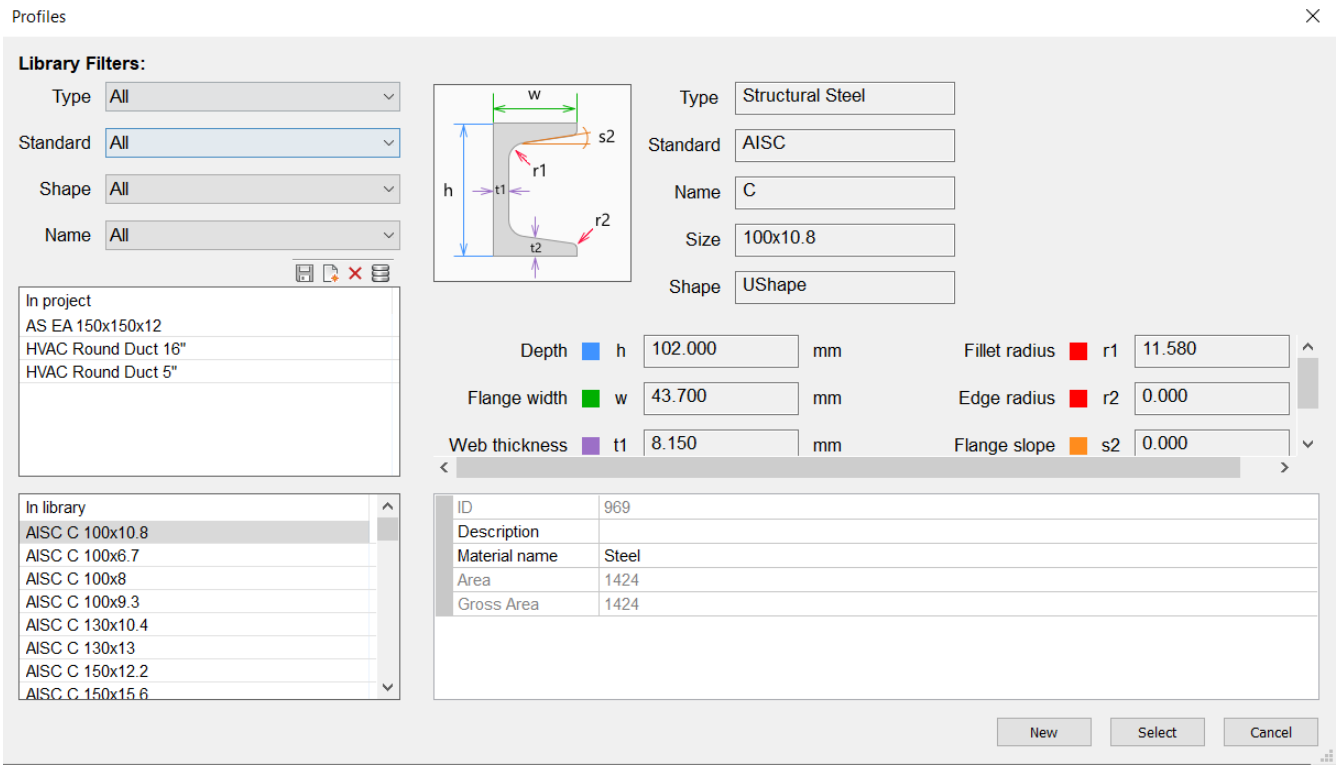


About BimApplyProfile

BricsCAD BIM makes it possible to apply a profile to an already existing line in your model by using the **BimApplyProfile** command.  Menubar: **BIM > Apply profile** Toolbar: **BIM > Attach profile** Quad: **Model tab > Apply profile** Keyboard: **BIMAPPLYPROFILE**

Applying profiles to an already existing line or solid Select the line or solid you want to apply the profile to and launch the **BimApplyProfile** command. After pressing Enter, the same DialogBox as explained above appears. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingBIMApplyPro Choose the desired profile and press the **Select**button in the bottom right corner. Pressing Enter ends the command; the profile is now applied to the line or solid in your model.

Figure 2.234



About BimLinearSolid

BimLinearSolid lets you draw default or chosen profiles like you would draw lines.  Menubar: **BIM > Linear Solid** Toolbar: **Structure/HVAC > Linear Solid** Quad: **Model tab > Linear Solid** Keyboard: **BIMLINEARSOLID**

Drawing with profiles

- Launch the **BimLinearSolid** command.
- You can use default **Rectangular** or **Circular** profiles or select one in the model or in the **Profiles** dialog box.
- After choosing your profile, you can define the start point and the next point(s) by clicking in the model space or using the dynamic dimension field.
- After reaching your final point, press Enter to end the command.

Note: The software automatically generated the connections between the different segments, except if you chose a Structural Steel profile.

About BimFlowConnect

It's also possible to connect different segments using the **BimFlowConnect** command.  Menubar: **BIM > BimConnect > FlowConnect** Toolbar: **Modify > Connect > FlowConnect** Quad: **Model tab > FlowConnect** Keyboard: **BIMFLOWCONNECT**

Connecting different segments to each other Select the segments you want to connect and launch the **BimFlow-Connect** command. If there are multiple options, you can cycle through these by pressing the Ctrl-button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingBIMFlowConnect1 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingBIMFLowCo End the command by pressing Enter, the segments are now connected.

Figure 2.235

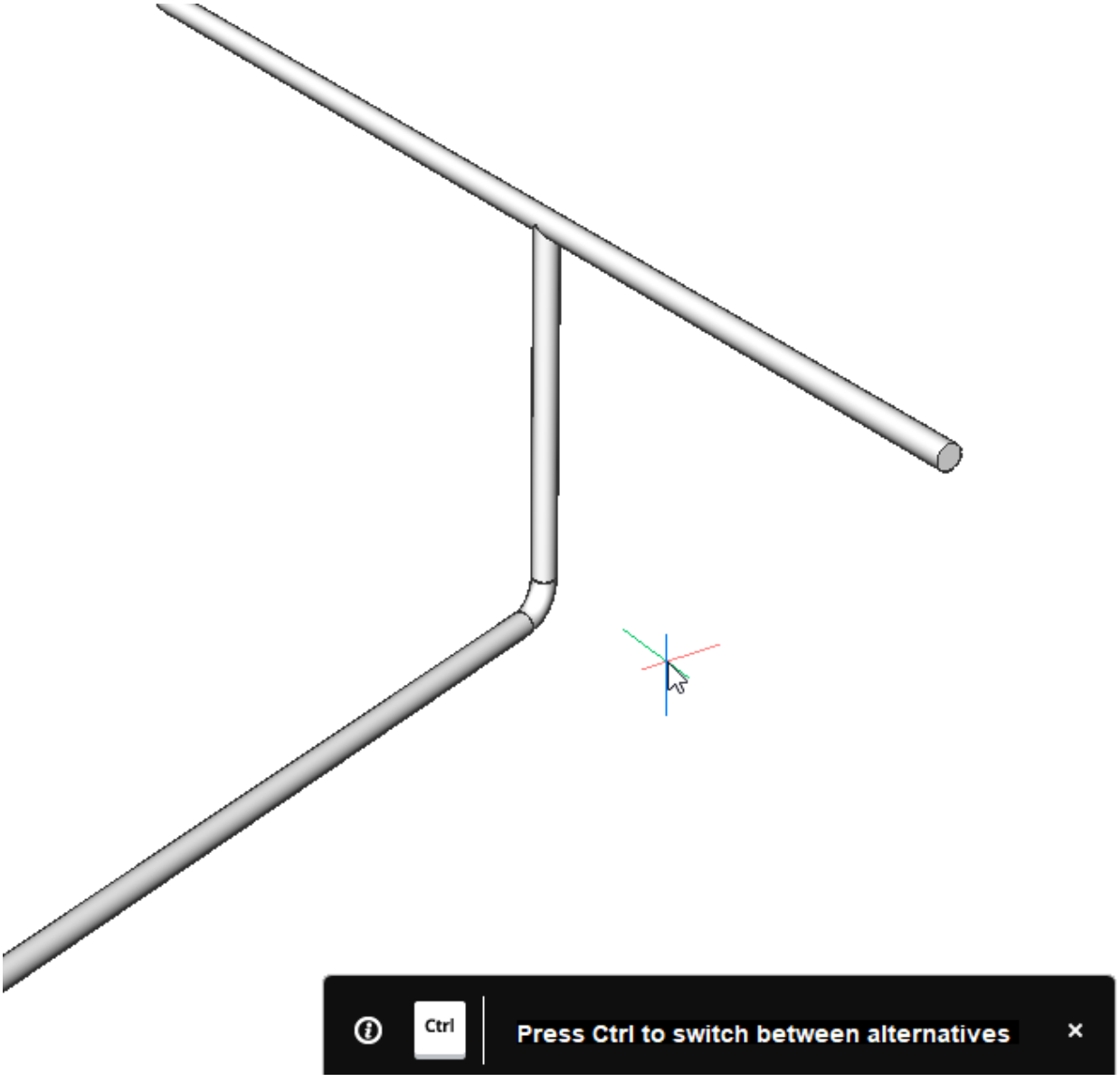


Figure 2.236



About Drag

The **Drag** command makes it possible to alter a segment of a drawn profile without losing its connection to the other segments.  Menubar: **BIM > Drag** Toolbar: **Modify > Drag** Quad: **Model tab > Drag** Keyboard: **DRAG**

Altering the positioning of segments Enable the selection of sides and ends: 

- Click the icon on the **BIM** toolbar.
- Type **DisplaySidesAndEnds,** then type **1** or **ON.**

If you now hover over a segment, you can select a face perpendicular to the section of the profile. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingBIMDrag2.png}) Select a face and launch the **Drag** command. Now you can click a new position for the segment or use the dynamic dimension field to enter a value for the displacement. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/M
Note: The software automatically regenerates the connection between the segments. If you do not want this, you can press Ctrl before choosing the new position of the segment to unconnect the profiles.

Figure 2.237

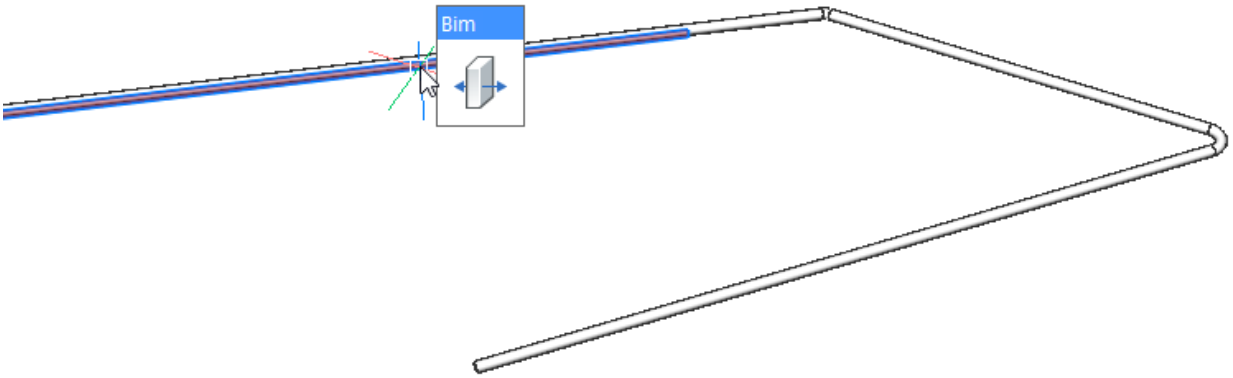
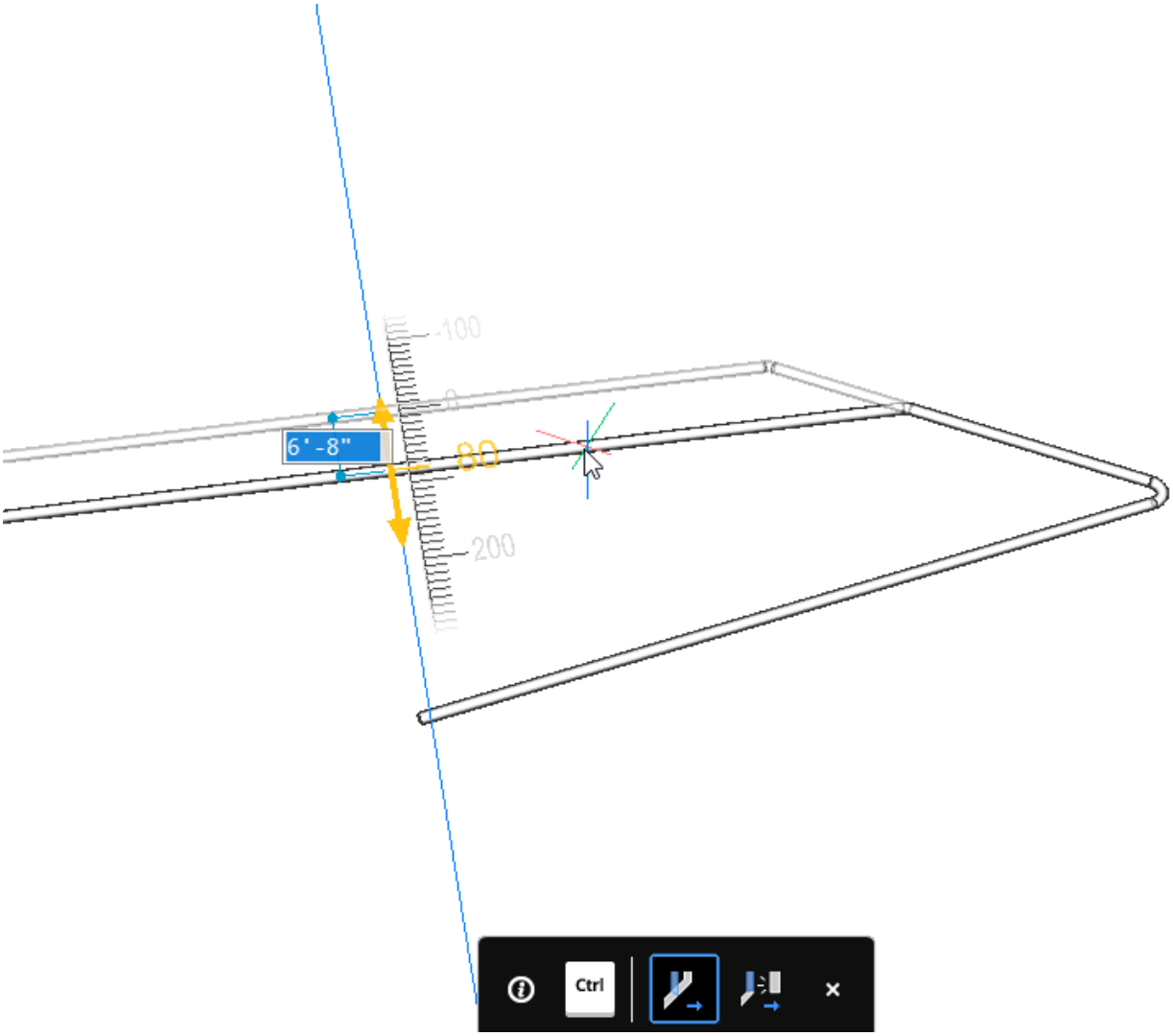


Figure 2.238



Procedure: Using the MEP Modeling tools to draw and connect profiles

1. Open the drawing that contains the MEP model. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/

2. Open the BIM Profiles panel on the right hand side of your screen (see Dockable Panels (1.12.7)) Filter on HVAC and select the Rectangular 12”x 8” profile, drag it to the model space. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingProcedureStep2.png})

3. Define the start point of the profile. Click on the midpoint of the yellow rectangle. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingProcedureStep3.png})

4. Turn **Ortho** on and define the next points of the profiles by using the dynamic dimension field. Go up 3.5, go 3.5 to the left and go 21 horizontally. If necessary, you can make the profile do a quarter turn by entering Q. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingProcedureSt
Note: The software automatically generates the connections between the segments.

5. Connect the other rectangular ducts to the one we have just created. Select the ducts together with the main duct and launch the **BimFlowConnect** ( )command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS

6. Create a duct from the flow terminal. Launch the **BimLinearSolid** ( ) command and select the flow connection point. The connection point contains information, so the software automatically creates a circular duct. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingProced

7. Connect the ducts from the flow terminal to the rectangular duct. Select the main duct and the duct from the flow terminal and launch the **BimFlowConnect** ( ) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h

8. Connect the other flow terminals to the rectangular duct. Select the other flow terminals and the rectangular duct and launch the **BimFlowConnect** ( ) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS
BricsCAD uses the information from the flow connection point to determine that the flow terminals should be connected using a circular duct. **Note:** The flow terminal on the back is connected to the back of the rectangular duct. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelin

Figure 2.239

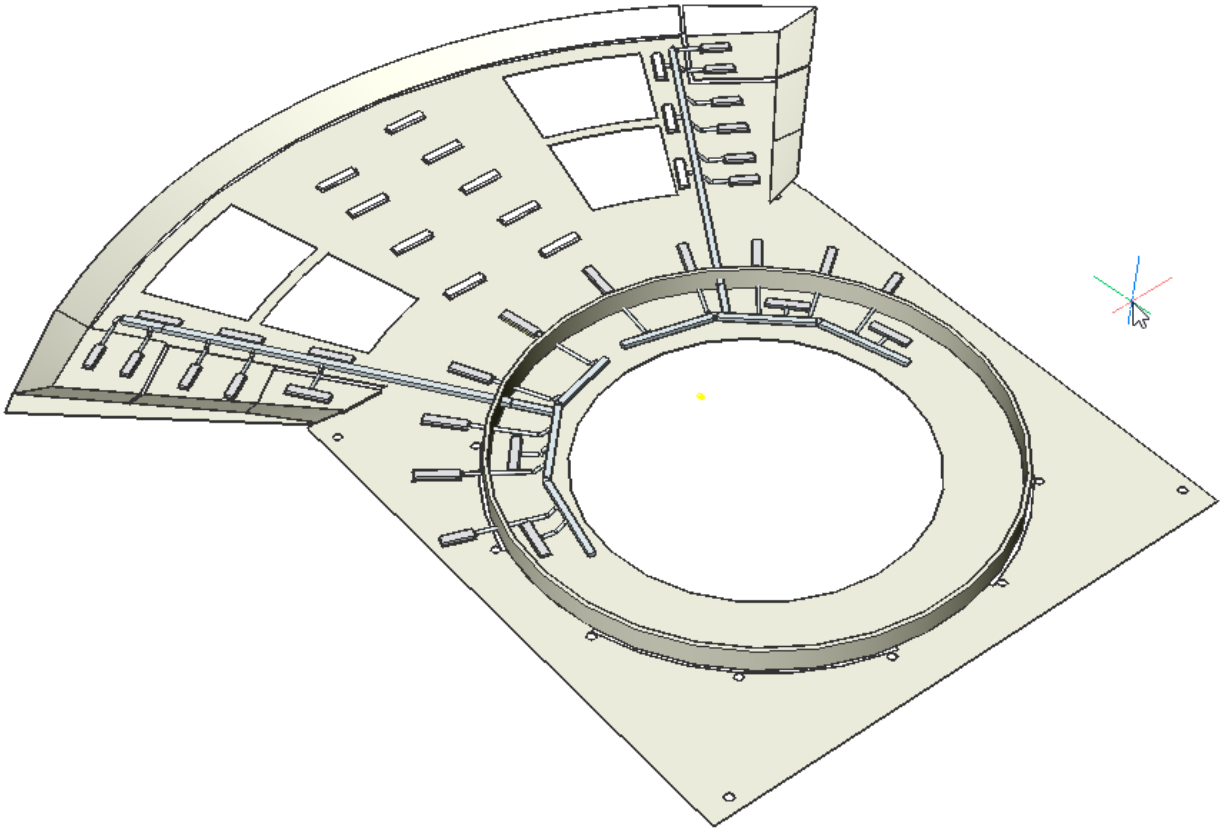


Figure 2.240

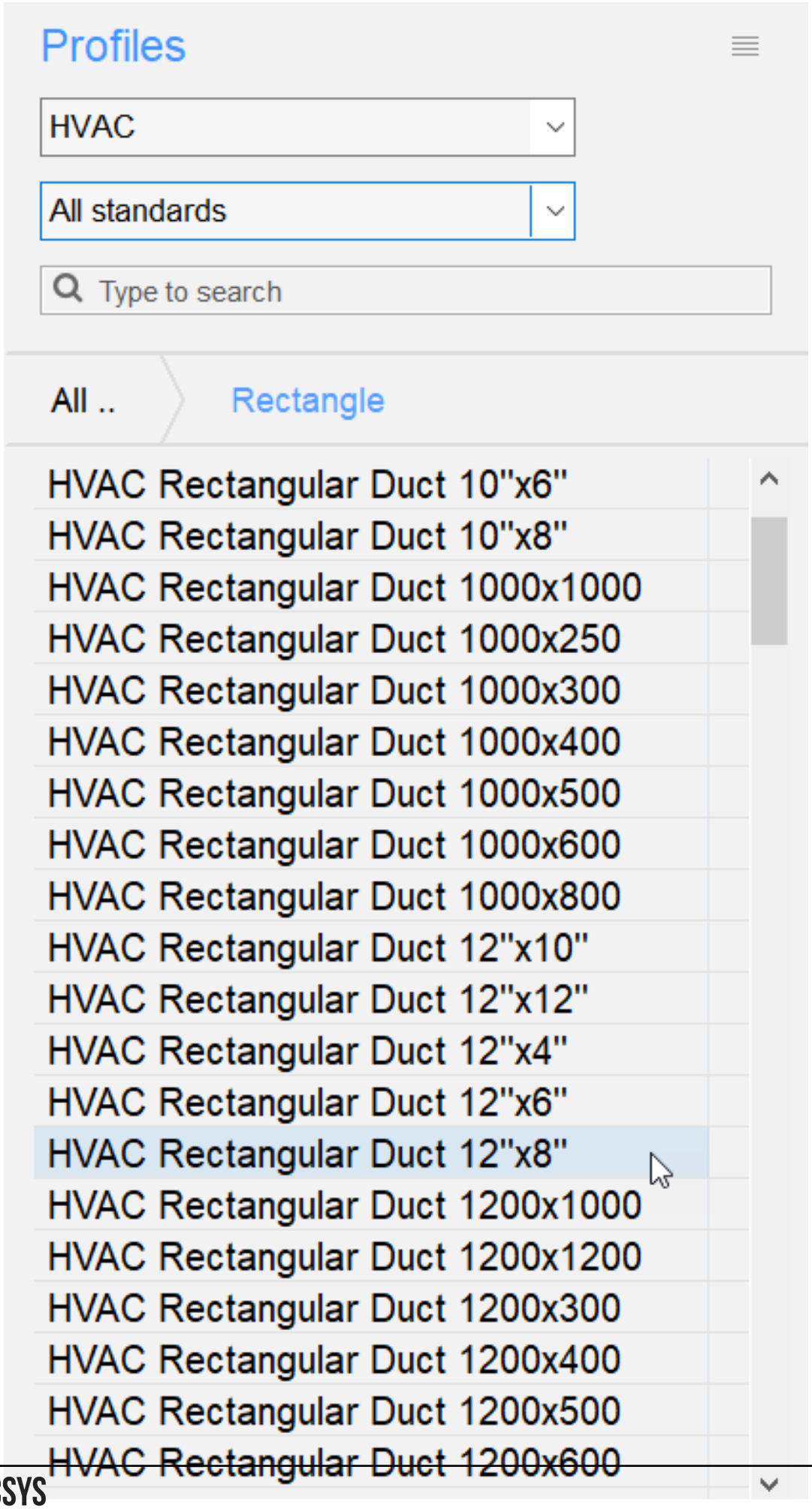

Profiles
HVAC
All standards
Type to search
All ..
Rectangle
HVAC Rectangular Duct 10"x6"
HVAC Rectangular Duct 10"x8"
HVAC Rectangular Duct 1000x1000
HVAC Rectangular Duct 1000x250
HVAC Rectangular Duct 1000x300
HVAC Rectangular Duct 1000x400
HVAC Rectangular Duct 1000x500
HVAC Rectangular Duct 1000x600
HVAC Rectangular Duct 1000x800
HVAC Rectangular Duct 12"x10"
HVAC Rectangular Duct 12"x12"
HVAC Rectangular Duct 12"x4"
HVAC Rectangular Duct 12"x6"
HVAC Rectangular Duct 12"x8"
HVAC Rectangular Duct 1200x1000
HVAC Rectangular Duct 1200x1200
HVAC Rectangular Duct 1200x300
HVAC Rectangular Duct 1200x400
HVAC Rectangular Duct 1200x500
HVAC Rectangular Duct 1200x600

Figure 2.241

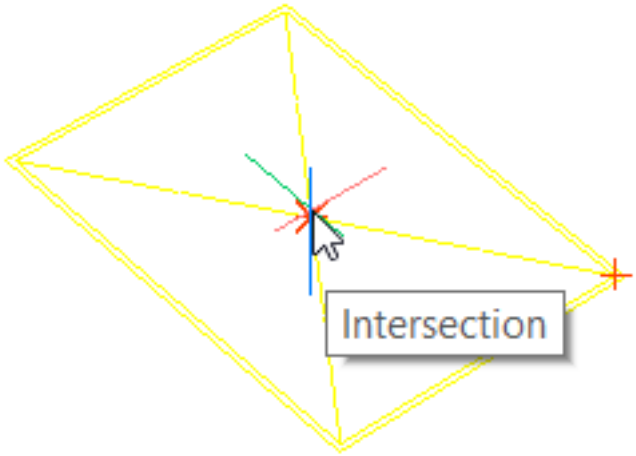


Figure 2.242

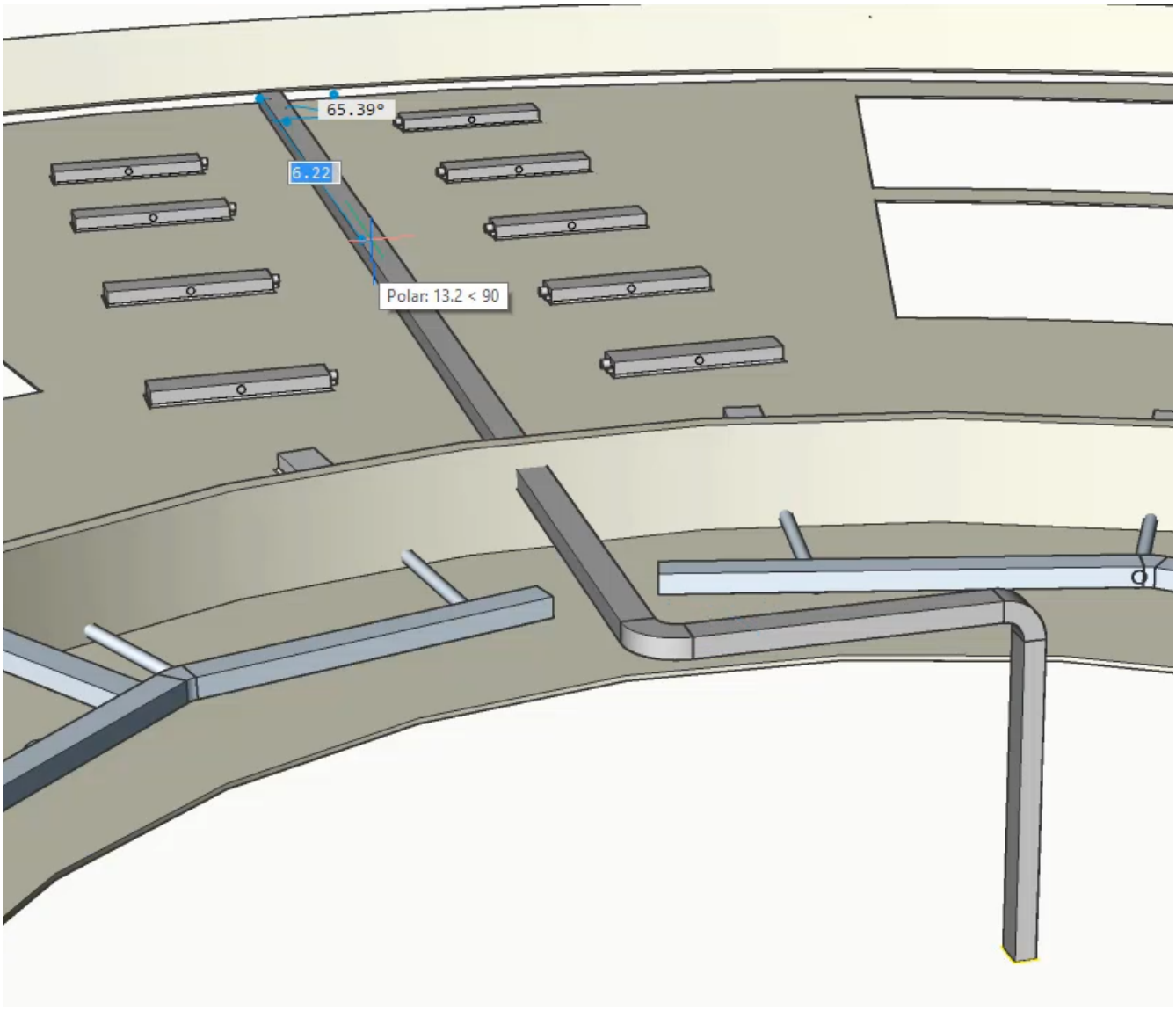


Figure 2.243

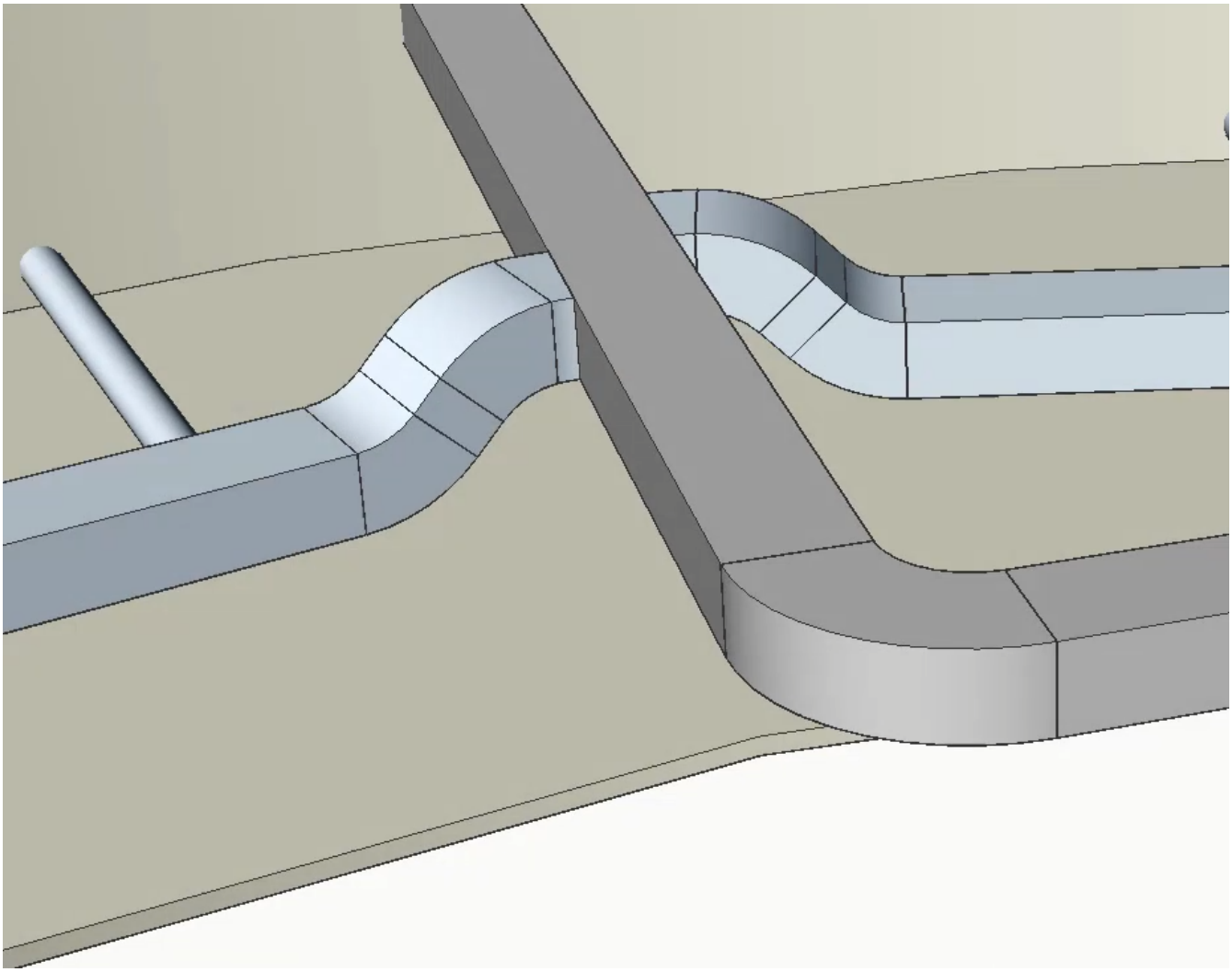


Figure 2.244

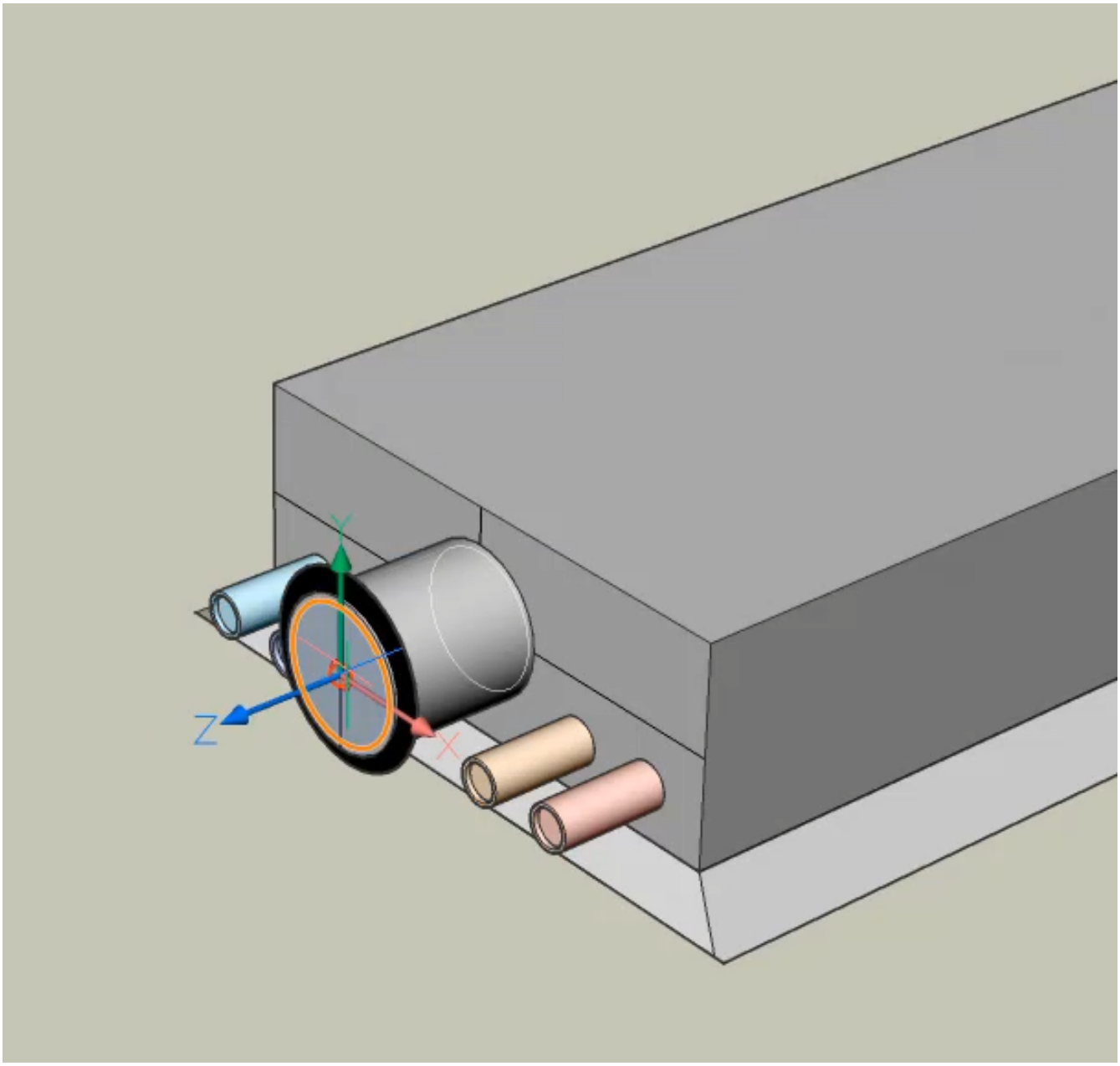


Figure 2.245

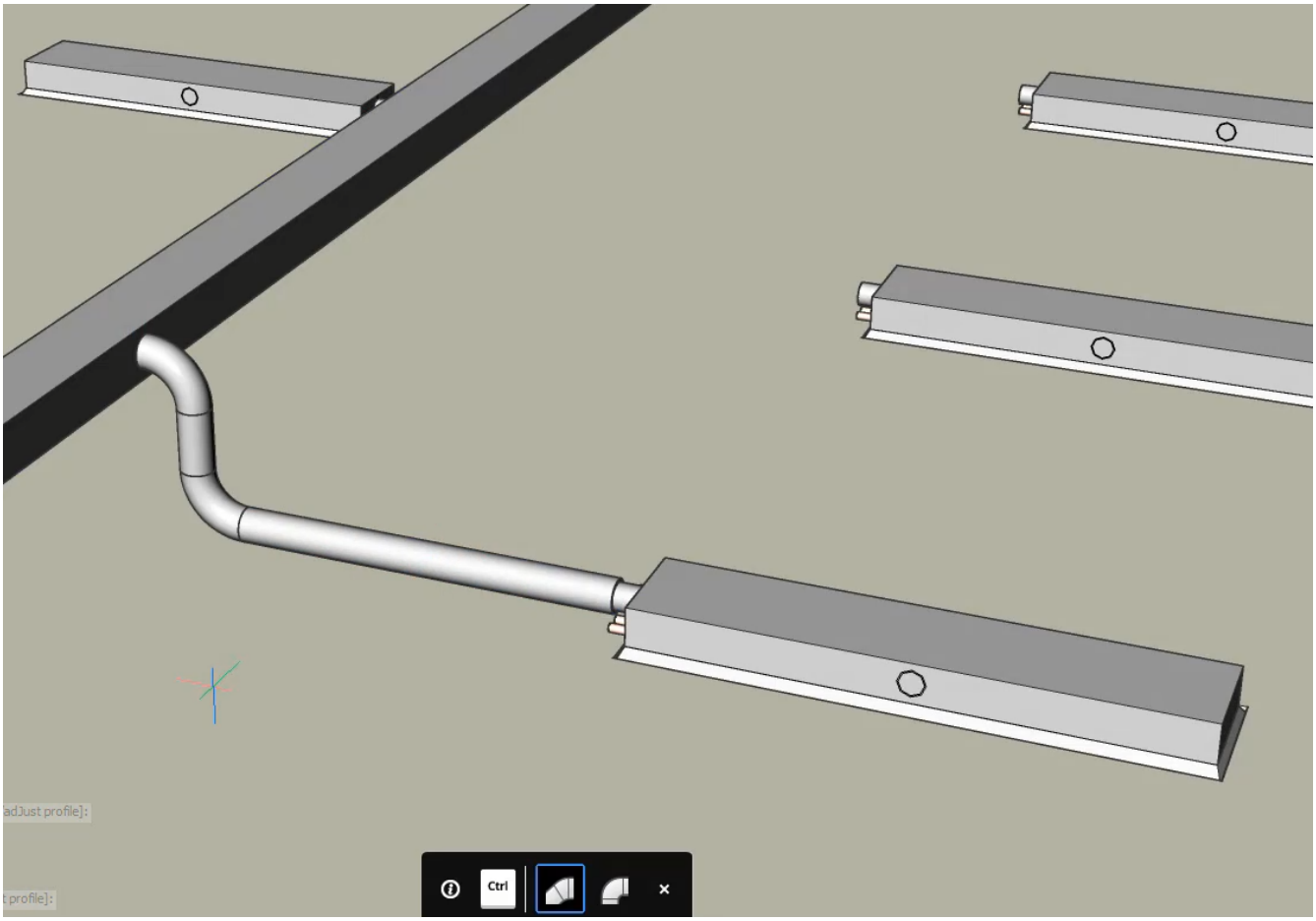


Figure 2.246

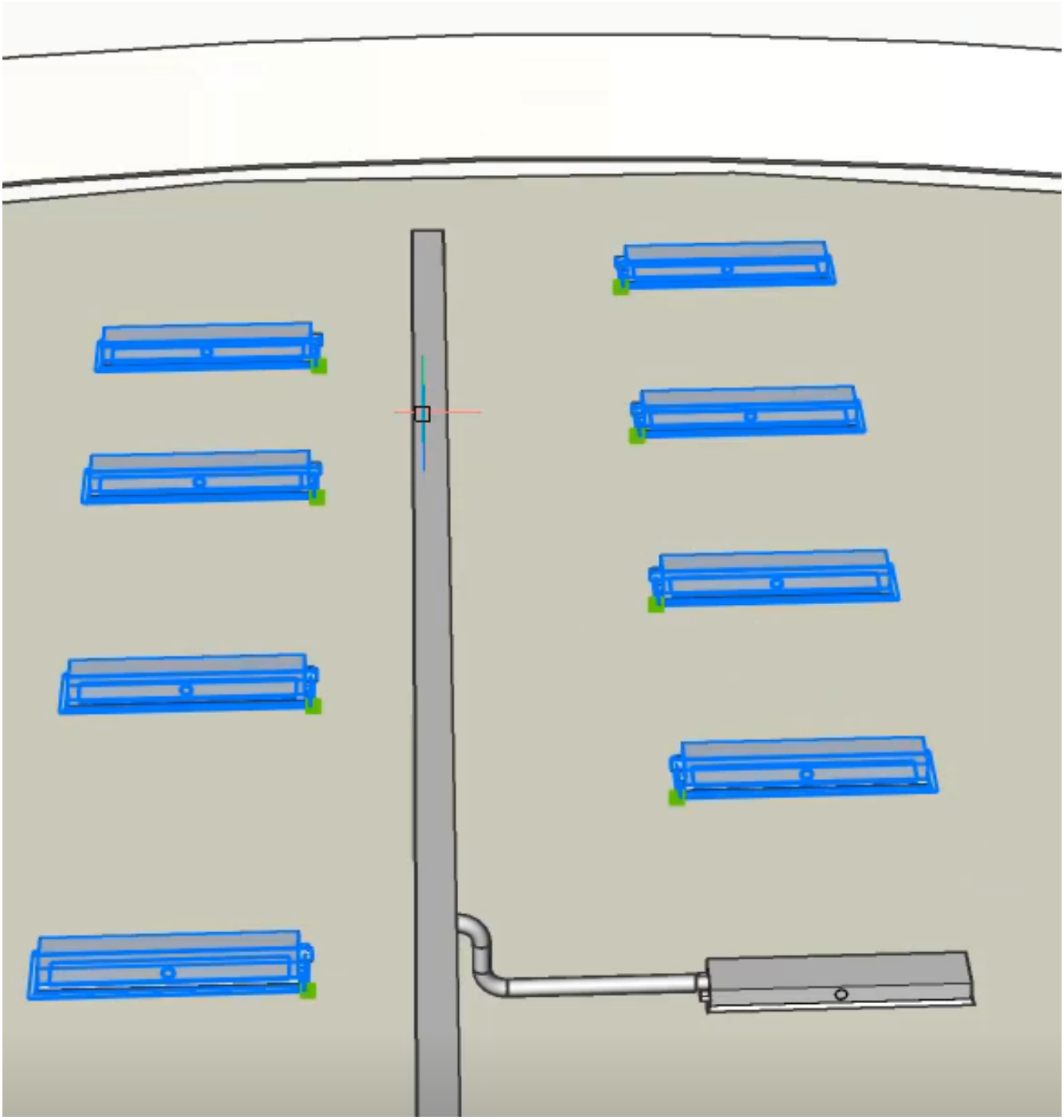
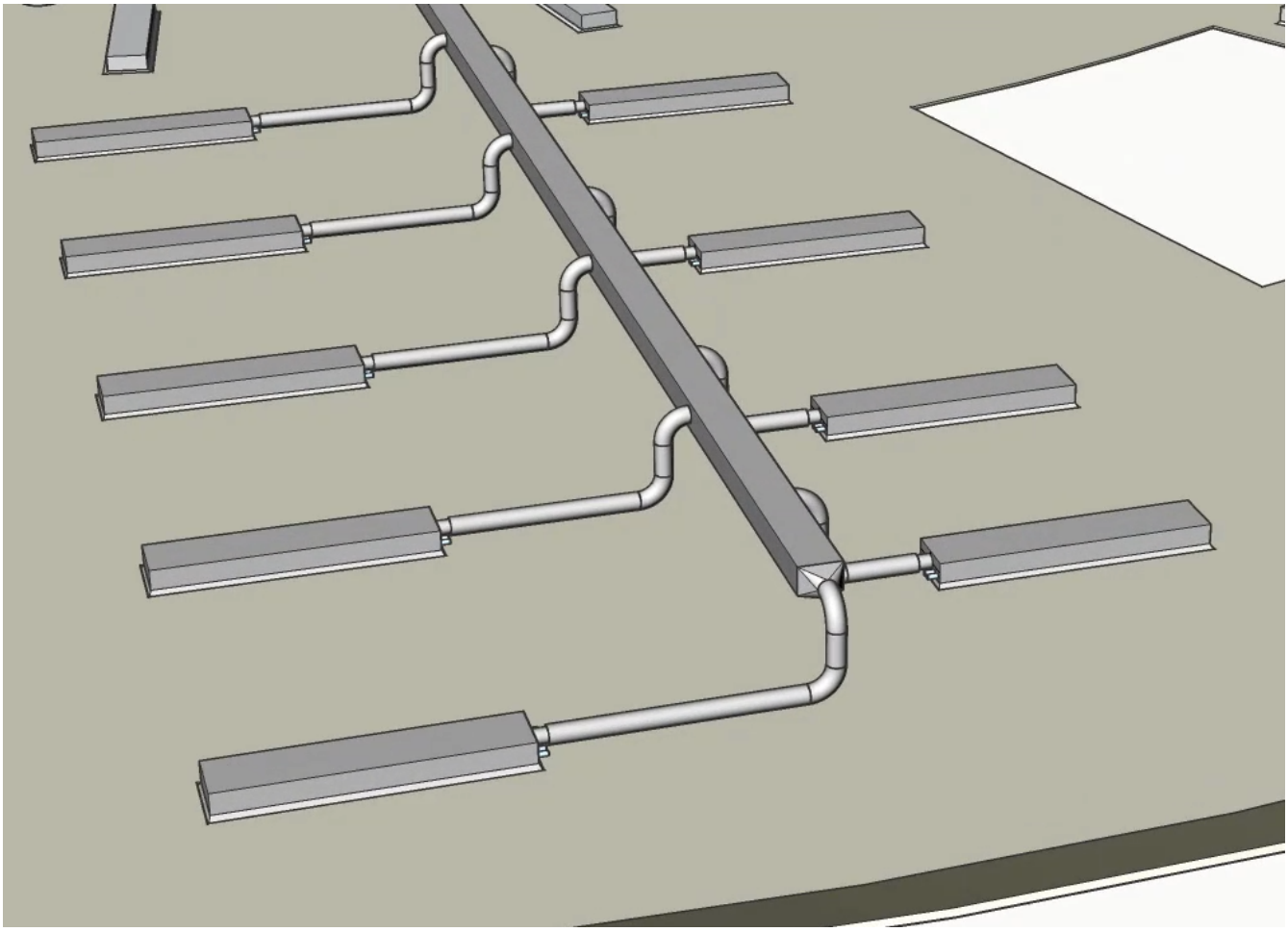


Figure 2.247



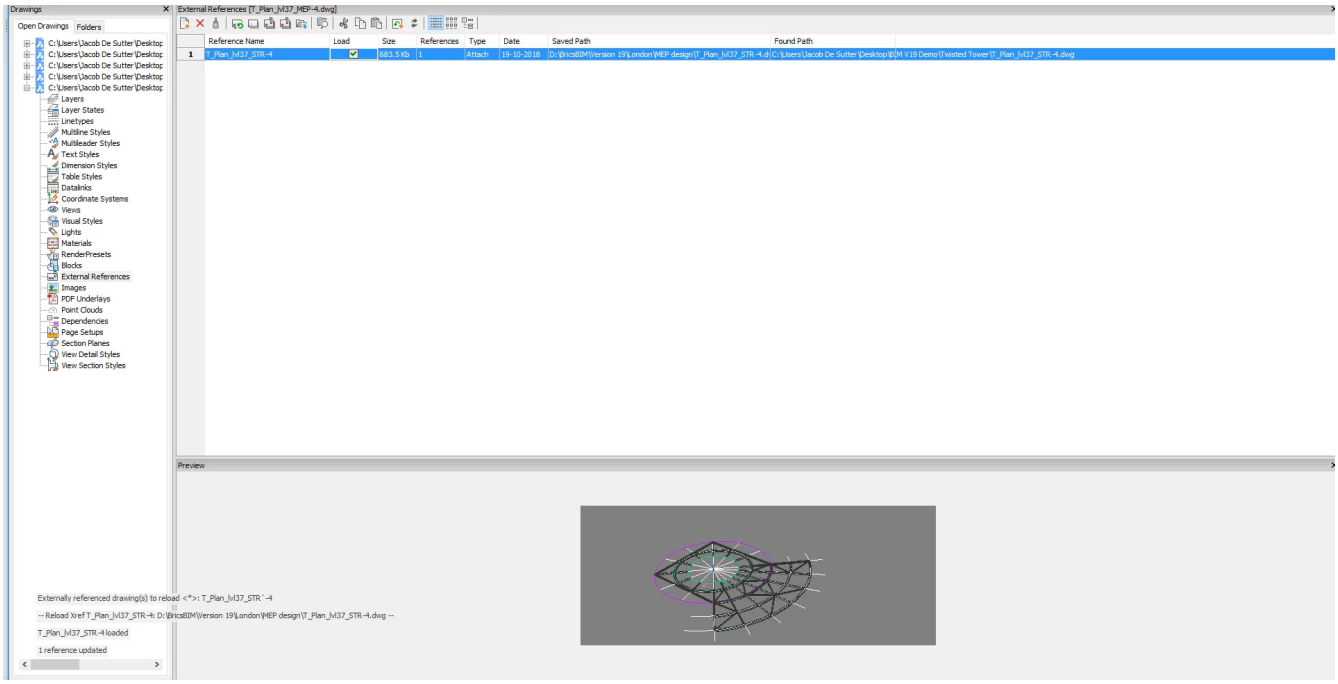
Procedure: Using the MEP Modeling tools to alter profiles.

1. Type **XRef** in the commandline and check the 'load' box next to the structural model. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingProcedureStep9.png})

2. Use **MultiSlice** ( ) to divide the rectangular duct. Enter 3.04 in the dynamic dimensions field and press Enter to end the command.

3. Alter the positioning of the rectangular duct, so it doesn't interfere with the structural model. Select the upper face of the duct and launch the **Drag** ( )command. Note: BricsCAD preserves the connections with the circular ducts, you can change this by pressing Ctrl if you want to unconnect the ducts. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/MEPModelingProcedureStep111.png})

Figure 2.248



Move the duct down 0.23.

4. The structural model doesn't interfere anymore with the MEP Model, while the connection between the ducts of the flow terminal and the main rectangular duct was preserved. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

Figure 2.249

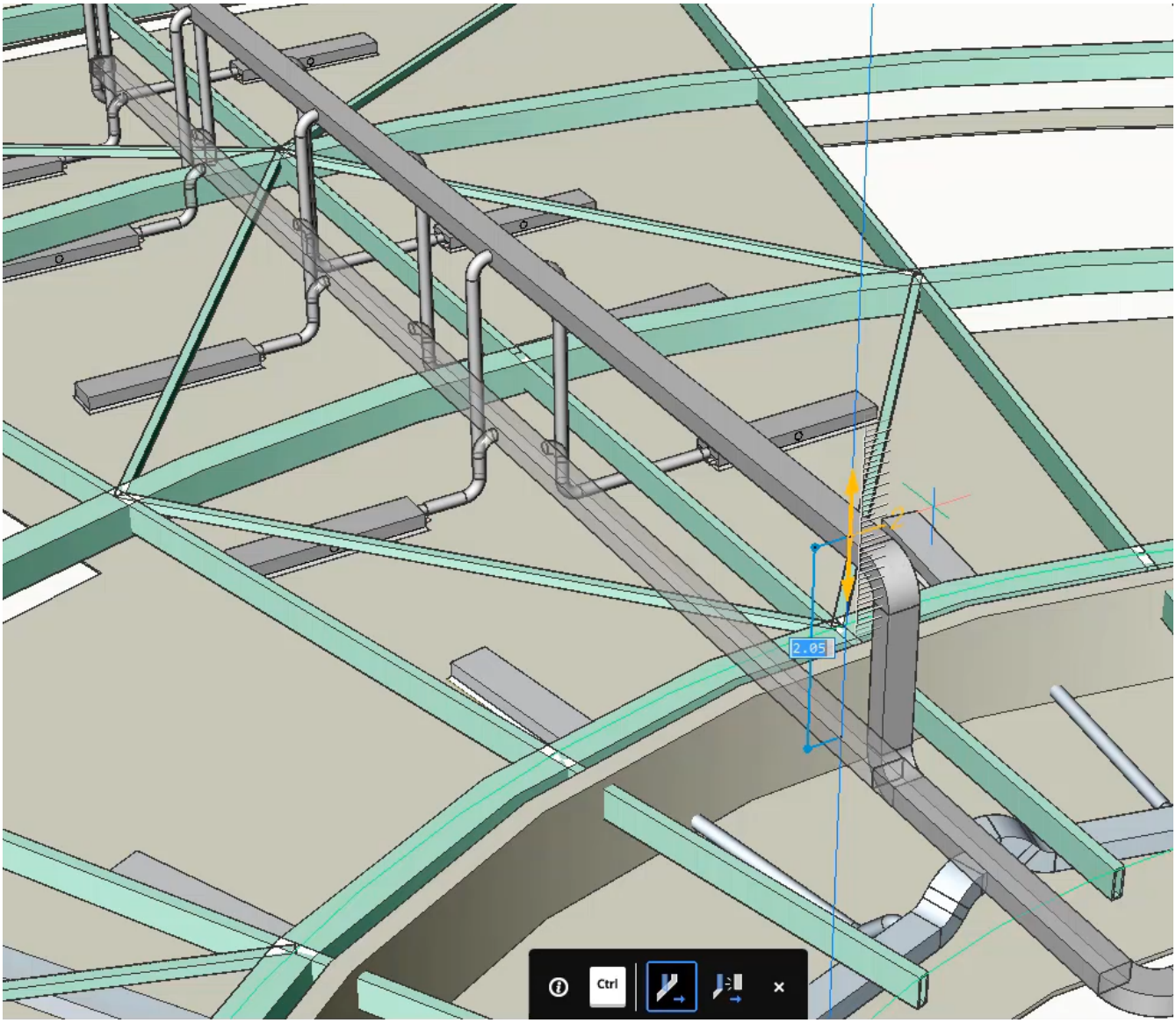
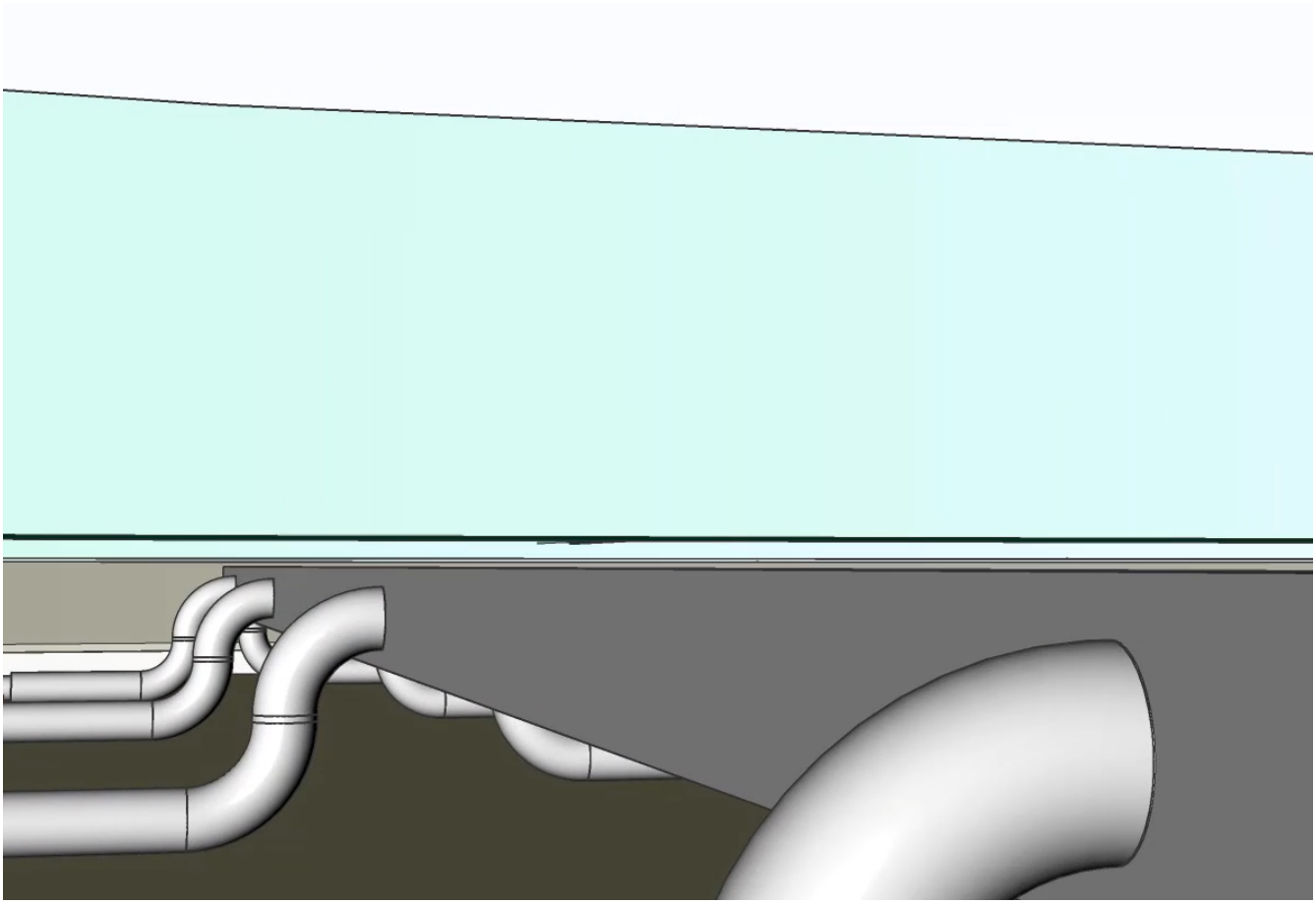


Figure 2.250



2.4.15 Offset

Command: Offset (6.23.3) **Overview:** In this lesson, you learn how to use the Offset tool to create parallel copies of entities. **Lesson Objectives** After completing this lesson, you will be able to:

- Make an offset of a 3D solid face.
- Make an offset of a 3D solid edge.
- Make an offset of a detected boundary.

Tutorial: How to offset Click here (2.3.9) to watch

About

The Offset tool allows you to create copies of linear entities and align them parallel to the original entities at a specified distance. Menubar: **Modify tab > Offset** Toolbar: **Modify > Offset** Quad: **Draw Tab > Offset** Keyboard: **OFFSET** Ribbon: **Model tab > Modify 2D > Offset** Depending on the selected options in the Selection Modes toolbar, the Offset tool allows to create a parallel copy of:

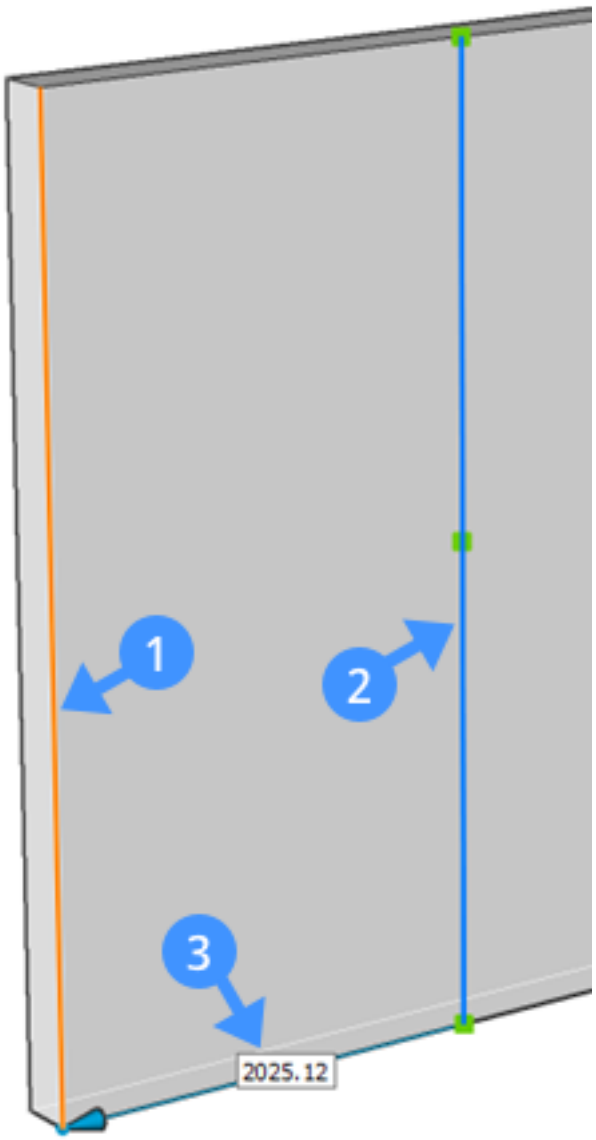
- An edge of a 3D solid if theSelect Edges option ( ) is enabled.
- The edges of a 3D solid face if theSelect Faces option ( ) is enabled.
- Detected boundaries if theSelect Detected Boundariesoption ( ) is enabled.

Procedure: Make a parallel copy of a 3D solid edge

Before starting the process, make sure that Quad is active and the Select Edges optionin the Selection Modes toolbar is on. Next, follow these steps:

1. Move the cursor over the edge of the solid to highlight the edge.
2. Choose the Offset tool from the Quad.
3. Specify the offset distance by dragging the edge of the solid face. **Note:** The distance of the offset displays in the dynamic dimension field The parallel copy of the edge is created dynamically.
4. While dragging the edge to set a precise offset distance, type a distance in the dynamic entry field and press Enter, or left-click to accept the offset. The following illustration shows the offset edge on the 3D solid. The edge highlights in orange is the original edge (1) of the 3D solid, whereas the blue highlighted line with grips (2) is the offset copy of the original one. The line (3) between the original edge and copied edge gives the offset distance. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/OffsetEdge

Figure 2.251

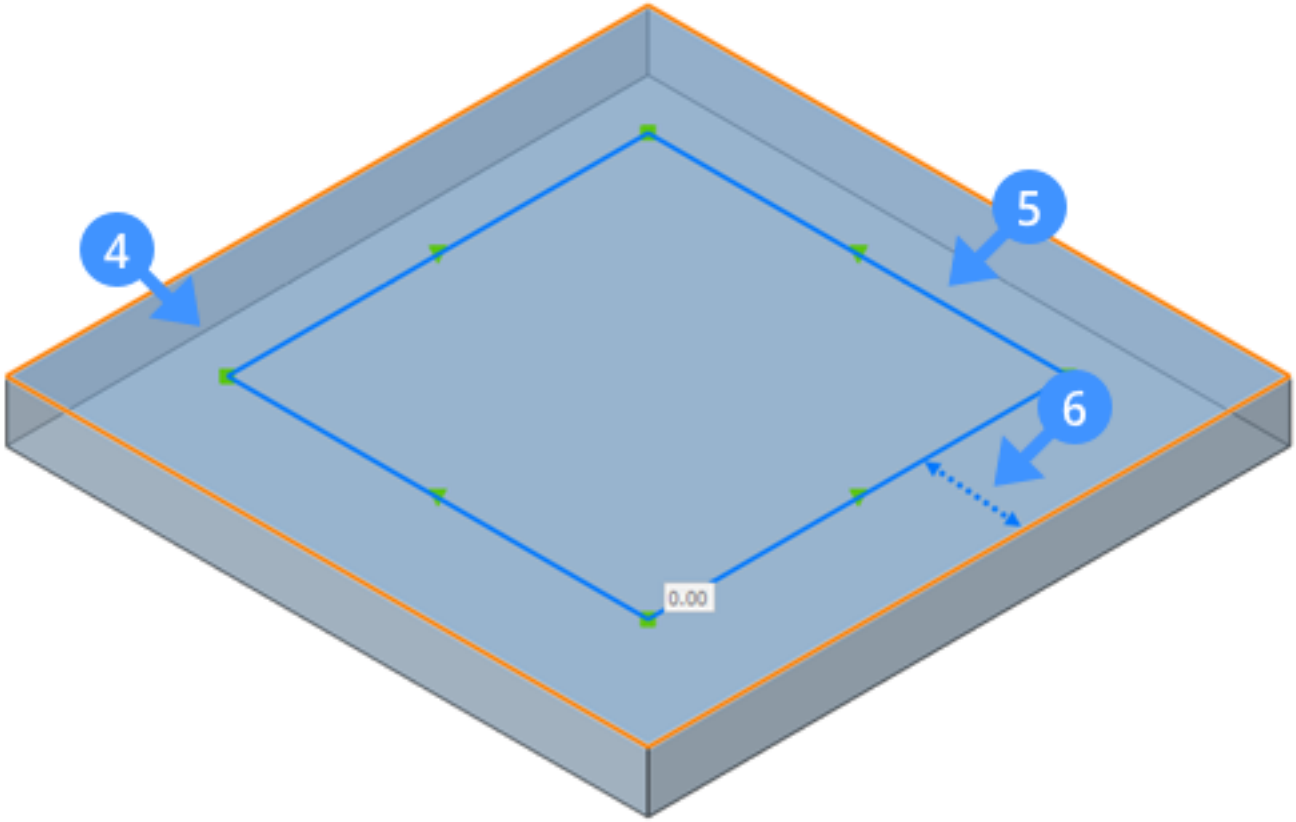


Procedure: Make a parallel copy of a 3D solid face

Before starting the process, make sure that Quad is active and the Select Faces option in the Selection Modes toolbar is on. Next, follow these steps:

1. Move the cursor over the face of the solid to highlight the solid face.
2. Choose the Offset tool from the Quad.
3. Specify the offset distance by moving the cursor outside or inside of the solid. The parallel copy of the face is created dynamically.
4. Enter a distance in the dynamic entry field and press Enter, or left-click to finish the offset. The following illustration shows the offset of the 3D solid face. The highlighted face is the original 3D solid face (4), whereas the created polyline (5) is the offset from the original one. The line (6) between the original 3D solid face and copied polyline gives the offset distance. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/M

Figure 2.252

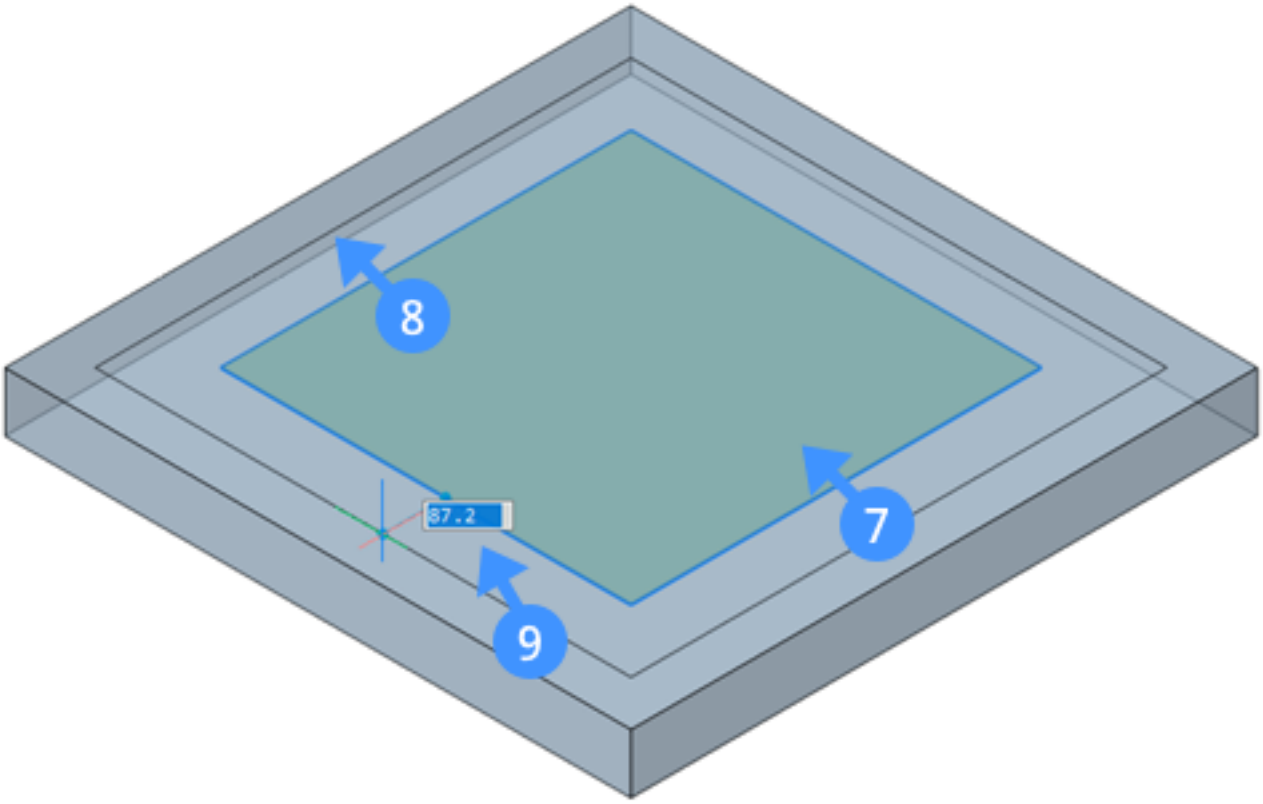


Procedure: Make a parallel copy of a detected boundary

Before starting the process, make sure that Quad is active and the Select Detected Boundaries option in the Selection Modes toolbar is on. Next, follow these steps:

1. Move the cursor inside the boundary to highlight the detected boundary.
2. Choose the offset tool from the Quad.
3. Move the cursor to define the offset distance and left-click. The parallel copy of a detected boundary is created dynamically.
4. Enter a distance in the dynamic entry field to set a precise offset distance and press enter, or left-click to accept the offset. The following illustration shows the offset boundary on the face of a 3D solid. The highlighted face is the detected boundary (7), whereas the created polyline (8) is the copy from the original one. The line (9) between the detected boundary and copied polyline gives the offset distance. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/OffsetBoundary.png})

Figure 2.253



2.4.16 Splitting Solids

Command:BimSplit (6.7.32) TheBimSplitcommand either splits segmented solids into separated solids automatically or allows to split a solid using a selection of cutting faces.

To split a solid automatically

Make sure that the value ofSelectionPreview (1.13.11)is 2 or 3.

1. Do one of the following:
   - Click theBimSplittool button ( ) on the Model tab of the ribbon.
   - ChooseSplitin theBIMmenu.
   - TypeBIMSPLIT in the command bar, then press Enter. You are prompted: Select solid for auto-splitting or cutting faces for manual splitting:
2. Place the cursor over a face of the solid. The face highlights.
3. Without moving the mouse, hit the TAB key until the complete solid highlights, then click. **You are prompted:** Entities in set: 1 Select solid for auto-splitting or cutting faces for manual splitting:
4. Right click. The selected solid is split. The number of parts is reported in the command window.

Using the Quad Make sure theSelect Facesoption ( ) of theSelectionModes (1.13.11)system variable is switched off. Place the cursor over a face of the solid. When the solid highlights, chooseSplit( ) in theBIMcommand group of the Quad. The solid is split. The number of parts is reported in the command window.

To split a solid using cutting faces

Make sure that the value ofSelectionPreview (1.13.11)is 2 or 3.

1. Do one of the following:
   - Click theBimSplittool buttonon Model tab of the ribbon.
   - ChooseSplitin theBIMmenu.
   - Typebimsplitin the command bar, then press Enter. You are prompted: Select solid for auto splitting or cutting faces for manual splitting:
2. Place the cursor over a face of the solid. Hit the TAB key to highlight an obscured face. When the face highlights, click to select the face. **You are prompted:** Entities in set: 1 Select solid for auto splitting or cutting faces for manual splitting:
3. (option) Repeat step 4 to select more faces.
4. Right-click to stop selecting faces. The solid is split using the selected faces as cutting planes. The number of parts is reported in the command window.

Using the Quad Make sure theSelect Facesoptionof theSelectionModes (1.13.11)system variable is set.

1. Place the cursor over a face you want to use as a cutting plane. When the face highlights, do one of the following:
2. Go to step 3.
3. Go to step 2 to select more faces.
4. (option) Place the cursor over a face you want to use as a cutting plane. When the face highlights, do one of the following:
5. Go to step 3.
6. Repeat this step to select more faces.
7. ChooseSplitin theBIMcommand group of the Quad. The solid is split using the selected faces as cutting planes. The number of parts is reported in the command window.

2.4.17 Using BIM Drag

Command: BimDrag (6.7.8) **Overview:** In this lesson, you learn how to move a 3D solid that is perpendicular to the selected face using the BIM Drag tool. Furthermore, you learn how to preserve connections between solids when BIM DRAG is executed on a face. **Lesson Objectives** After completing this lesson, you will be able to:

- Move 3D solids along the normal of their selected face.
- Preserve connections between solids while dragging a single solid or multiple solids.
- Increase or decrease the geometry of the solid in size according to the process you use.
- Disable the connection between solids using the CTRL key.

Tutorial: How to use BIM Drag Click here (2.3.27) to watch

About

The BIM Drag tool allows you to move walls, slabs or linear elements on the highlighted face of a solid. For instance, BIM Drag tool can be used to move linear solids by highlighting/selecting the end or side face of a linear solid. Depending on the modeling operation, the connection between solids can be either preserved while dragging one or more solids at a time or solid (s) can be moved independently by disjoining the connectivity to other solid(s).

Procedure: Using BIM Drag on a 3D Solid

1. Highlight/select any faces of the solid and select BimDrag (6.7.8) from the Quad (**Quad:** Model tab). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMDragonsurface.png})

2. Move the cursor to adjust the position of the moving solid in the dynamic dimension field, then click or enter a precise value in the dynamic dimension field and press Enter. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
Note: A ruler appears to visually represent the distance over which you will move the entity.

3. The Hotkey Assistant appears at the bottom of the screen showing the available options.(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMDRAGHotkeyAssistant.png}) **Note:** The default mode is indicated by a blue frame.

- Select the **Enable connectivity mode(1)** option, to move the solid while preserving the connections between solids. In the following illustration, the solid behavior is shown before and after applying the BIM DRAG in **"Enable connectivity mode"**. The connectivity between walls, i.e. T- (3) and mitered (4) connections are preserved after dragging an inner wall surface (5). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/B
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMDRAGEnableCon
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMDRAGEnableCon

- Select the **Disable connectivity mode (2)** option to move the solid while breaking the connections between solids. **Note:** Hit the CTRL key to switch the connectivity modes from one to another. Alternatively, type "**d**" to disable connectivity or type "**e**" to enable connectivity. In the following illustration, the solid behavior is shown before and after applying the BIM DRAG in **"Disable connectivity mode".**The connectivity between walls, i.e. T- (3) and L- (6) connections are disjointed after dragging an inner wall surface (5). **(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMDRAGDisableConnectivityMode1.png}**
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMDRAGDisableC
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMDRAGDisableC

Figure 2.254

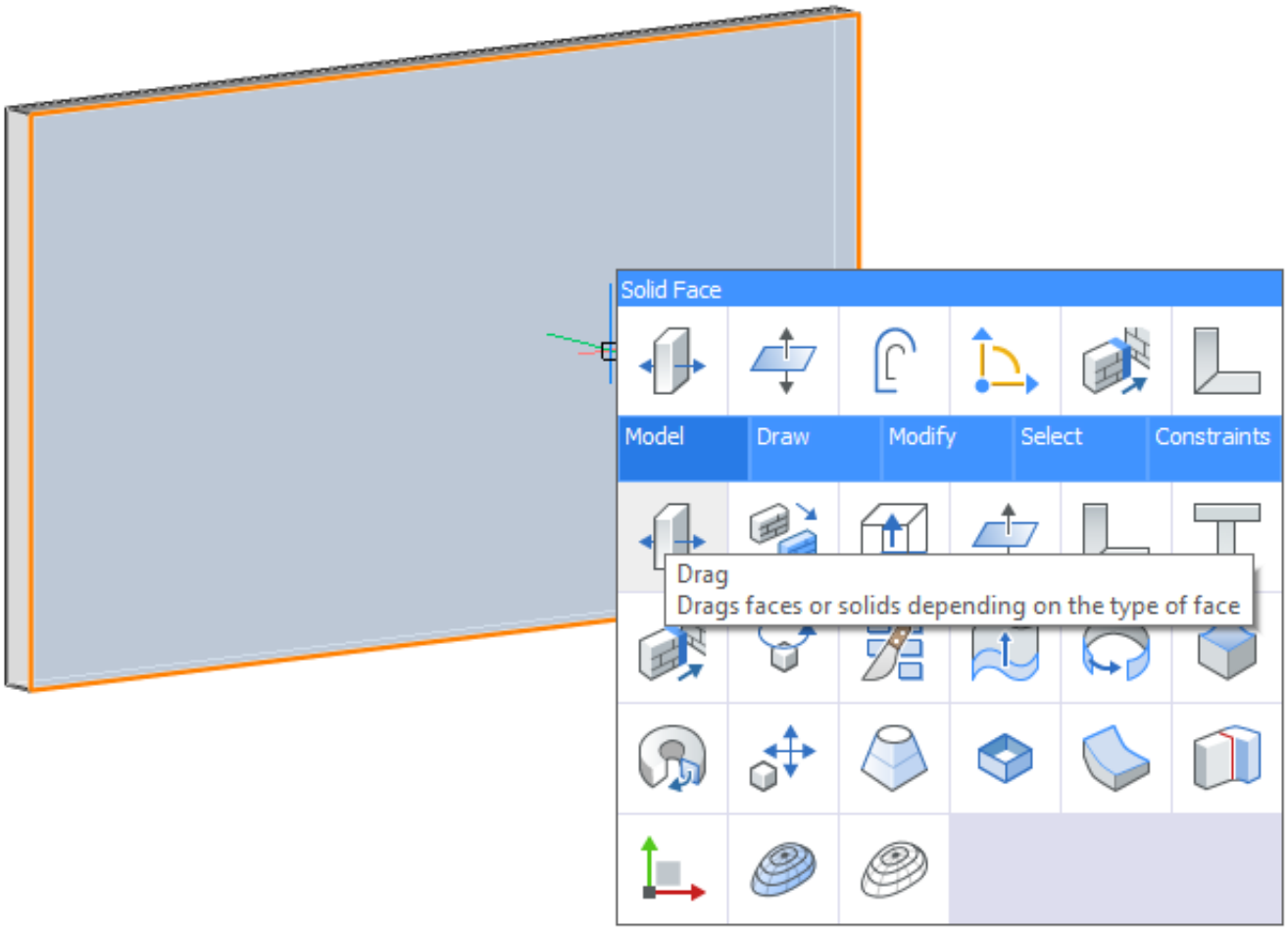


Figure 2.255

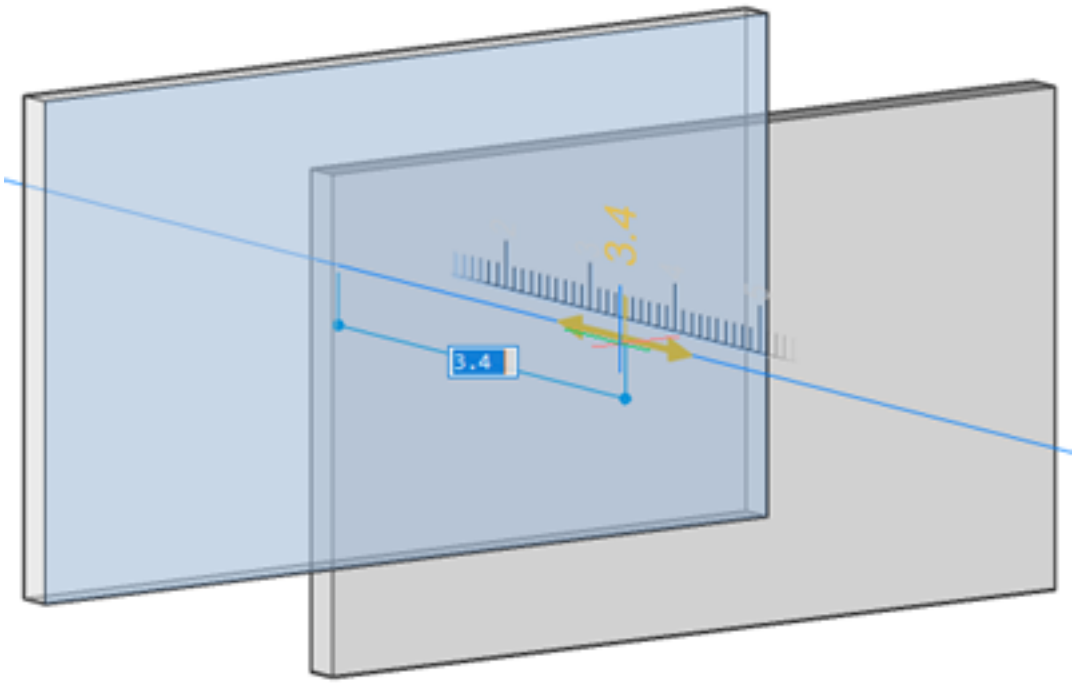


Figure 2.256



Figure 2.257

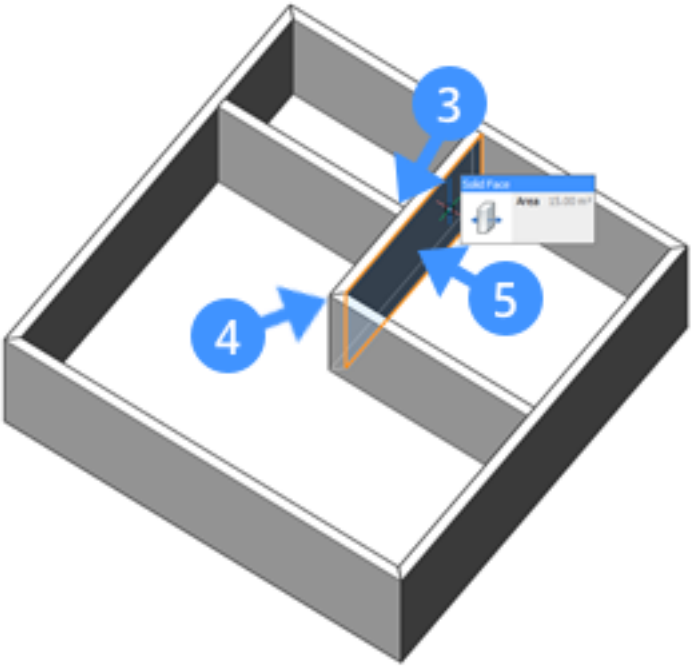


Figure 2.258

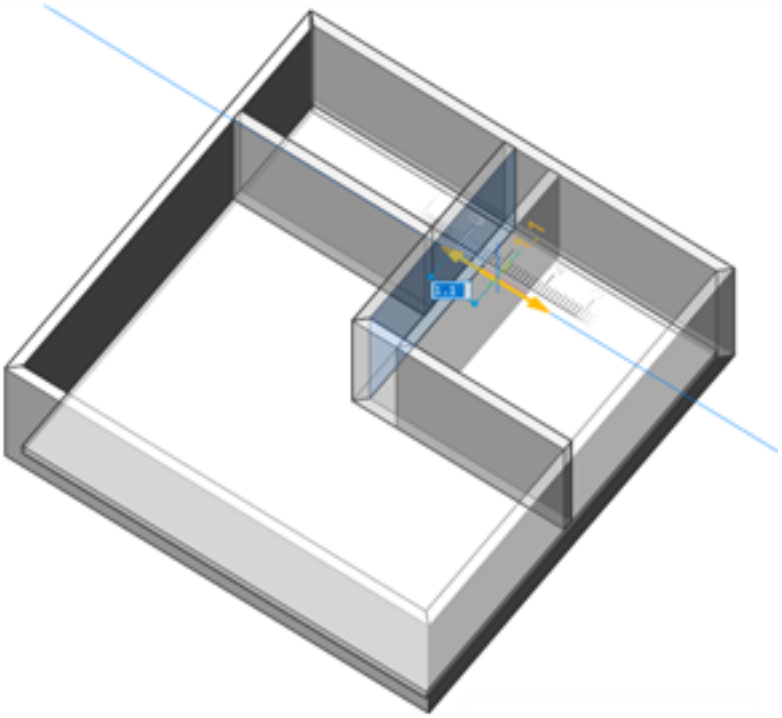


Figure 2.259

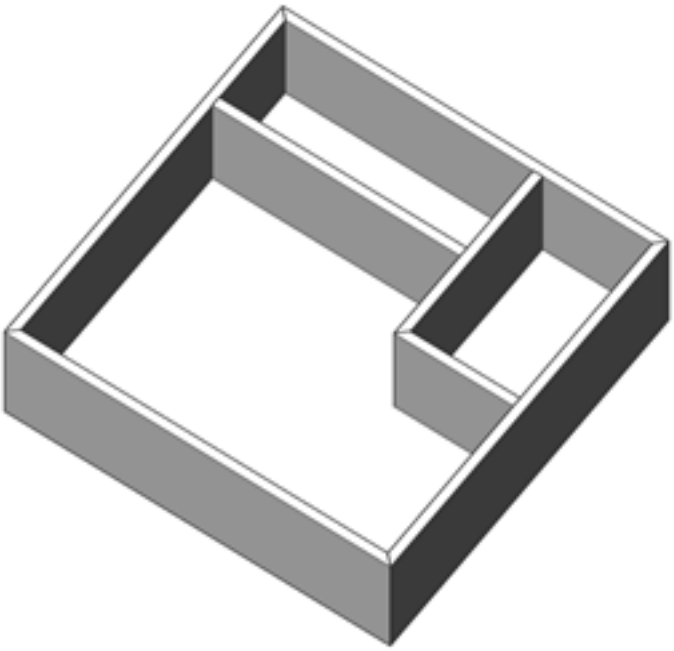


Figure 2.260

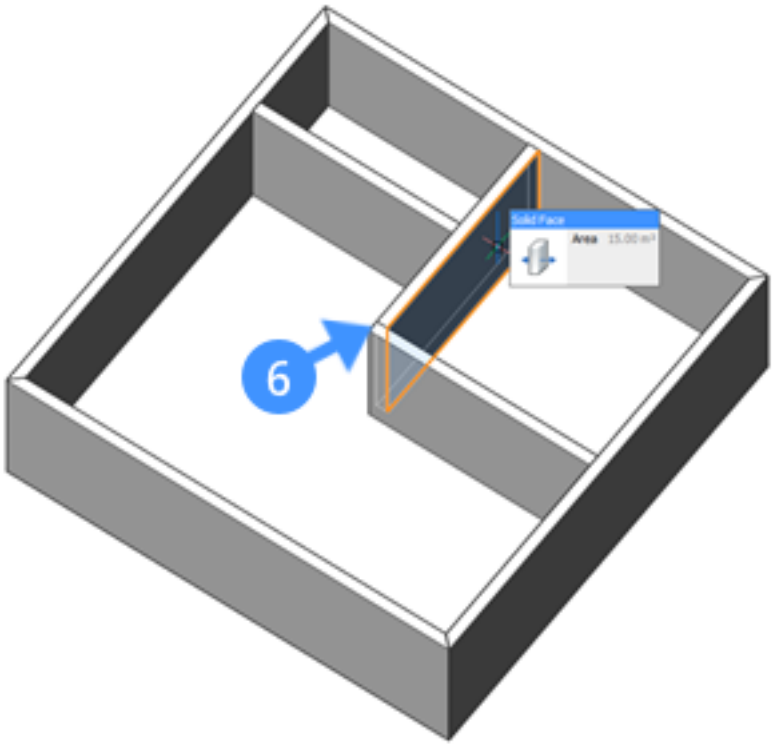


Figure 2.261

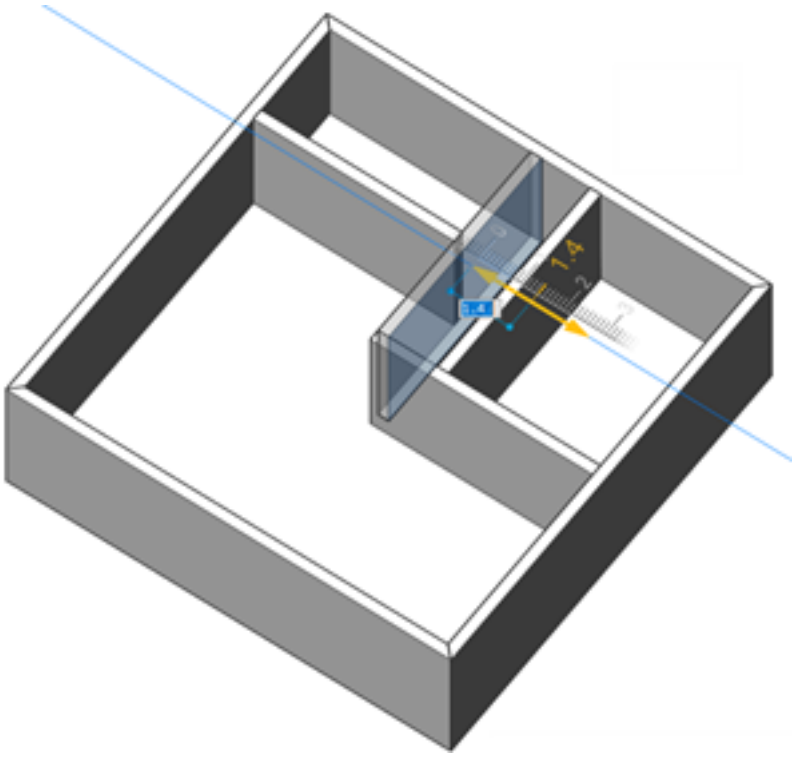
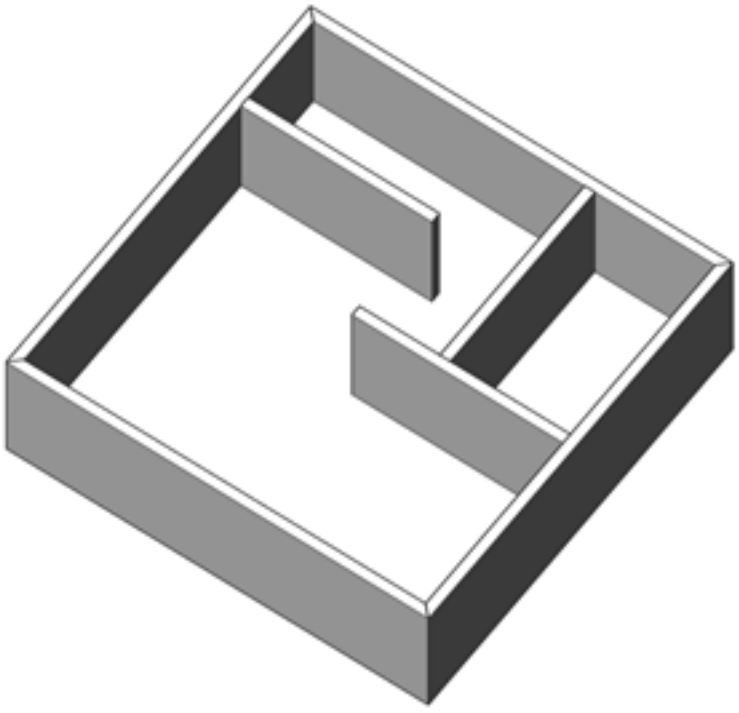


Figure 2.262



2.4.18 Using Parametrize

Command: Parametrize, (6.24.6)ParametersPanelOpen (6.24.8) **Overview:** In this lesson, you learn how to automatically parametrize pure geometric models using the Parametrize tool. **Lesson Objectives:** After completing this lesson, you will be able to:

- Use Paramettrize to

Tutorial: How to automatically Parametrize objects Click here (2.3.21) to watch

About

Parametrizing is a process that automatically adds constraints and parameters to a set of 3D solids, without any manual work. For each unique value, a specific parameter is created.The constraints and parameters are represented by their name and expression in the Mechanical Browser.Each parameter can be animated to show what the parameters correspond to in the model. The parametrized geometry becomes controllable in terms of shape and size.When one of the parameters is changed, the geometric model will be updated automatically.

Procedure: Parametrizing a component automatically

1. Open the geometric model that represents your component. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help **Figure 1, a geometric model of the sample chair**

2. Open the Mechanical Browser clicking its icon ( ) in the Toolbar Panel. **Note:**If the Browser icon is not displayed in the Toolbar Panel,place the cursor on a toolbar, then right click and chooseMechanical Browser (1)in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/Co If the model does not contain any parameters and constraints it will display only the 3D solids under the Bodies (2) section. If the model contains any constraint or parameter, the parameterized tool ask whether you want to reuse the existing ones (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechnique

3. Launch the PARAMETRIZE tool in the Constraints tab from the Quad. **Or**type PARAMETRIZE in the command line. **You are prompted: Select entities to auto constrain:**

4. Select the component geometry you want to parametrize, and press Enter. (\vref {fig:C:/Dev/Task-2448/images/staticbri **Figure 2, selected object** A set of parameters and constraints are defined automatically. These parameters and constraints are shown in the Mechanical Browser (3, 4). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help **Note:** Parametrize creates Distance, Radius, Rigid Sets, Fix, Coincidence, Concentric constraints (2.4.19) depending on the geometric model. **Note:** Perpendicularity and tangency can be set (if they are applicable) by changing the DmRecognize (1.18.3) system variable in the Settings dialog.

5. To check easily which parameter corresponds to what, right-click one of the parameters, and choose Animate (5) in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniqu **Note:** In the attached .dwg file after parametrizing the chair geometry, parameters and constraints will be assigned and shown in the Mechanical Browser. Each parameter corresponds to one measurementin the chair geometry. For instance, Length_X_Y controls the width and depth, Length_Z controls the overall height,p_1 controls the seat thickness p_2 controls the leg thickness ratio_1 controls the ratio between seat height and overall height.

Figure 2.263

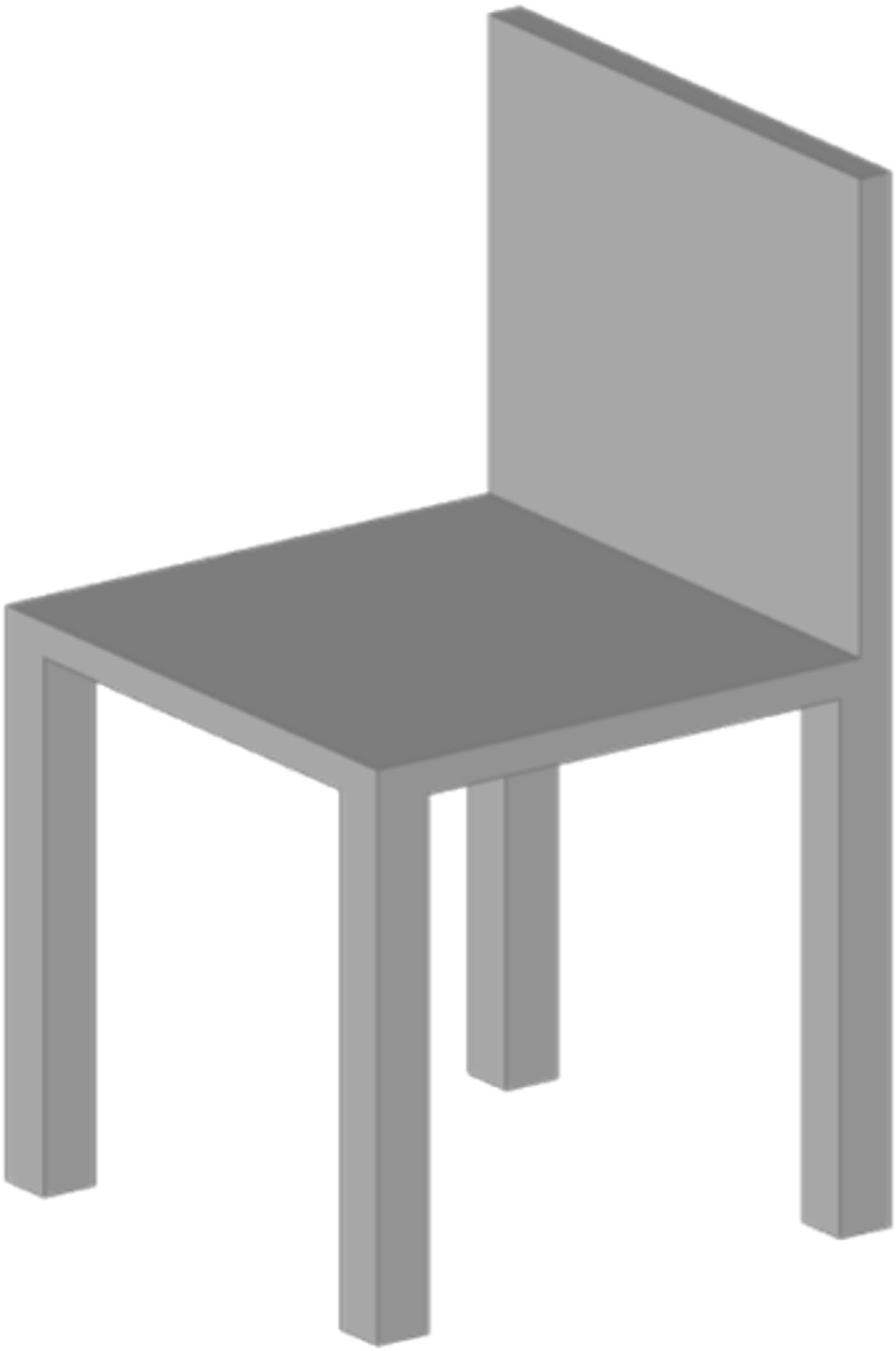


Figure 2.264

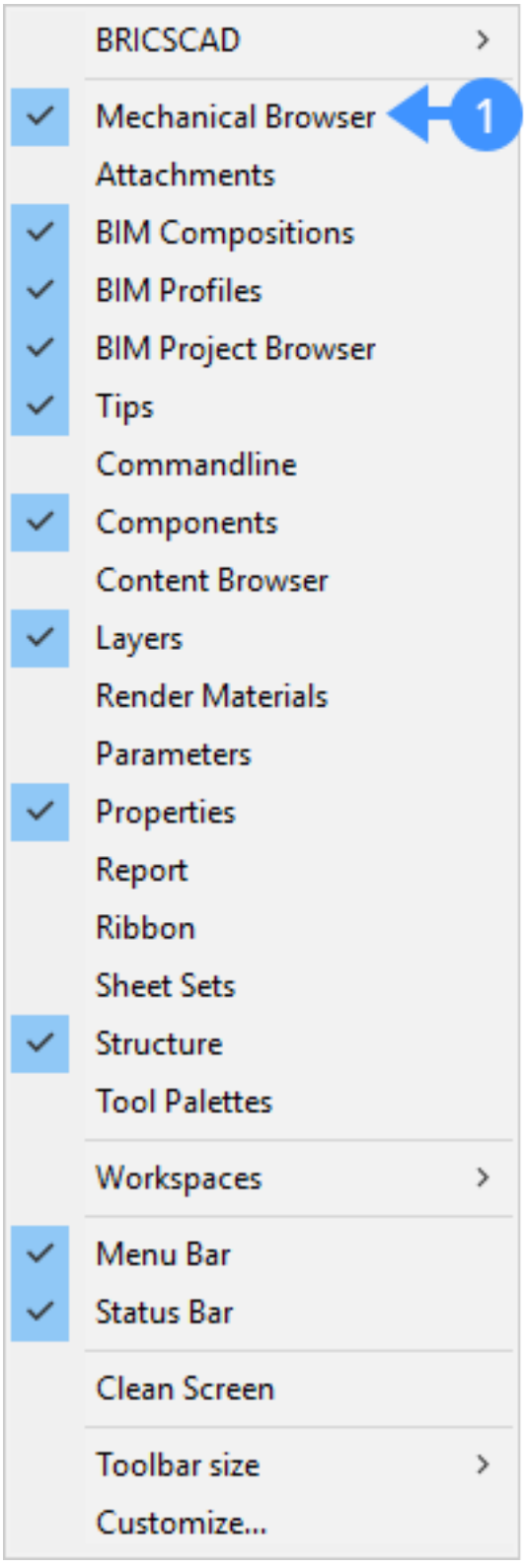

BRICSCAD
Mechanical Browser
1
Attachments
BIM Compositions
BIM Profiles
BIM Project Browser
Tips
Commandline
Components
Content Browser
Layers
Render Materials
Parameters
Properties
Report
Ribbon
Sheet Sets
Structure
Tool Palettes
Workspaces
Menu Bar
Status Bar
Clean Screen
Toolbar size
Customize...

Figure 2.265

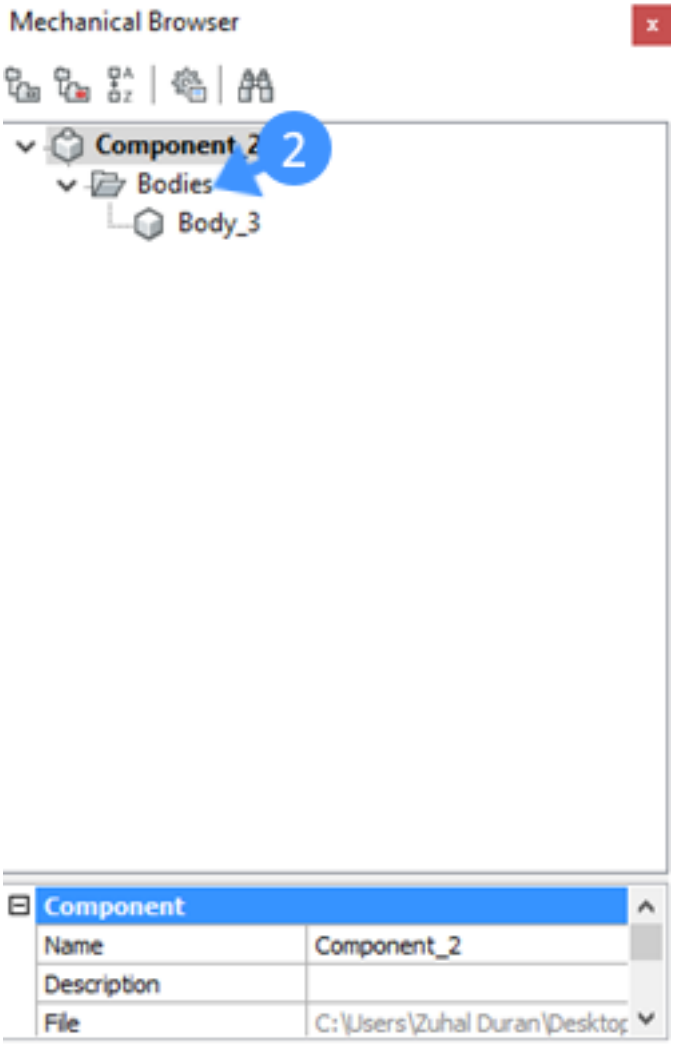

Mechanical Browser
Component 2
Bodies
Body_3
2
Component
Name
Component_2
Description
File
C:\Users\Zuhal Duran\Desktop

Figure 2.266

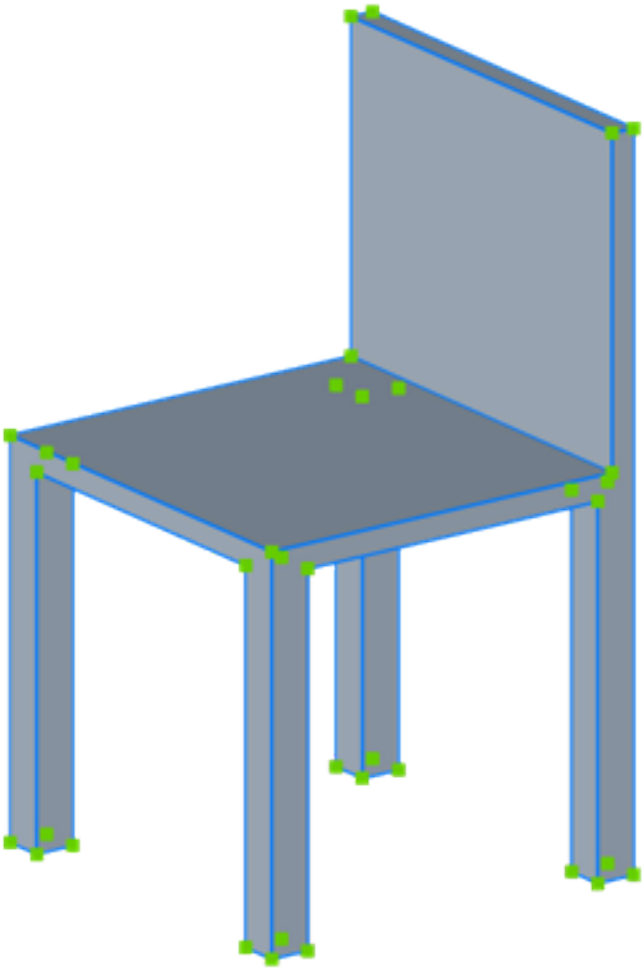


Figure 2.267

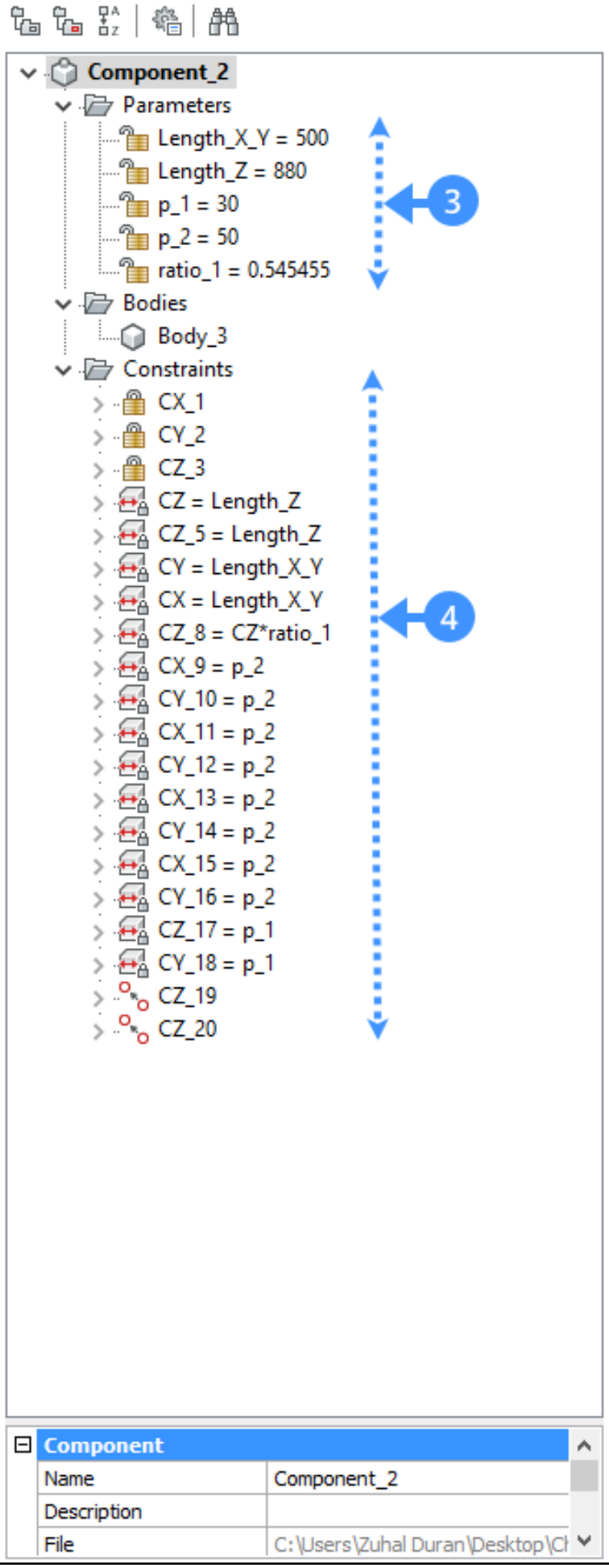
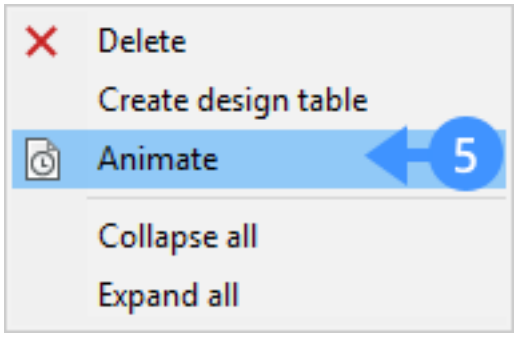

Component_2
Parameters
Length_X_Y = 500
Length_Z = 880
p_1 = 30
p_2 = 50
ratio_1 = 0.545455
3
Bodies
Body_3
Constraints
CX_1
CY_2
CZ_3
CZ = Length_Z
CZ_5 = Length_Z
CY = Length_X_Y
CX = Length_X_Y
CZ_8 = CZ*ratio_1
CX_9 = p_2
CY_10 = p_2
CX_11 = p_2
CY_12 = p_2
CX_13 = p_2
CY_14 = p_2
CX_15 = p_2
CY_16 = p_2
CZ_17 = p_1
CY_18 = p_1
CZ_19
CZ_20
4
Component
Name
Component_2
Description
File
C:\Users\Zuhal Duran\Desktop\Cl

Figure 2.268



Procedure: Changing the parameters and constraints in the parameters manager

Note: After the auto parametrizing, it's possible to change what you do not find clear about the parameters and constraints using the parameters manager.

1. Select one of the faces (6) of your component. Theconstraint type (e.g. distance, fix) regarding the face is displayed. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ChairDistanc

2. Click the distance constraint (7) icon on the element. The editable dimension(s) is shown inside the box.

3. Click the editable dimension box (8). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Mode
A parameter manager displays (9). The parameter manager shows the expressions of the distance constraint (10) and the parameter (11) depending on the selected face. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help

4. Change the parameter name by clicking its field (12) in the parameters manager. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ParameterManagerChairHeight.png})

Figure 2.269

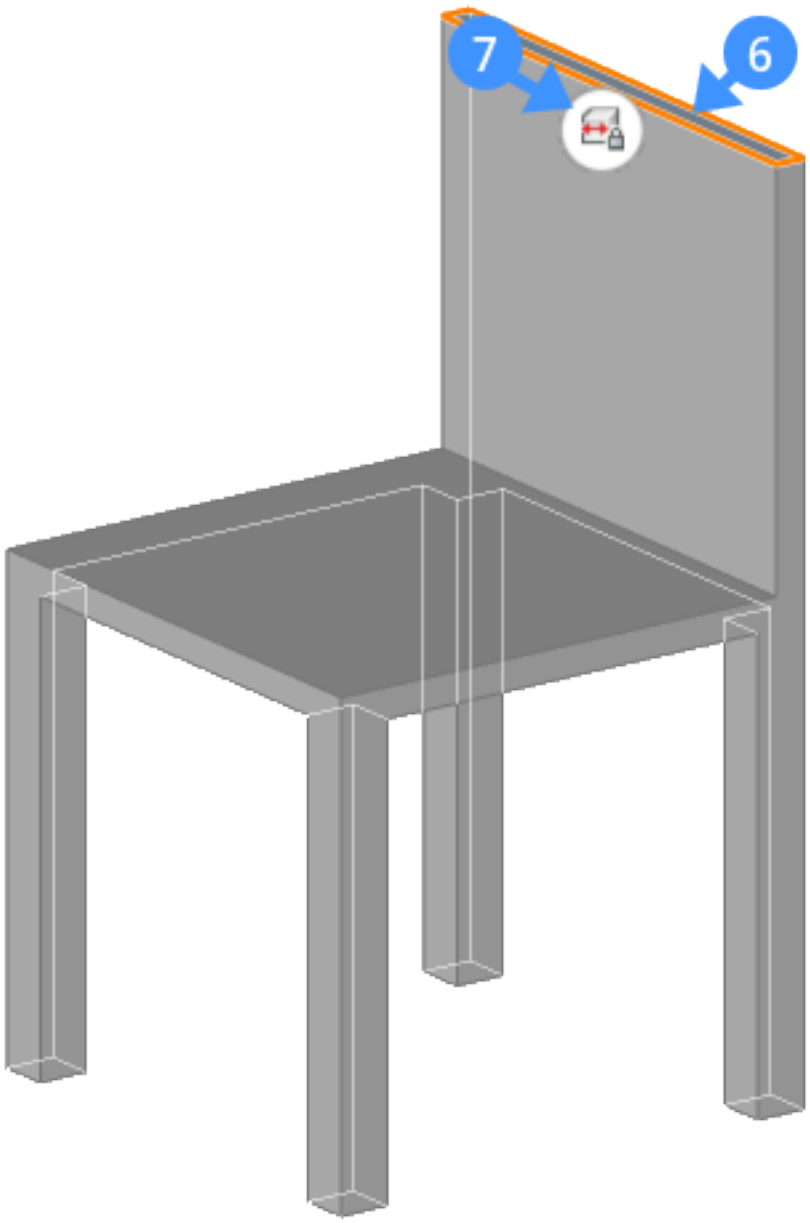


Figure 2.270

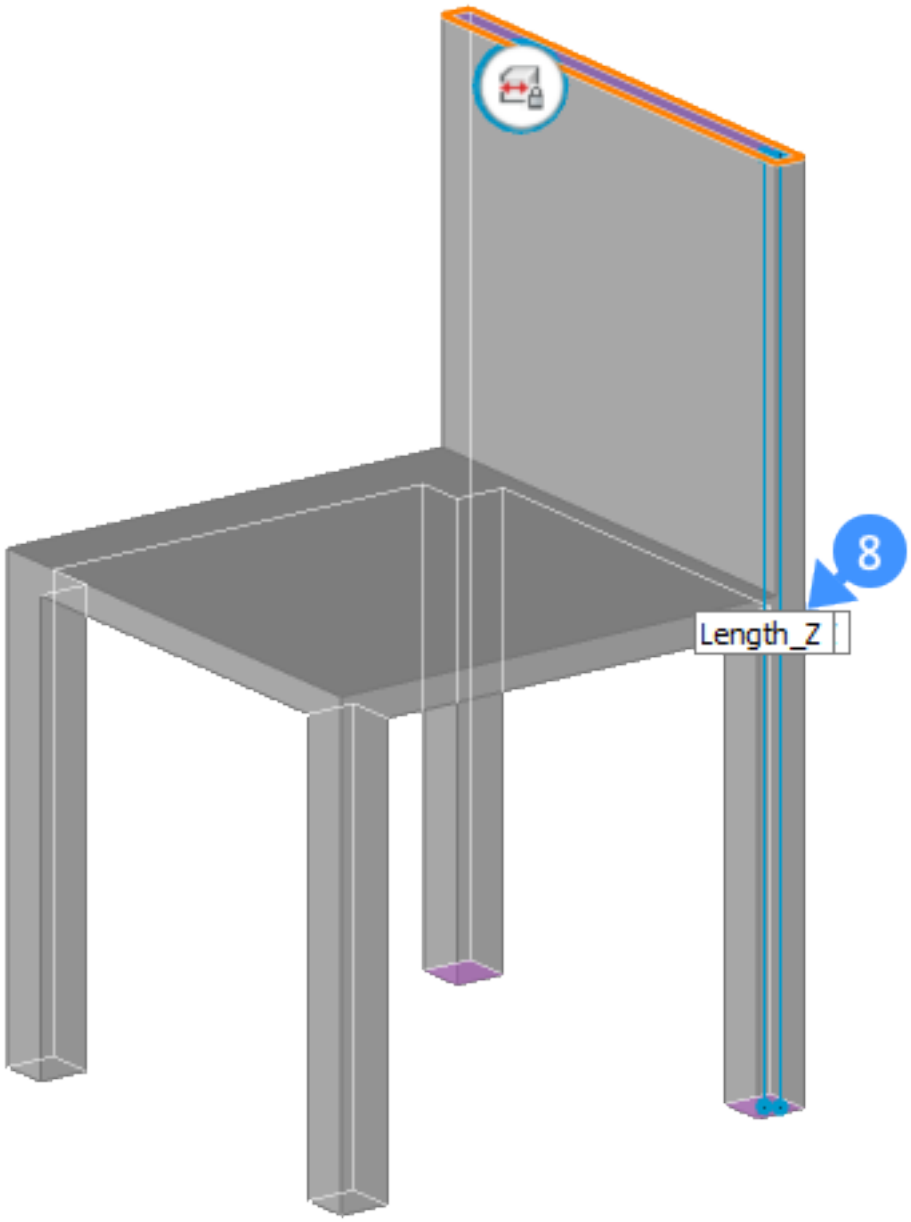
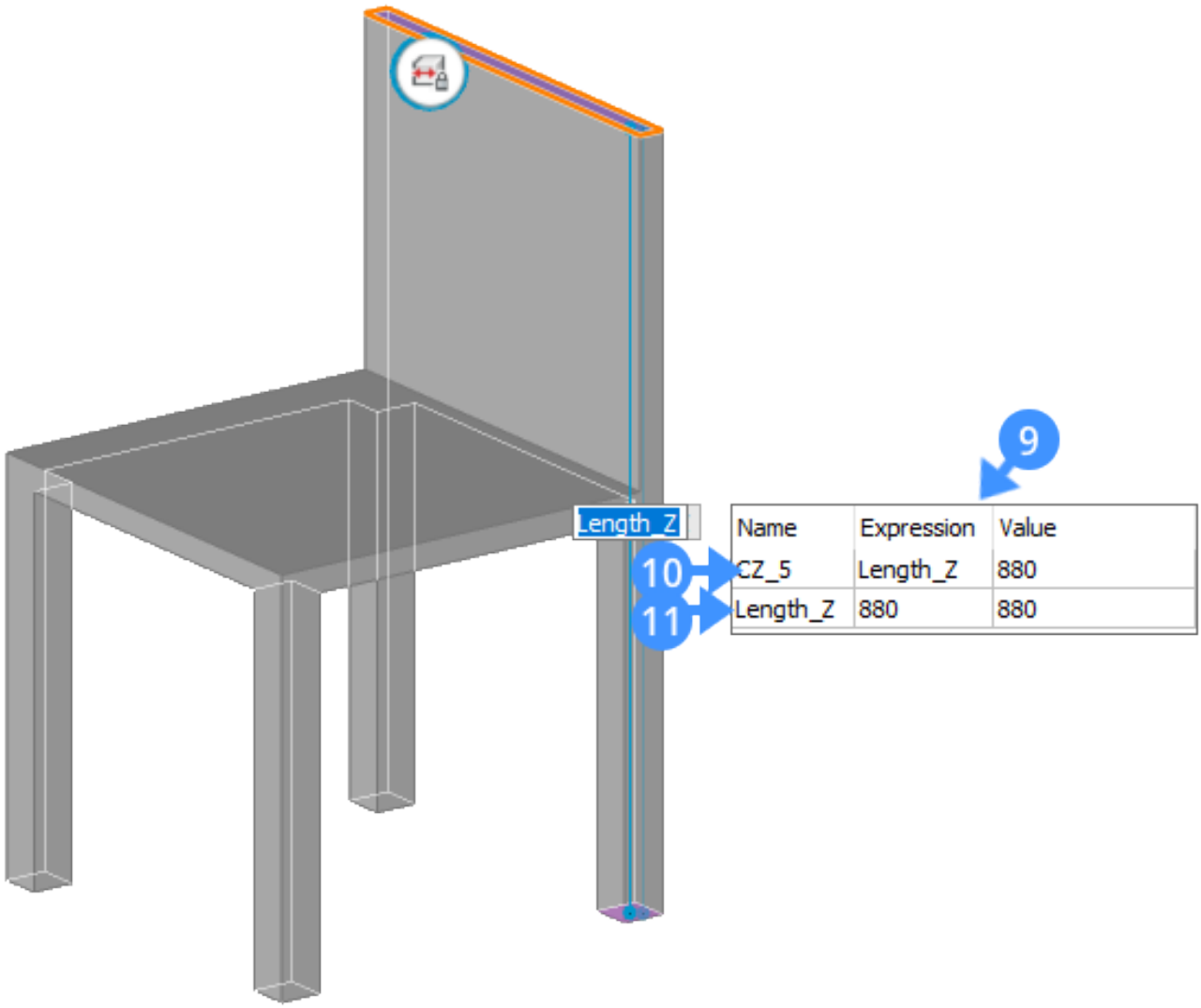


Figure 2.271


Length_Z
9
10
11
Name
Expression
Value
CZ_5
Length_Z
880
Length_Z
880
880

Optional a new parameter can be defined using the New option in the parameter manager's context menu. To do so, right-click the parameters manager and select New (13). A new parameter added in the list (14). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ParameterManagerOption
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ParameterManagerN
Specify a name (15) for the newly created parameter and enter an expression using the expression field (16).
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ParameterManagerN
To link the newly created parameter with an existing constraint, change the expression field of the constraint with the new parameter name (16). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modeli
Note: The above procedure can also be done using the mechanical browser (3.2.2).

Figure 2.272

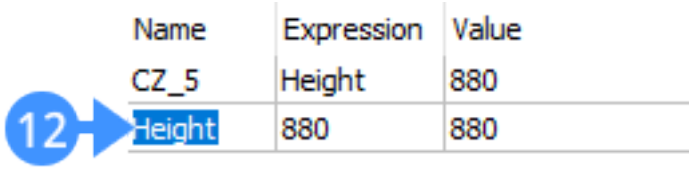


Figure 2.273

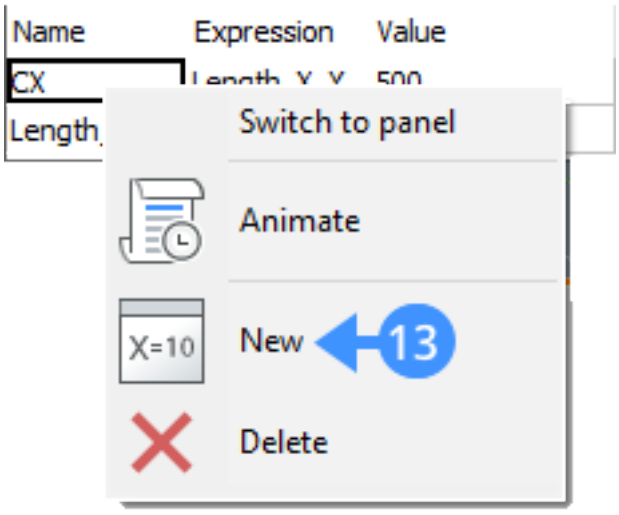


Figure 2.274

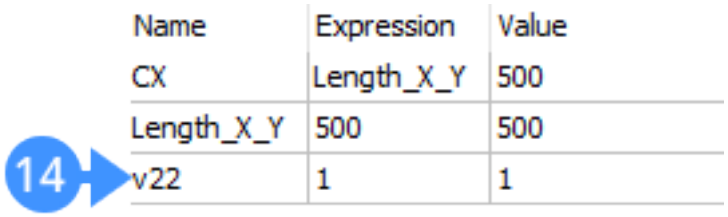


Figure 2.275

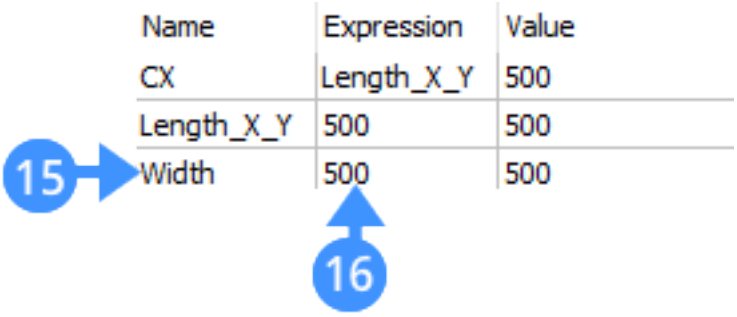
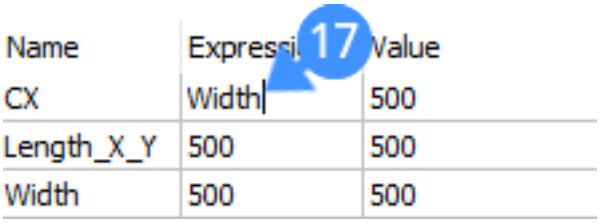


Figure 2.276



2.4.19 Working with Parameters and Constraints

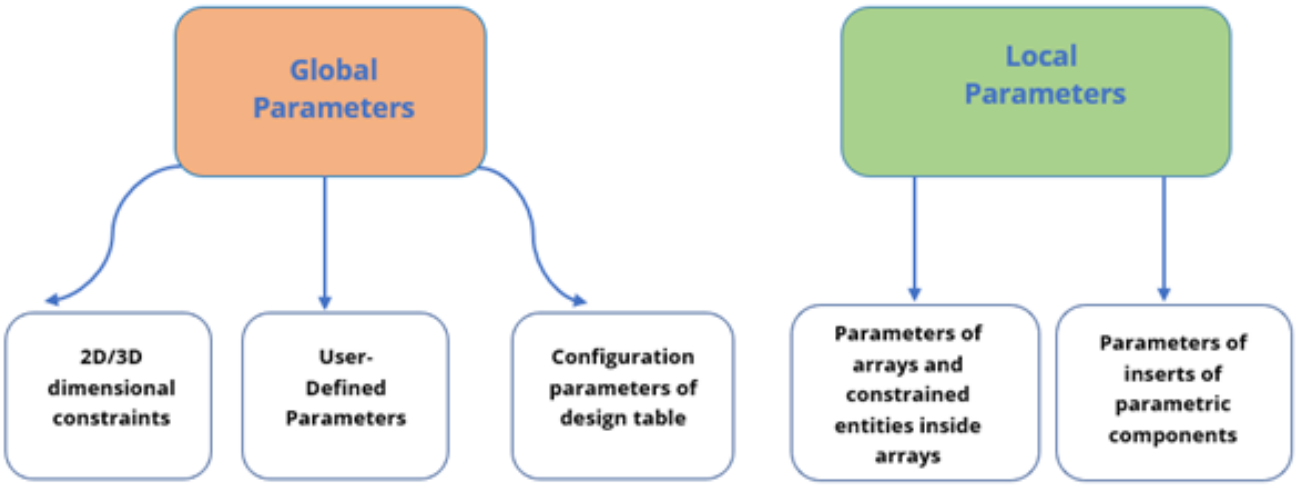
Overview: In this lesson, you learn how to parametrize your BricsCAD BIM entities. **Lesson Objectives** After completing this lesson, you will be able to:

- Understand the difference between global and local parameters.
- Identify the geometric and dimensional constraints.
- Set an expression to a parameter.
- Parametrize your custom component.
- Control your 3D geometry with parameters.

Parametric Modeling

In BricsCAD, parametric modeling is done through the use of geometrical and dimensional constraints. A rich toolset controls the 2D and 3D geometric model with a set of parameters. Each parameter will be displayed with a value in the Mechanical Browser. The value from the different parameters can be linked together via expressions. When one of the parameter values is changed, the geometric model is updated automatically. The toolset allows BricsCAD users to add parametric behavior to any geometry and easily explore design intent. **Note:** In BricsCAD, there are no parent-child dependencies between geometric elements. For instance, if you change a sketch used to create an extruded 3D solid, the solid is not changed accordingly. Any kind of dependencies can be created using the parameters and expressions. There are two types of parameters: Local parameters are attached to a particular entity. Global parameters are not attached to a particular entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ParametersSchema.png})

Figure 2.277



Working with constraints

If you create some components (e.g. windows, doors,...) in your BIM model, you can parametrize them by applying 3D constraints. Defining constraints allows you to control the shape and size of the elements. Together with constraints, parameters determine the positions of entities through an expression. There are two types of 3D constraints in BricsCAD: one specifies the size of the entities, the other locate their positions.

Geometrical constraints



Geometric constraints allow you to control the position of 3D entities with respect to each other.
Toolbar: **Parametric > 3D constraints** Quad: **Constraints** The following table shows the 3D geometrical constraints.

Fix		Keeps solids, edges or faces of solids in-place in the drawing.
Coincident		Applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
Concentric		Keeps two cylindrical, spherical or conical surfaces centered.
Parallel		Keeps the two faces of a solid or of different solids parallel.
Perpendicular		Keeps the two faces of a solid or of different solids perpendicular.
Tangent		Keeps a face and a curved surface of different solids tangent.
Rigid set		Makes a set of entities or sub-entities a rigid body.

Dimensional constraints

Dimensional constraints allow you to control the sizes of 3D entities in drawings, and the distances between them.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/DimensionalConstraintsAc
Toolbar: **Parametric > 3D constraints** Quad: **Constraints** Ribbon: **Parametric Tab > 3D constraints** The following table shows the dimensional constraints.

Distance		Creates a distance between two sub-entities.
Radius		Creates a radius to cylindrical surfaces or circular edges.
Angle		Creates an angle between two faces of a solid or of different solids.

Figure 2.278



Setting an expression to a parameter

In BricsCAD, you can set an expression to any parameter, be it global or local. As an example, you can create a simple expression that only contains a number or a name of a global parameter and apply it to the related geometry in the drawing. You can also use more complex formulas that include the use of standard operators and functions.

NOTES

- Local parameters cannot be referenced by names in expressions.
- The constants **Pi=3.14...** and **e=2.72...** can be used in expressions. The constants names are not allowed to be used as parameter name and name of 3D constraint.

The following table shows the operators that can be used in expressions.

Addition(+)	Produces the sum of numeric values.
Subtraction or Negative (-)	Subtracts the two numeric values.
Multiplication (*)	Multiplies the numeric values.
Division (/)	Divides two numeric values.
Exponentiation (^)	Calculates the exponential value of the given numbers.
Modulo or Remainder operator (%)	Gives the remainder after the division of one numeric value to another e.g. The expression ”5 % 2” would evaluate to 1, because 5 divided by 2 leaves a quotient of 2 and a remainder of 1.

The following table shows the functions and the syntax which can be used in expressions.

Cosine	cos(expression)
Sine	sin(expression)
Tangent	tan(expression)
Arc cosine	acos(expression)
Arc sine	asin(expression)
Cosine	cos(expression)
Arc tangent	atan(expression)
Hyperbolic cosine	cosh(expression)
Hyperbolic sine	sinh(expression)
Hyperbolic tangent	tanh(expression)
Arc hyperbolic cosine	acosh(expression)
Arc hyperbolic sine	asinh(expression)
Arc hyperbolic tangent	atanh(expression)
Square root	sqrt(expression)
Signum function (-1,0,1)	sign(expression)
Round to nearest integer	round(expression)
Truncate decimal	trunc(expression)
Round down	floor(expression)
Round up	ceil(expression)
Absolute value	abs(expression)
Largest element in array	max(expression1;expression2)*
Smallest element in array	min(expression1;expression2)*
Degrees to radians	d2r(expression)
Radians to degrees	r2d(expression)
Logarithm, base e	ln(expression)
Logarithm, base	10 log(expression)
Exponent, base e	exp(expression)
Exponent, base 10	exp10(expression)
Power function	pow
Random decimal, 0-1	Random(expression1;expression2)*

*Use the list separator character as defined on your system: , (comma) or ; (semicolon).

Parametric Blocks

Parametric Blocks refer to blocks that have parameters that determine the size and shape of the 3D geometry. Once the parametric block is created, it can be inserted as a block reference in the drawing. The parameters of your block can be changed after you insert it in your model. Thus, no need to modify the parametric block file individually. The parametric blocks also enable you to use the same block in different sizes and in different shapes in your model.

Procedure: Parametrizing a custom component

The window that was created in the previous lesson is used to demonstrate how the parametric design works. This window consists of three solids: subtractor, fixed frame, and glass pane.

Step 1: Before starting to add constraints:

1. Open the Mechanical Browser with the **MechanicalBrowserOpen** command, to control and manage the values of 3D constraints and parameters.

Note: The Mechanical Browser allows to navigate through all the constraints in the drawing and to edit dimensional constraints.

2. Make sure that the **Enable selection of 3D solid faces** ( ) is toggled on in the **Selection Modes** To easily follow the steps, make sure **Boundary Detection** is toggled off.

Step 2: Applying fix constraints

1. Turn on the BIM_Subtract layer.
2. Select the entities (1) you want to add a fixation to.

(Optional) Press the TAB key to select obscured geometry.

3. Choose Add Fixation (2) in the 3D constraints command group in the Quad.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ApplyFixConstraints.png})
The fixation appears in Mechanical Browser as ‘ Fix_1’. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19

Figure 2.279

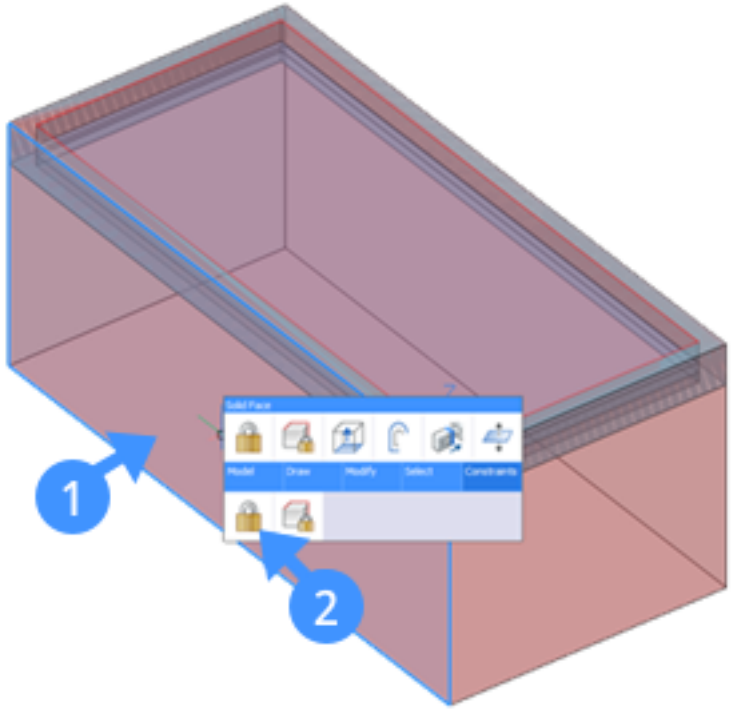
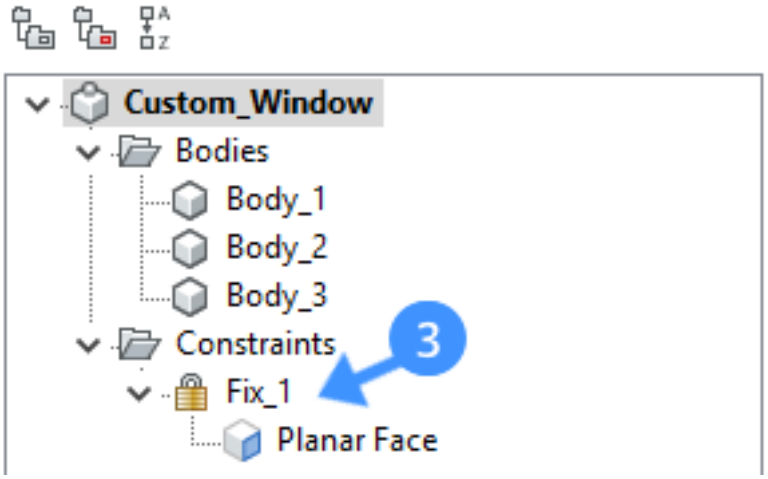


Figure 2.280



Step 3: Applying rigid constraints

1. Select the entities to apply the Rigid Set constraints.

The entities remain in the same position with respect to each other when they are in a Rigid Set. To select more than one sub-entity of the same type at a time, take the following steps:

- Use Look From Widget (4), by default it is at the top right of your screen, to change the view from 3D to TOP.
- Select the entities using the selection box (5). While drawing a selection box, press CTRL key to change the type of sub-entity selection. Every time you press CTRL key the type of the sub-entity selection will change. The type of the sub-entity can be seen under your mouse cursor, for instance, faces (6) or edges (7).

Note: By default, the selection boxes see and select solids. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ApplyRigidConstraints2.pn
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ApplyRigidConstraints3.pn

2. Choose Add Rigid Set ( ) in the 3D constraints command group in the Quad.
3. Repeat the same process for the other sides: top, bottom, and right side.

Figure 2.281

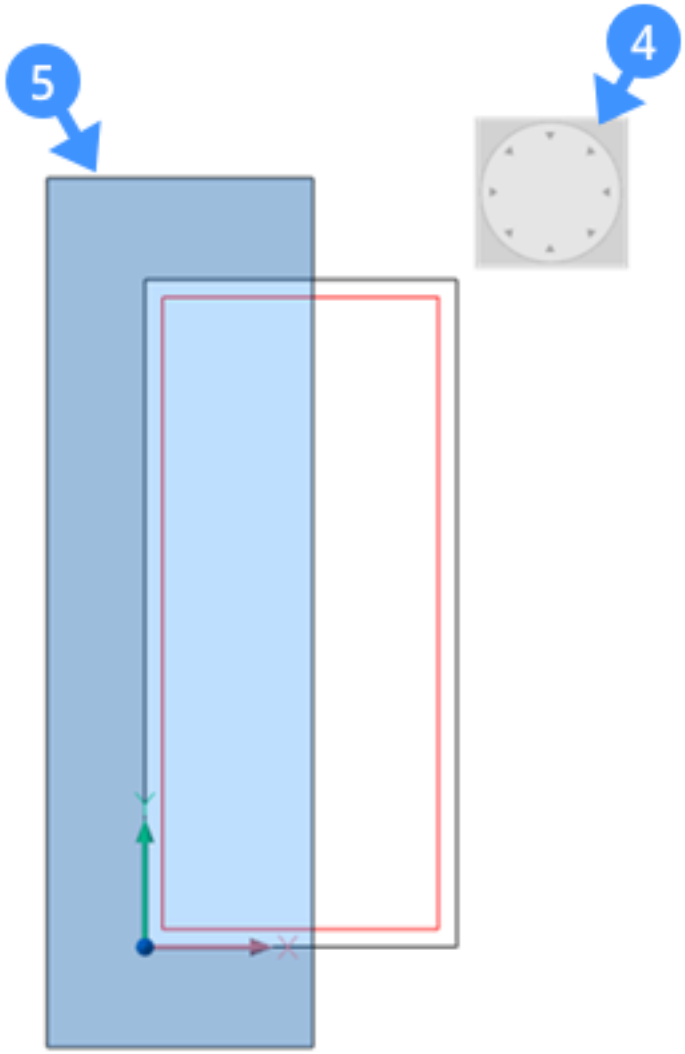


Figure 2.282

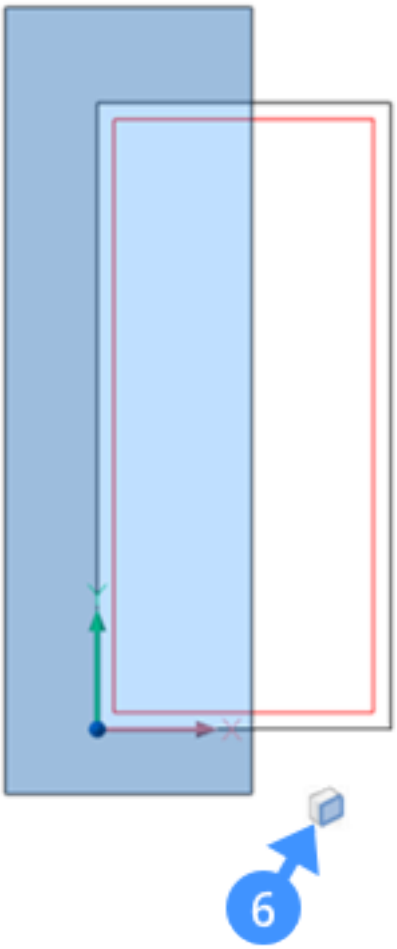
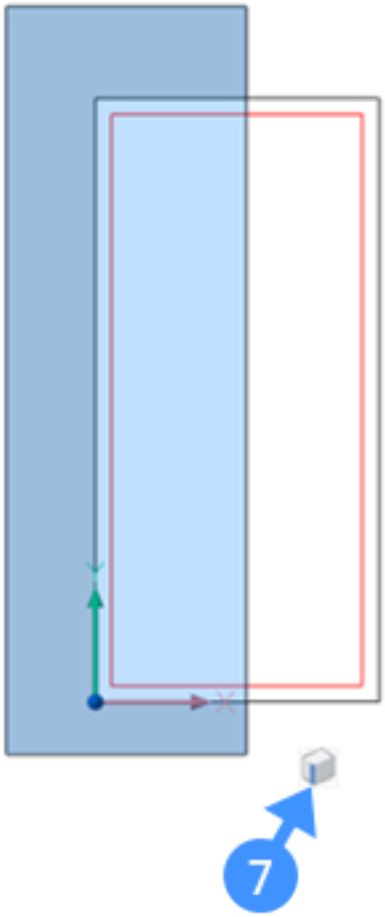


Figure 2.283



Step 4: Applying distance constraints Dimensions of the window and the thickness of the glass pane.

1. Turn off the BIM_Subtract layer.
2. Select the first face.

The face highlights.

3. Hover over the second face.

Hit the TAB key to highlight an obscured face.

4. When the second face highlights, choose**Add Distance**(8) in the3D Constraintscommand group in the Quad.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ApplyDistanceConstraints1
You are prompted: Enter distance value:

5. Do one of the following:

- Press Enter key or right-click to accept the current value.

The current value displays in the dynamic dimension field (9) when the Dynamic Dimensions (DYN) is active. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ApplyDistanceConstraints2.png}

- Type a value in the command line.

6. The distance value between the two surfaces appears in the Mechanical Browser.

Note: To remove any constraint in your model, select the constraint in the Mechanical Browser, e.g. fix, rigid set..., and press the Delete key or right-click and select the Delete option.

Figure 2.284

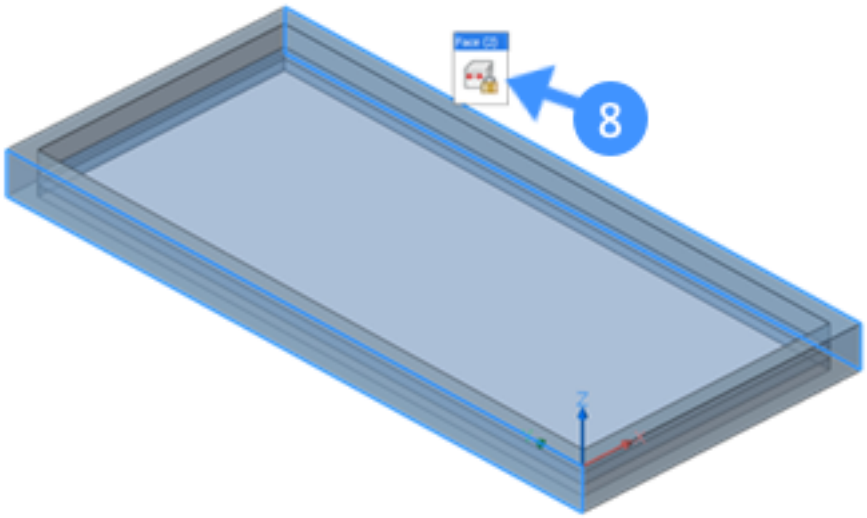
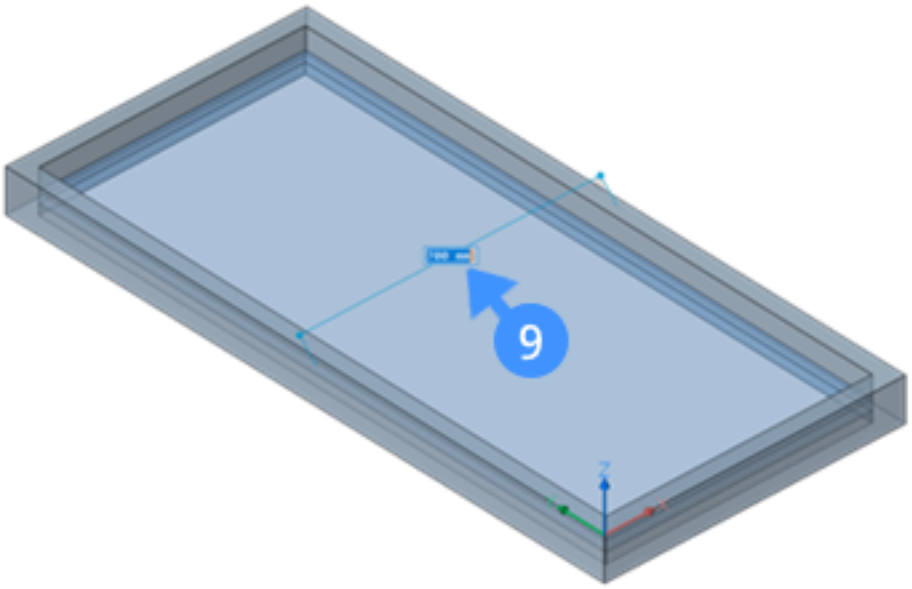


Figure 2.285



Step 5: Applying parameters We will create a parameter that controls the width of the window.

1. Right-click the file name (10) in the Mechanical Browser.
2. Select **Add New Parameter** (11)in the context menu.

The parameter settings grid is created and the parameter is added in Mechanical Browser as 'v1 = 1' (12). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ApplyParameter.png})

3. Select the parameter, then edit its properties in the settings grid.

The following illustration shows the properties of a Width parameter of the custom window. The table below demonstrates each property. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/Appl

Name	Type a parameter name.
Expression	Type a value or formula.
Value	Shows the current value of the parameter.
Description	Define an optional description of the parameter.
Exposed	Controls whether the parameter is available in the properties panel when the component is inserted into a model.
Units	Specifies whether the parameter is linear, areal or volumetric.

4. To link the parameter you just created with the dimensional constraint, select the distance constraint in the Mechanical Browser and use the parameters to formulate an expression in the Expression (13) field in the settings grid of the constraint.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/ApplyParameterConstraint

5. Optionally, change the expression value of the parameter and Press Enter. The assigned distance constraint is changed accordingly.

*In your drawing, you can design as many constraints as possible. Note that the software will keep you away from applying more constraints than are necessary.

Figure 2.286

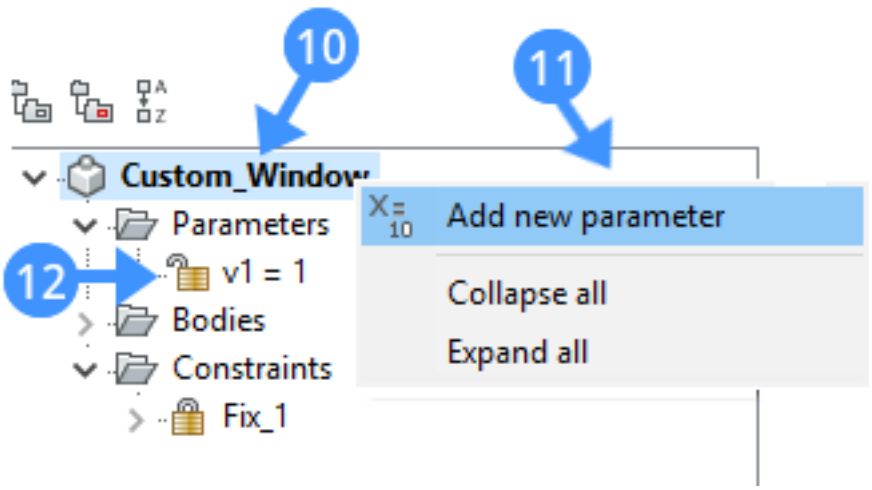


Figure 2.287

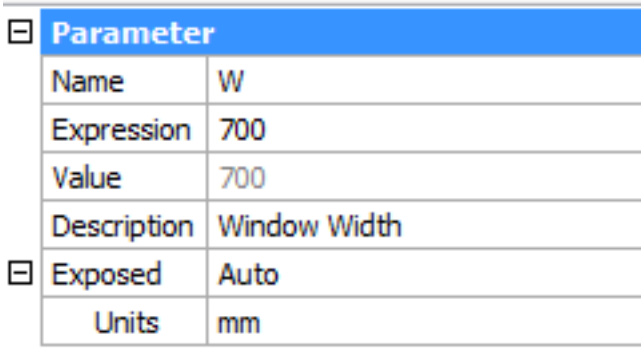
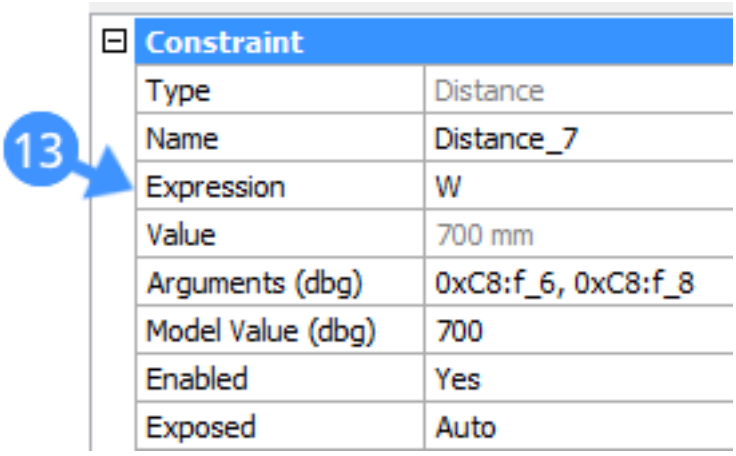


Figure 2.288



2.5 Building Data

2.5.1 How to add and create Compositions

https://www.youtube-nocookie.com/embed/FUsFkvbY0bo This video shows how to create and edit your own multiply compositions, and how to assign these compositions to walls, slabs and other entities. Back to Tutorial Overview

2.5.2 How to add Building & Story information

https://www.youtube-nocookie.com/embed/zHIVWoX1azc This video shows how to use spatial locations: add building and storey information to your project, and assign this information to different objects, manually or automatically. Back to Tutorial Overview

2.5.3 How to classify BIM elements

https://www.youtube-nocookie.com/embed/RMxR0v3q6Jg

This video shows how to classify an object manually or automatically according to the IFC classifications. Back to Tutorial Overview

2.5.4 How to create and edit Custom Properties

https://www.youtube-nocookie.com/embed/18QA3Mq7h1Q This video shows how to define custom properties, how to assign these properties to certain object types and how to modify the property values of a certain entity.
Back to Tutorial Overview

2.5.5 How to define Rooms

https://www.youtube-nocookie.com/embed/1LRN6m_NQMU This video shows how to define room objects, and how to change a room's representation. Back to Tutorial Overview

2.5.6 How to extend a Ply

https://www.youtube-nocookie.com/embed/DyM9A9j7ZS8 This video shows how Push/Pull can be used for the local editing of a single composition ply. Use this technique in combination with Propagate for an incredibly accurate BIM model. Back to Tutorial Overview

2.5.7 How to use Structure Browser

https://www.youtube-nocookie.com/embed/EuCiCDYXKuI This video shows how the Structure Browser can be customized for your needs. Add filters and sort on property values, and save your customizations in dedicated .cst files. Back to Tutorial Overview

2.5.8 How to use the Project Browser

https://www.youtube-nocookie.com/embed/FFzPWPt6hQs This tutorial shows you how to set up the Project Browser, and how it can be used for navigating through your project: models, sheets, sections and schedules can all be found here. Back to Tutorial Overview

2.5.9 How to use the Profile Library for linear solids

https://www.youtube-nocookie.com/embed/WiMp4lla-24 This video shows how to use the Profile Library to save your own sections, and how to assign these sections to structural steel profiles or MEP flow segments. Back to Tutorial Overview

2.5.10 How to use the Properties panel

https://www.youtube-nocookie.com/embed/wMNJ2yA2DJE

This tutorial shows how the Properties Panel is structured and which properties it shows. Back to Tutorial Overview

2.5.11 How to use Classification Codes

https://www.youtube-nocookie.com/embed/DYYpbpDECmo This video shows how the most commonly used Classification Systems can be used inside BricsCAD BIM. Back to Tutorial Overview

2.6 Building Data

2.6.1 BIM Classify

Command: BimClassify (6.7.5) **Overview:** In this lesson, you learn how to classify an entity according to the corresponding building element type. **Lesson Objectives** After completing this lesson, you will be able to:

- Classify the entities by the appropriate building elements.
- Use the classification categories.
- Convert an entity to a block and classify the block reference as a BIM element.
- Automatically classify a set of entities.

Tutorial: How to classify BIM elements Click here (2.5.3) to watch

About

If you have a conceptual model without any classification you can turn these elements into classified BIM elements using BIM Classify.  Menubar: **BIM > Classify** Toolbar: **BIM > Classify** Quad: **BIM >** Keyboard: **BIMCLASSIFY**

Classifying entities

1. Select an entity in the drawing to classify.
2. Choose a classification type (1) in the BIM command group in the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
   The BIM data is assigned to the selected entity.
3. **(Optional)** Choose the classify manually option from the Quad to access the BIM Element classification types.
   This displays the Classify as BIM Element dialog box (2): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
   Note: The classification types are grouped into the following categories:

   - Building Core Elements
   - Building Architecture Elements
   - Building Service Elements
   - Building Structure Elements

   Do one of the following:

   - Enter a specific name of the BIM Element type in the **Search** field (3) to filter the available element types
   - Select a Building Element type from the list
   - (Optional) Check the **Convert to block option** (4) and enter a name in the **Block name field** (5)

   This option will convert the selected entity to a block and classify the block reference to the defined Building Element type.

   - Click the OK button (6) to accept it.
4. **(Optional)** Use the Auto () option from the Quad to classify the selected entities automatically. This automatic classification is done based on the geometrical features of the object. Always check whether it was correctly classified.
5. To remove the assigned BIM data from the entity, select the entity in the drawing and choose the Unclassify () option from the Quad. The BIM data is removed from the selected entity.

Figure 2.289

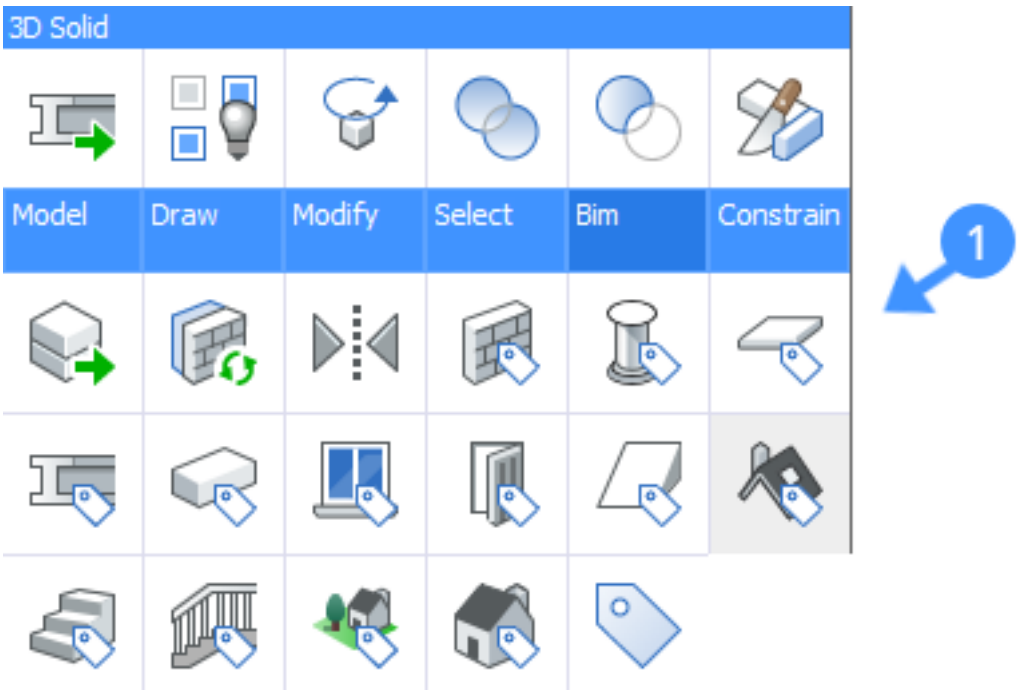
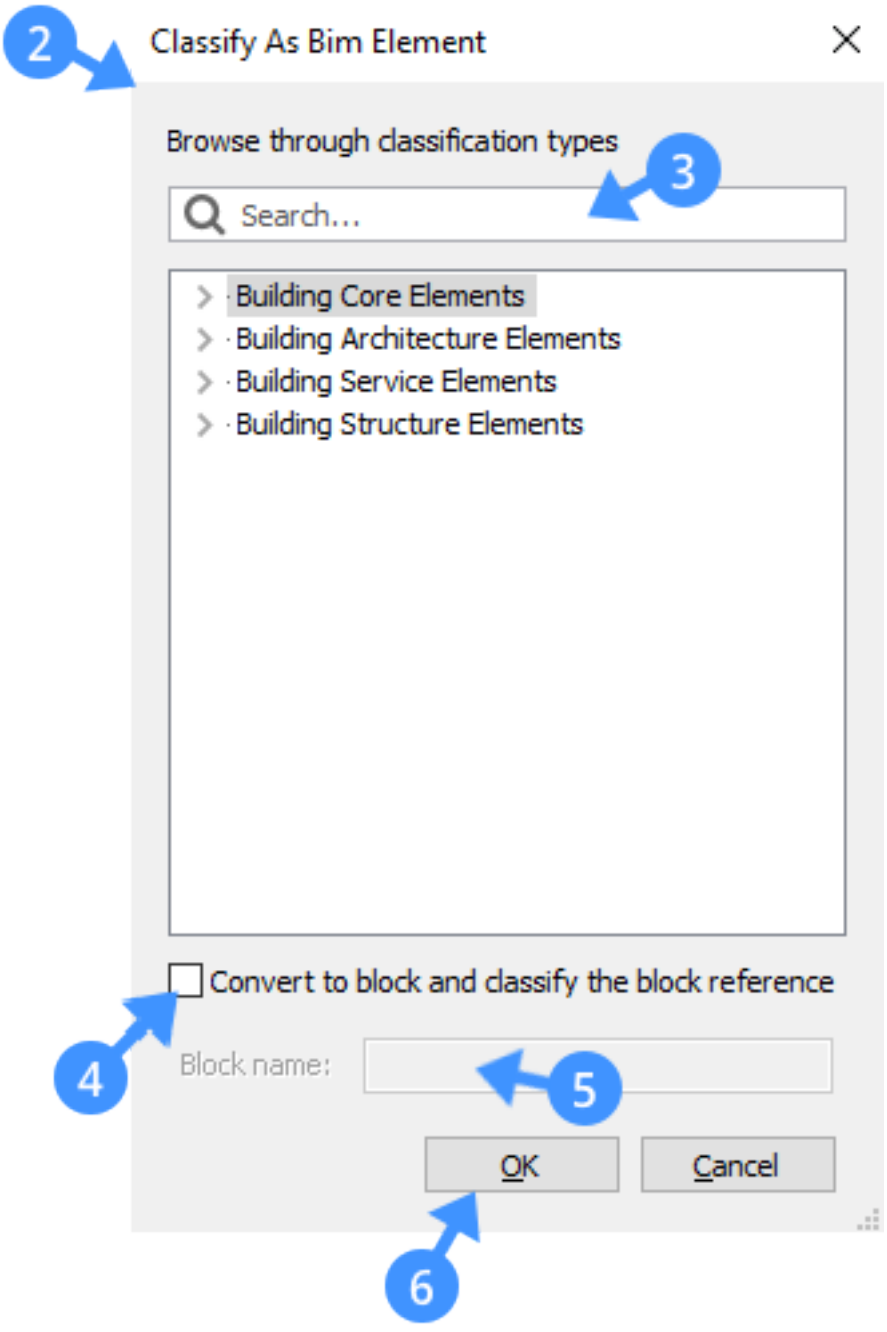


Figure 2.290


2
Classify As Bim Element
Browse through classification types
3
Search...
Building Core Elements
Building Architecture Elements
Building Service Elements
Building Structure Elements
Convert to block and classify the block reference
4
Block name:
5
OK
Cancel
6

Properties of the classified entities

When you classify an entity, a number of properties with regard to the building element type is displayed in the Properties panel. The following illustrations show the properties that are available for windows and doors: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMPropertiesPanelWindow1.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMPropertiesPanelWindow2.png) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMPropertiesPanelWindow4.png) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMPropertiesPanelWindow3.png)

Figure 2.291

BIM	
Type	Window
Name	
Description	
Building	
Story	
Composition	
Display composition	Off
GUID	3XJd32NVX2vPl36xCkPE1
Room bounding	Off
Construction type	Not defined
Operation type	Not defined
Window inset	
Overall height	0 mm
Overall width	0 mm
Sill height	0 mm
Head height	0 mm
Assembly code	
Assembly subtype	

Figure 2.292

Door Window Glazing Type	
Glass layers	0
Glass thickness 1	0 mm
Glass thickness 2	0 mm
Glass thickness 3	0 mm
Fill gas	
Glass color	
Is tempered	Off
Is laminated	Off
Is coated	Off
Is wired	Off
Translucency	0
Reflectivity	0
Beam radiation transmi	0
Solar heat gain transmi	0
Thermal transmittance	0 W/m²·K
Thermal transmittance	0 W/m²·K

Figure 2.293

Door Window Shading Type	
External shading coeffi	0
Internal shading coeffic	0
Inset shading coefficier	0

Figure 2.294

Window Common	
Reference	
Fire rating	
Acoustic rating	
Security rating	
Is external	Off
Infiltration	0 m^3/s
Thermal transmittance	0 $W/m^2 \cdot K$
Glazing area fraction	0
Smoke stop	Off

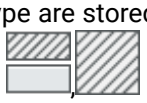
2.6.2 BIM Compositions

Commands: BimAttachComposition (6.7.3), BLComposition (6.7.38),BimClassify (6.7.5) **Overview:** In this lesson, you will learn how to apply compositions to 3D solids. **Lesson Objectives** After completing this lesson, you will be able to:

- Use the BIM Compositions panel to attach a composition to a solid
- Use the Compositions editor to create, delete, or edit compositions in a project
- Utilize the physical materials library to create single-ply compositions
- Display composition plies
- Remove a composition from a solid

Tutorial: How to add and create Compositions Click here (2.5.1) to watch

About

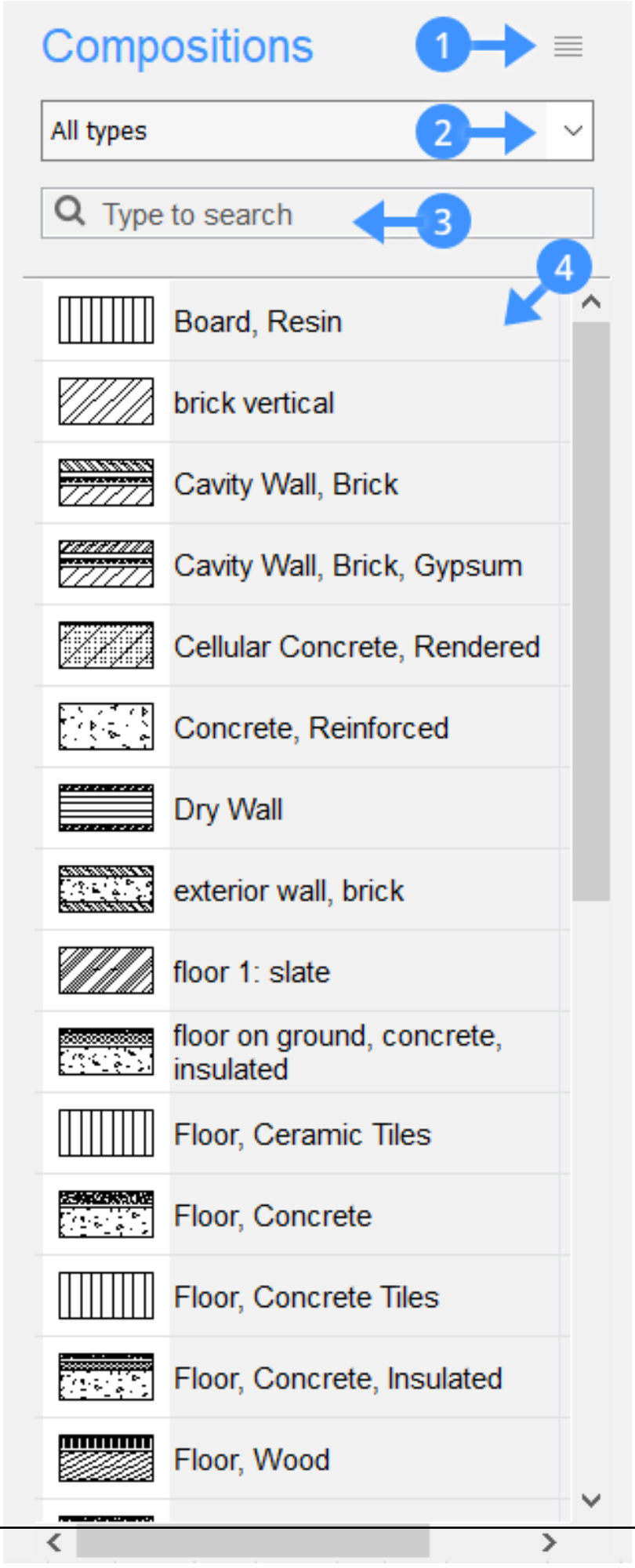
Compositions can be assigned to the various solids of a BIM model. They can be either multiple or single ply. In BricsCAD, predefined compositions of each building element type are stored in the library database. They can be found in the **BIM Compositions**panel and **Compositions Editor**.  Keyboard:

- **BLCOMPOSITIONS** () to open the compositions dialog box.
- **BLMATERIALS** () to open the materials dialog box.

Or use the BIM Compositions (1.12.7) dockable panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19

(1) Menu	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
(2) Composition types list	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
(3) Search field	Filters the compositions list as you type. Only compositions that contain the current search term are displayed.
(4) Available compositions	All compositions of the selected type in both the Library and the Project database display, unless the 'Show only compositions in project ...' menu option is selected.

Figure 2.295

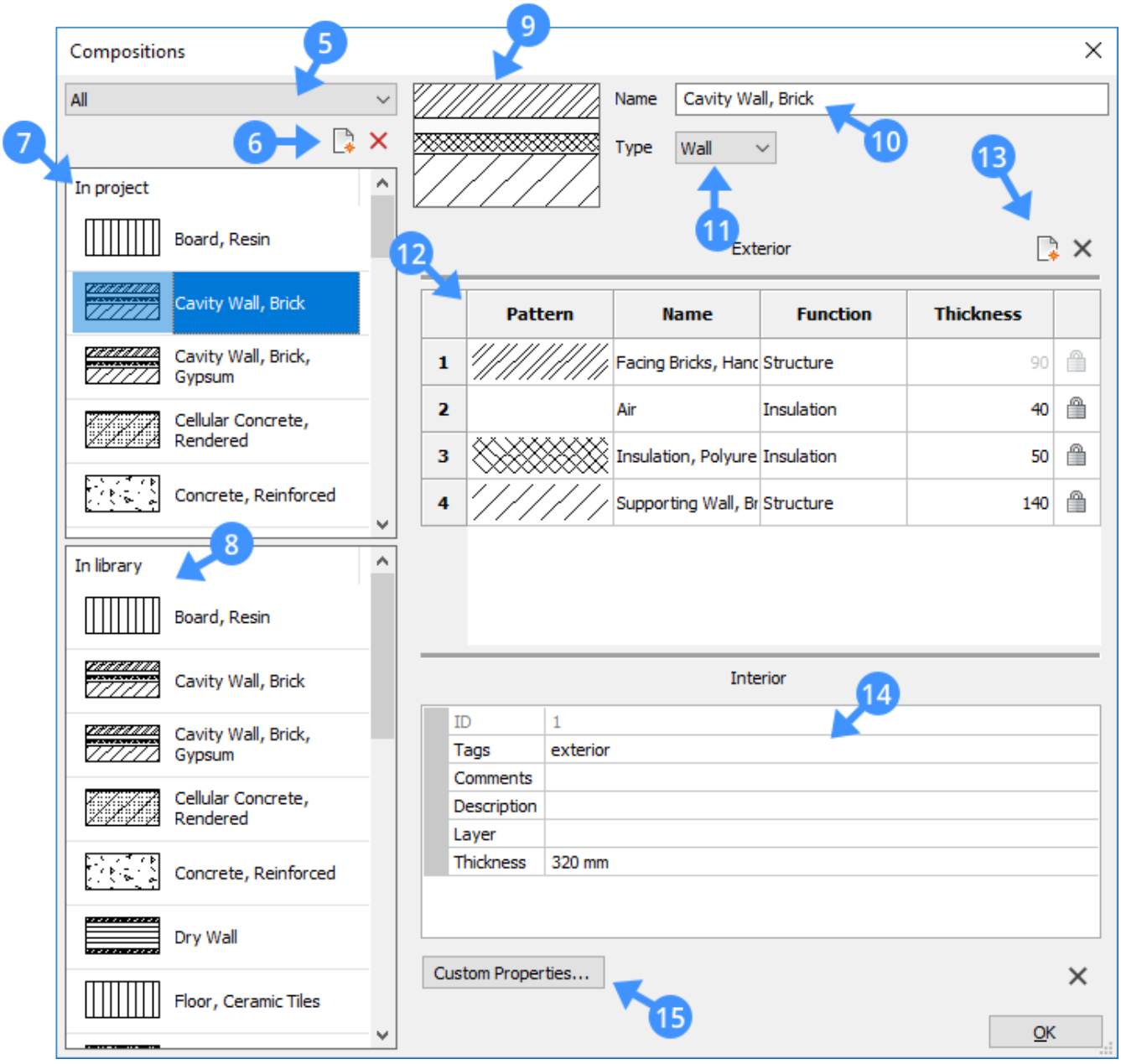

Compositions
1
All types
2
Type to search
3
4
Board, Resin
brick vertical
Cavity Wall, Brick
Cavity Wall, Brick, Gypsum
Cellular Concrete, Rendered
Concrete, Reinforced
Dry Wall
exterior wall, brick
floor 1: slate
floor on ground, concrete, insulated
Floor, Ceramic Tiles
Floor, Concrete
Floor, Concrete Tiles
Floor, Concrete, Insulated
Floor, Wood

BIM Composition Editor

The BIM Composition editor is a dialog box that contains options to create, modify, and delete a composition. Using the composition editor you can also sort the compositions by category and view the properties of the composition. The BLCompositions (6.7.38) command opens the BIM Compositions Editor: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/Compositioneditor2.png})

(5) Composition category selection	Filters by category: Wall, Slab, Roof or Generic.
(6) New composition	Creates a new composition.
(7) In project	Displays the compositions of the selected category (wall, slab...) in the Project database.
(8) In library	Displays the compositions of the selected category (wall, slab...) in the Library database.
(9) Preview pane	Displays the currently selected composition.
(10) Name field	Displays the selected composition name. You can change the default name.
(11) Category drop-down box	Displays a category for the new composition.
(12) Structure grid	Displays the material and thickness of the various composition plies. The ply on top will become the outer face of the solid.
(13) Add ply button	Opens the physical materials display box which allows you to choose a material from the library or project database. Here you can also create a new material.
(14) Properties grid	Shows the properties of the selected composition.
(15) Custom properties	Opens the custom properties dialog.

Figure 2.296


Compositions
All
In project
Board, Resin
Cavity Wall, Brick
Cavity Wall, Brick, Gypsum
Cellular Concrete, Rendered
Concrete, Reinforced
In library
Board, Resin
Cavity Wall, Brick
Cavity Wall, Brick, Gypsum
Cellular Concrete, Rendered
Concrete, Reinforced
Dry Wall
Floor, Ceramic Tiles
Name
Cavity Wall, Brick
Type
Wall
Exterior
Pattern
Name
Function
Thickness
1 Facing Bricks, Hanc Structure 90
2 Air Insulation 40
3 Insulation, Polyure Insulation 50
4 Supporting Wall, Br Structure 140
Interior
ID 1
Tags exterior
Comments
Description
Layer
Thickness 320 mm
Custom Properties...
OK
5
6
7
8
9
10
11
12
13
14
15

Procedure: Create a new composition

Use either the **Project** or the **Library** database to create BIM compositions.

Step 1: Add a composition

1. Open the BIM composition editor. Do one of the following:
   - Select the **Open the compositions dialog ...** in the menu on the Compositions panel.
   - Type BLCompositions (6.7.38) in the command line.
2. Click the **New** icon on the left-hand side of the compositions toolbar. A new (empty) composition is added in the selected database.

Step 2: Edit the composition

1. Select the new composition from the database.
2. Type a new name in the **Name** field**.**
3. Use the**Type** drop-down box to specify the composition type.
4. Click the **Add Ply** buttonon the right-hand side of the compositions editor to include a ply. The **Physical Materials** dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/Physica
5. Select a material either in the Project **or** the Library database, then double-click **or** press the **OK** button to include this material in the composition.
6. **Optional** Right-click the material and choose **Make composition** (16) from the context menu to create asingle-ply composition.
7. Click the **Thickness** field (17) to modify the thickness of a ply in the **Structure** grid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/CompositionsDialogBox.png})
8. Enter a value in the field (18) and press Enter to accept it. The overall thickness of the composition is updated accordingly.
9. Repeat the previous steps to add more plies.
10. To change the order of the plies, drag the ply number (21) to the desired position.

Figure 2.297

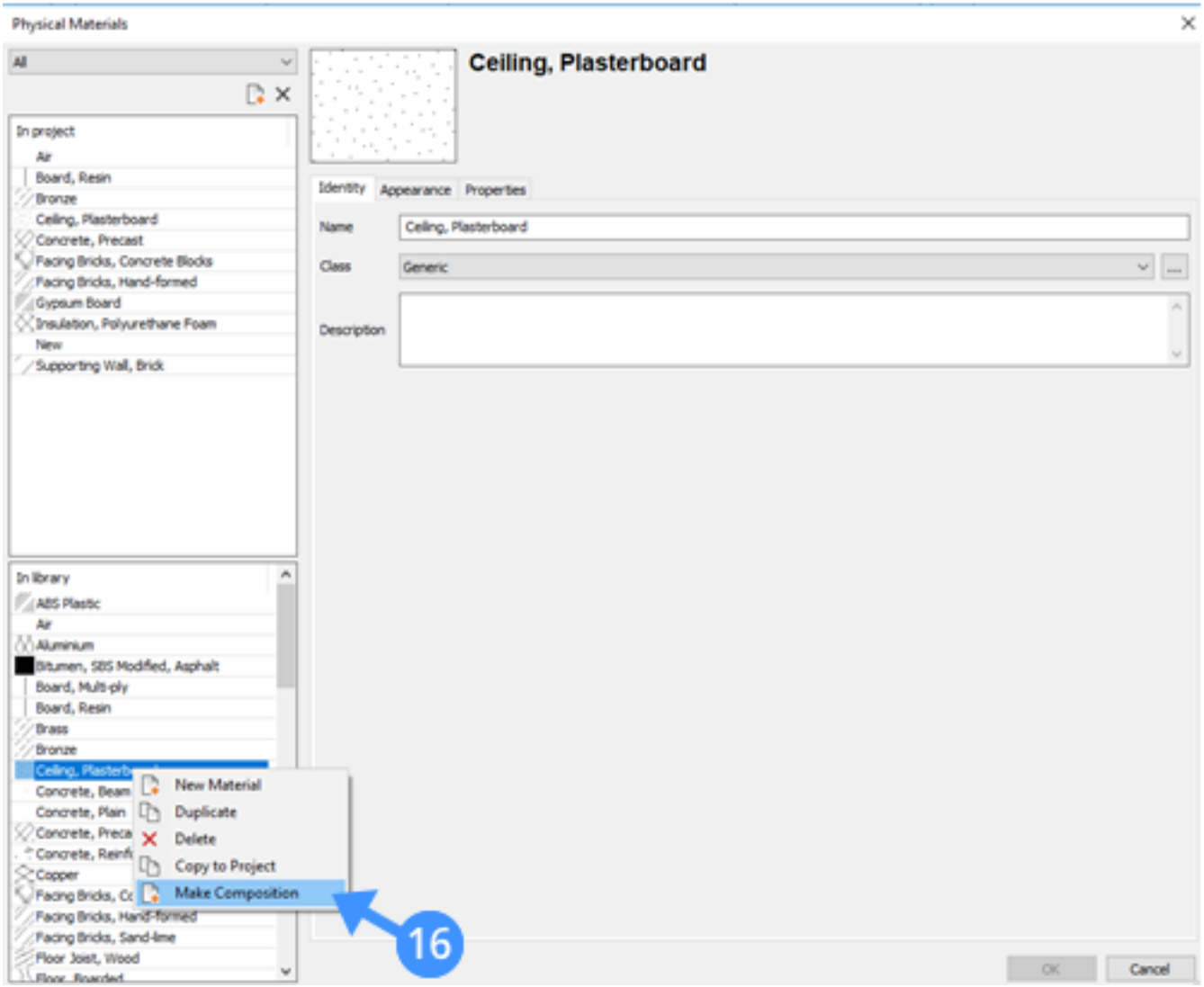
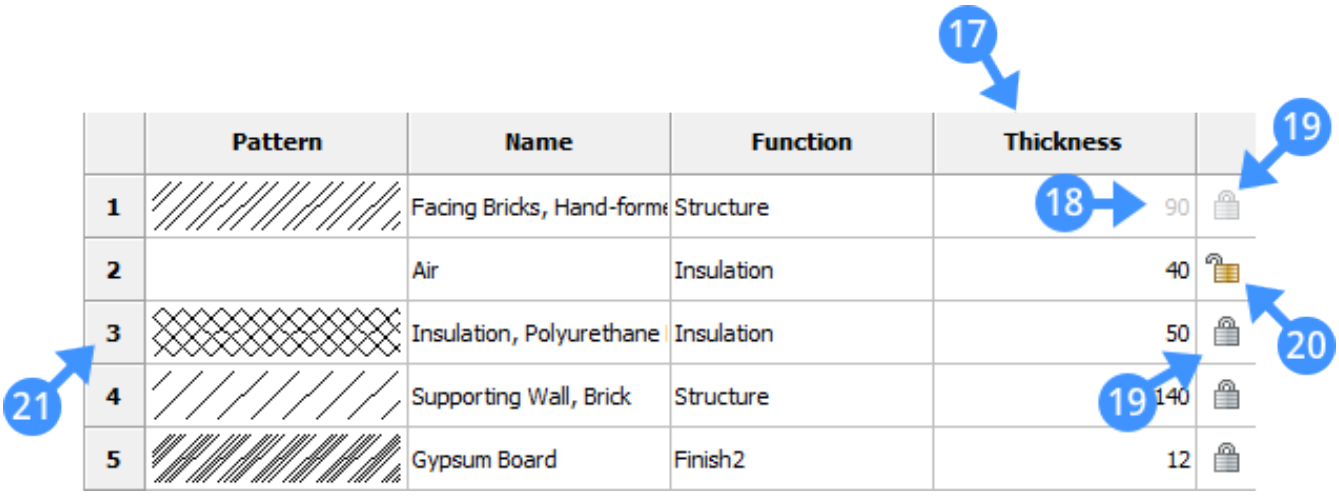


Figure 2.298



Composition thickness

The overall thickness of a composition equals the sum of the thicknesses of the composing plies. The thickness of a ply can be either locked or unlocked.

- Locked thickness (19): The thickness of the ply is fixed.
- Unlocked thickness (20): The thickness of the ply is variable.

Note Only one ply in a composition can have an unlocked thickness. As a result, the total thickness of a composition can be:

- **Fixed:** all its plies have locked thicknesses.
- **Minimal:** the compositioncontains at least two plies and one ply has an unlocked thickness. The minimum thickness equals the sum of the thicknesses of the locked plies.
- **Free:** a single ply composition with an unlocked thickness.

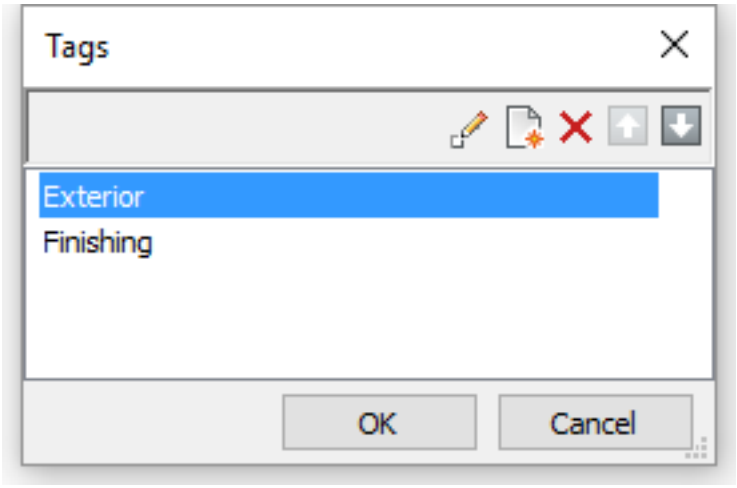
The thickness of a ply can be unlocked only if the **Allow Custom Thickness** property of the material is **Yes**.

Note	When the thickness of a ply in a composition is modified, the solids which include this composition will not be updated automatically. To re-apply the overall thickness of a composition to the solid use the BimUpdateThickness (6.7.35) command.

Editing a composition

1. Select the composition in either the **In Project** or the **In Library** list on the Compositions dialog. The current settings of the selected composition are displayed.
2. Type a name in the **Name** field.
3. Click the **Category** button, then select a composition category: **Wall**, **Slab**, **Roof** or **Generic**. Composition categories are used to filter the compositions list and the Compositions dialog.
4. Edit the composition plies in the **Ply** grid. Here you can:
   - Modify the thickness of a ply
   - Change the material of a ply
   - Add a ply
   - Modify the order of the plies
   - Delete a ply
1. Edit the properties of the composition in the **Properties** grid.
2. Edit the **Tags** field: tags are separated by commas. Tags are case sensitive.
3. Click the **Browse** button ( ) to open the **Tags** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
4. Click the **New Item** button ( ) to add a new tag at the bottom of the list.
5. Select a tag in the list then click a tool button:
   - : edit the selected tag
   - : delete the tag
   - : move the tag up
   - : move the tag down
6. Edit the **Comments** field.
7. Click the **OK** button to save the changes.

Figure 2.299



Ply Editing

Modify the thickness of a ply The thickness of a ply can be modified only if the Allow Custom Thickness (2.6.10) property of the Material is **Yes.**

1. Click the **Thickness** field in the **Ply** grid of the composition.
2. Type a value and press Enter. The overall thickness of the composition is updated accordingly.

Note	When the thickness of one or more plies in a composition have been modified, the solids which have this composition applied are not updated automatically. Use the BimUpdateThickness (6.7.35) command to reapply the overall thickness of a composition to the solid.

Change the material of a ply

1. Double click either the **Pattern** or the **Name** field of the ply. The **Materials** dialog displays.
2. On the **Materials** dialog double click a material in either the **In Project** or the **In Library** list. The selected material is applied to the ply.
3. **Optional** Edit the thickness of the ply.

Add a ply A new ply will be added above the selected ply. You can drag the ply to a different position afterward.

1. Select a ply in the composition.
2. Click the **Add Ply** tool button. The **Materials** dialog is displayed.
3. On the **Materials** dialog double click a material in either the **In Project** or the **In Library** list. The selected material is applied to the new ply.
4. **Optional** Edit the thickness of the ply.
5. **Optional** Drag the ply to a different position.

Modify the order of the plies The plies of a composition are ordered, top to bottom, from outside to inside. Drag the ply number (first column in the ply grid) to the desired position.

Delete a ply

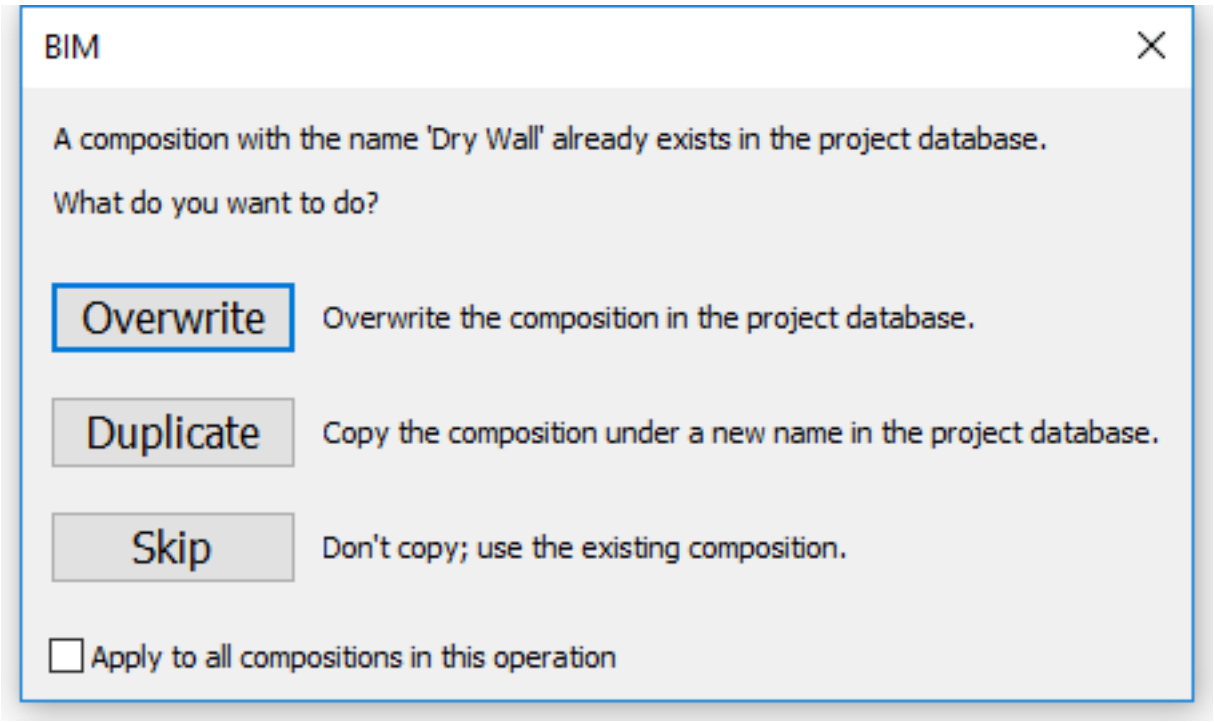
1. Select the ply in the composition.
2. Click the **Delete** tool button.

Copy compositions

The following procedure can be used in both the BIM Compositions panel (1.12.7) and the Compositions editor.

1. Select the compositions(s) in the source database. Press and hold the Ctrl key to select multiple composi-

Figure 2.300



Delete compositions

1. Select the compositions in the **In Project** or **In Library** list on the BIM Compositions panel (1.12.7) or the Compositions editor.
2. Do one of the following:

- Right-click and choose **Delete** in the context menu.
- Click the **Delete** button (**Compositions** editor only).

Note	Compositions that are used in the project cannot be deleted from the project database

Figure 2.301

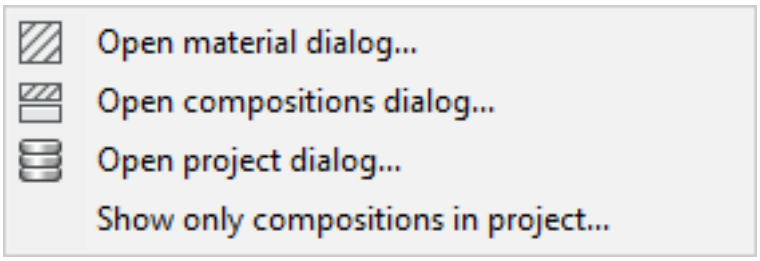


Figure 2.302



2.6.3 BIM Data Structure

Commands: BIMSPATIALLOCATIONS (6.7.31), BIMATTACHCOMPOSITION (6.7.3), BLCOMPOSITIONS (6.7.38), BIMPROFILES (6.7.17), BLMATERIALS (6.7.39), BIMPROPERTIES (6.7.41), BIMPROJECTINFO (6.7.19)

Project database and Library database

A BIM Project consists of dwg entities, classified as Building Elements, to which Compositions can be attached. All Compositions being used in a project, are stored in the Project database. The Project database can be stored inside a dwg file, or in a separate file with extension **.bsyslib:**

- When the Project database is embedded in a dwg file, that dwg file contains the complete BIM Project.
- When the Project database is stored as an external file, the BIM Project can consist of multiple dwg files, all using the same Project database.

To reuse compositions and building materials across multiple BIM Projects, a Library database can be specified, in addition to the Project database. The Library database is a **.bsyslib**file, usually stored in a location which is common to several projects. When Compositions and Building Materials from the Library database are used in a BIM Project, they are automatically imported in the Project database. To maintain and extend BIM databases, users can drag Building Materials and Compositions from one database to the other, provided that they have write access to the destination database.

NOTE	Sample databases are installed in the **Support**subfolder the Roamable root (1.13.2) folder. When a BricsCAD update is installed, the BricsCAD User File Manager (1.13.2)will prompt you whether to overwrite the sample databases or not. Therefore it is recommended to rename your library databases and/or save them at a different location. The location of the library databases is defined in the **Library DB** tab of the BIM Project Info (2.6.5) dialog.

Materials and Compositions

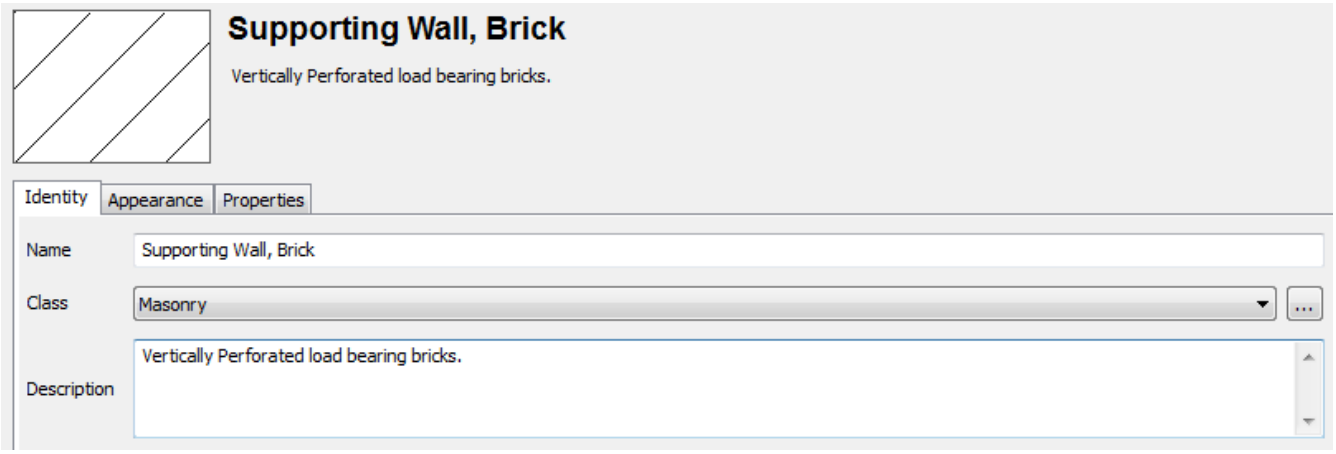
A BIM database consists of Building Materials (2.6.10) and Compositions (**??**). A Building Material contains information about one specific material: its appearance, cost, manufacturer, etc. A Composition contains information about the structure of a building element, by defining Composition layers. Each Composition layer refers to a Building Material and has a certain thickness.

Flexibility The same flexibility you have in modeling is available in attaching building information. Although Compositions are grouped into **Wall**, **Slab**, **Roof** and **Generic**Compositions, any type of Composition can be attached to any dwg entity. When attached to a 3D Solid, with specific geometrical properties, the thickness of the Composition is used to define the thickness of walls and slabs, and the cross section will show the wall or slab structure by applying the hatch patterns from the various layers on the section geometry. When a 3D Solid is assigned a Composition with a fixed thickness or a minimal thickness, and this thickness can not be applied to the geometry, the section appears in red to indicate the incompatibility between the selected Composition and the geometry of the solid.

BIM Compositions panel All BIM database features are accessible from the dockable BIM Libraries (**??**) panel.

Materials A Material definition describes the appearance and properties of a Building Material. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMMaterial.png})

Figure 2.303



Composition layers A **Composition** describes the structure of a building element by an ordered set of composition layers. Each composition layer has the following fields:

- Pattern and Material Name: The Building Material for this layer.
- Function: One of (None, Structure, Substrate, Insulation, Finish1, Finish2, Membrane). Currently, this field serves for information purposes only.
- Thickness: Defines the thickness of the layer.

2.6.4 BIM Ply Editing

Overview: In this lesson, you learn how to edit the composition plies in your BIM model. **Lesson Objectives** After completing this lesson, you will be able to:

- Use the display composition setting.
- Manipulate the composition plies.

Tutorial: How to extend a Ply Click here (2.5.6) to watch

About BIM Ply Editing

The BIM Ply Editing function allows you to manipulate each composition plies for your building elements. In BricsCAD, each building element which has a composition attached includes a Display composition property. When the composition plies are displayed in the model you can select and edit them individually.

Access  **Properties panel:** Display Composition to activate the composition plies.

Procedure: Manipulating the composition plies

1. Select the building elements that have composition properties.
2. Turn the display composition on using the display composition drop-down list in the properties bar.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMPropertiesPanelDisplayCompo
The composition plies display in the model, as well as the render materials. **Note:** If the render materials of the selected composition are not downloaded, the entity will be displayed in red.

3. Place your mouse cursor near the composition ply that you want to edit, then press the TAB key until its Ply Face (2) highlights.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMPlyEditing1.png}) **Note:** The highlighted Ply Face takes the detected boundary color (default = 95).

4. To extend the highlighted composition ply, select BIM Drag from the Quad and drag it to another building composition ply (3).

In the following illustration, the ply editing process is shown through the floor and wall elements. For this example, the concrete layer of the floor is dragged to the insulation layer of the wall. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

Figure 2.304

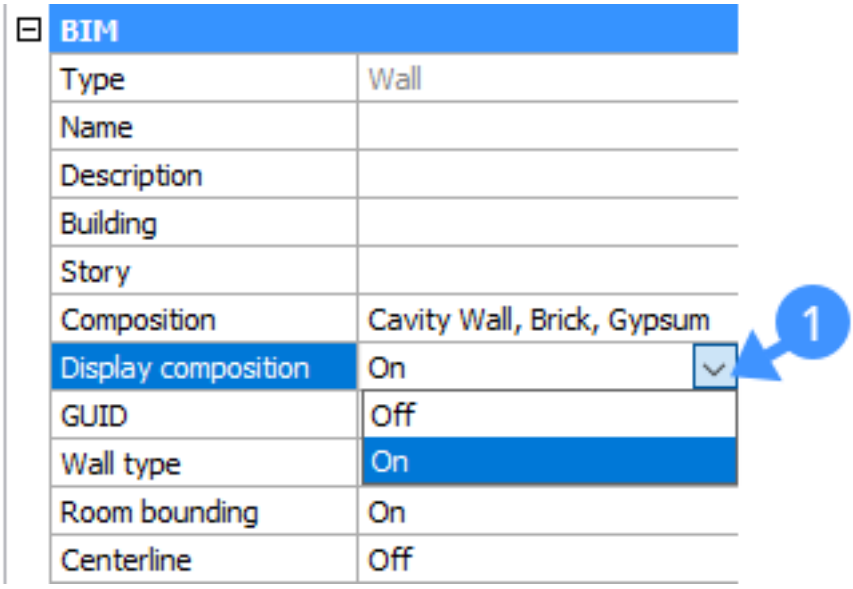


Figure 2.305

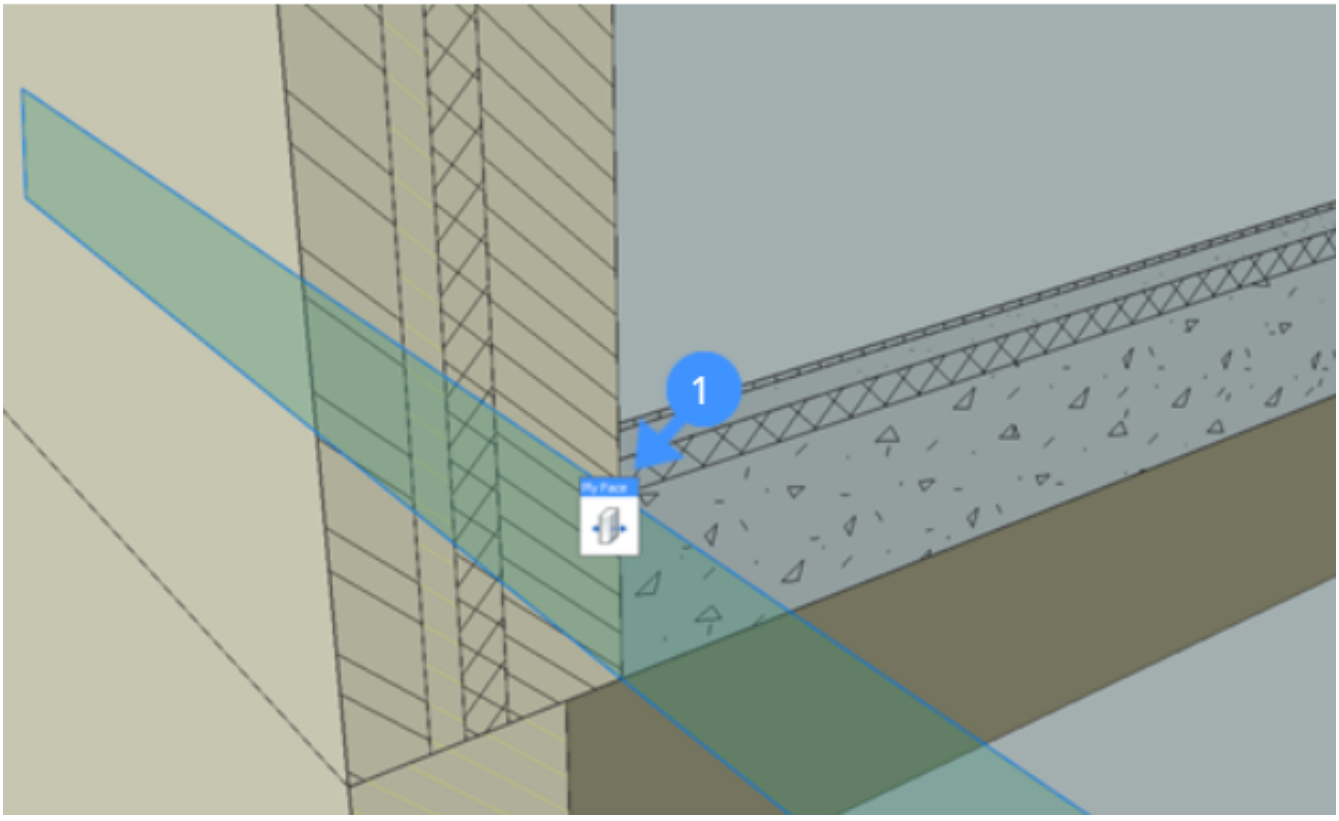
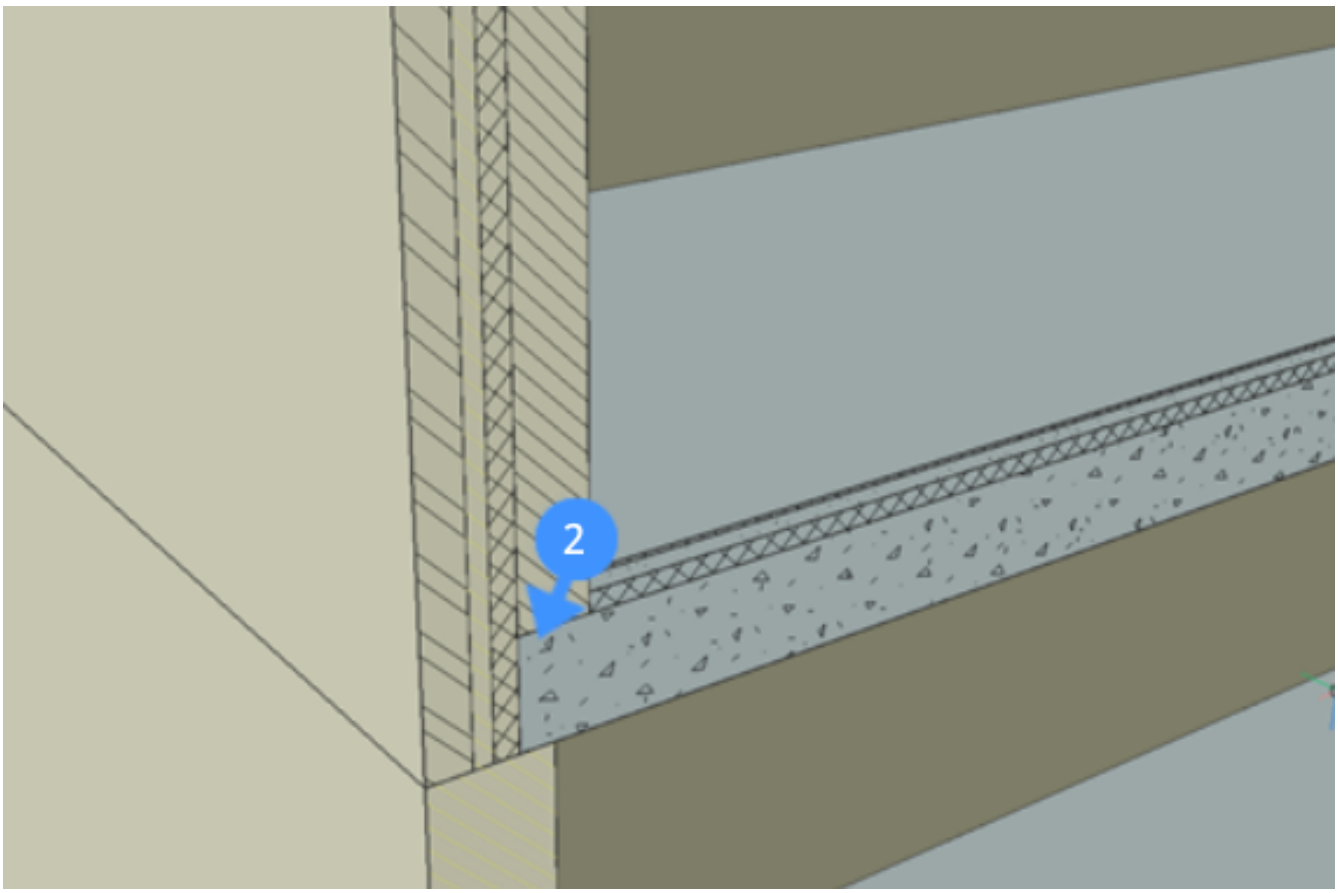


Figure 2.306



2.6.5 BIM Project Info

Command: BimProjectInfo (6.7.19) **Overview:** In this lesson, you learn how to use the Project Info dialog box to set up your project and library database of your BIM models. **Lesson Objectives** After completing this lesson, you will be able to:

- Specify your project and library databases.
- Understand the difference between embedded and external databases.
- Use the BIM database file to set the library database.
- Filter the attached tags of the building materials and compositions on the database.

About

The BIM Project Info allows you to control the location, properties, and content of the Project and Library databases. When you launch the BIMPROJECTINFO command in BricsCAD, the BIM project info displays in a dialog box that includes certain components and tabs. Project and library databases of BIM models can be changed using this dialog box.  **Keyboard:** bimprojectinfo

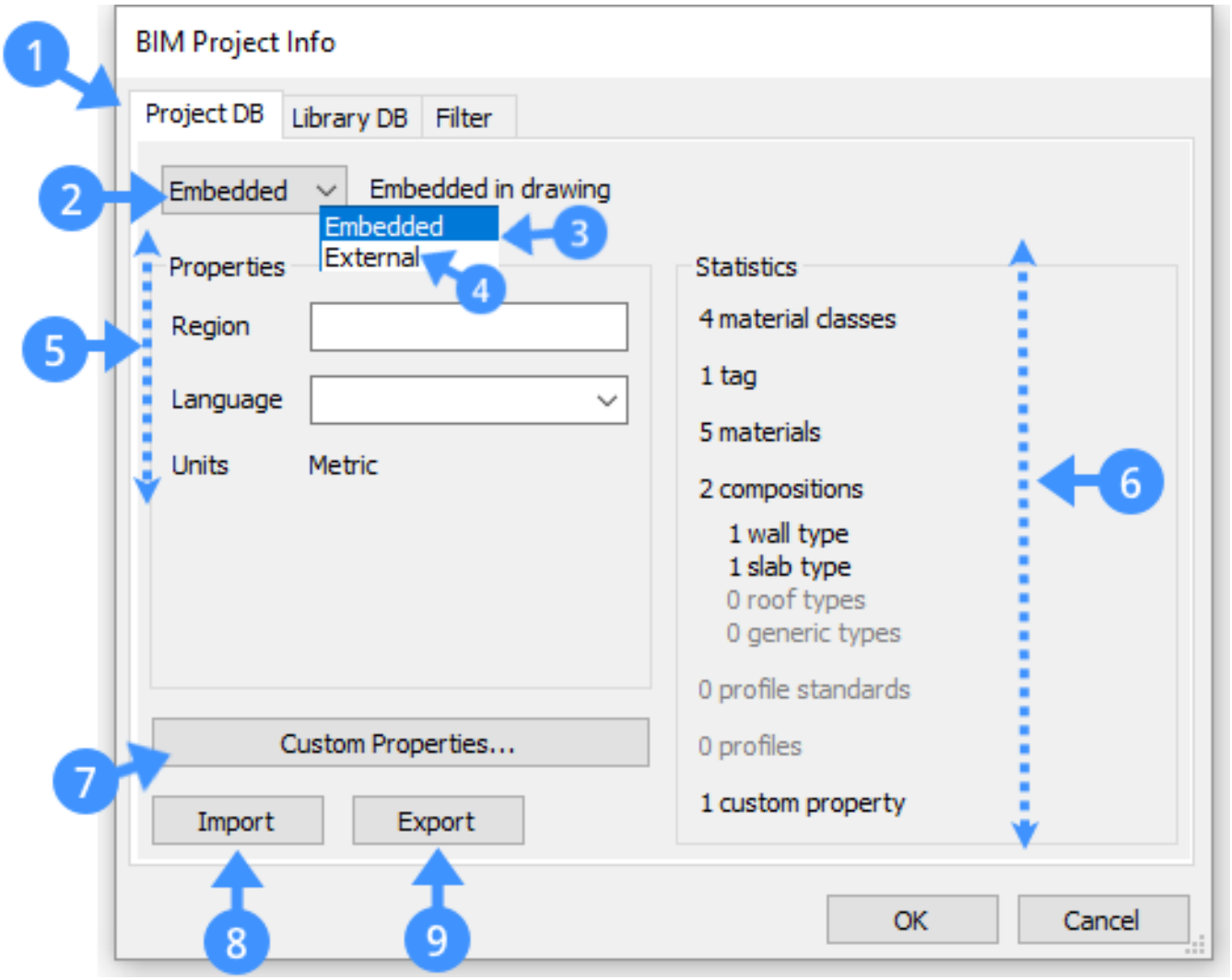
Procedure: Setting the project database

1. Open the BIM Project Info dialog box by typing BimProjectInfo in the command line. The following illustration shows the dialog box when the Project DB tab is current. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enl

Project database (1)	Shows the BIM project info.
Project database drop-down list (2)	Choose between Embedded or External.
Embedded (3)	Saves the project library in the drawing file.
External (4)	Saves the project library in a BIM database file (.bsyslib).
Properties (5)	Shows the properties of the BIM project e.g. region, language, and units.
Statistics (6)	Shows the location, properties, and the content of the current project library of the BIM model.
Custom properties (7)	Adds custom properties to materials and compositions.
Import (8)	Imports the selected .xml library files.
Export (9)	Saves .xml library files in the defined location.

2. To change the project database from Embedded to External or vice versa, select the desired option in the Project Database drop-down list. **Note:** The external database keeps the database independent from the current drawing. This is especially useful if you want to share the library between multiple drawings.
   - When switching from Embedded to External a File dialog displays. Do one of the following:
     - Type a name in the File Name field to export the content of the project library to a new .bsyslib file, then click the Open button.
     - Select an existing .bsyslib file, then click the Open button. The current content of the project library is merged with the selected .bsyslib file.
   - When switching from External to Embedded the content of the external library is copied to the embedded project library.
3. To specify the Project Database properties, such as Region and Language, fill out the Region field and choose one of the language options in the drop-down list.
4. Optionally, use the custom properties button to add material and composition custom properties in the Project Database.
5. Press OK button to accept it.

Figure 2.307


BIM Project Info
Project DB
Library DB
Filter
Embedded
Embedded in drawing
Embedded
External
Properties
Region
Language
Units
Metric
Statistics
4 material classes
1 tag
5 materials
2 compositions
1 wall type
1 slab type
0 roof types
0 generic types
0 profile standards
0 profiles
1 custom property
Custom Properties...
Import
Export
OK
Cancel
1
2
3
4
5
6
7
8
9

Procedure: Setting the library database

1. Click the Library DB tab page in the BIM Project Info dialog box. The following illustration shows the Library Database section in the dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Buildin

2. To select a BIM database file (.bsyslib), click the **Browse (10)** button of the **Path** field, then choose a database file in the file dialog.

3. Optionally, check the **Set as default (11)** option to set the selected database as default library database for next projects.

Note: You can edit the default library settings (12, 13) in the Settings dialog under Program options/ System. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/DefaultLibSettings.png})

Figure 2.308

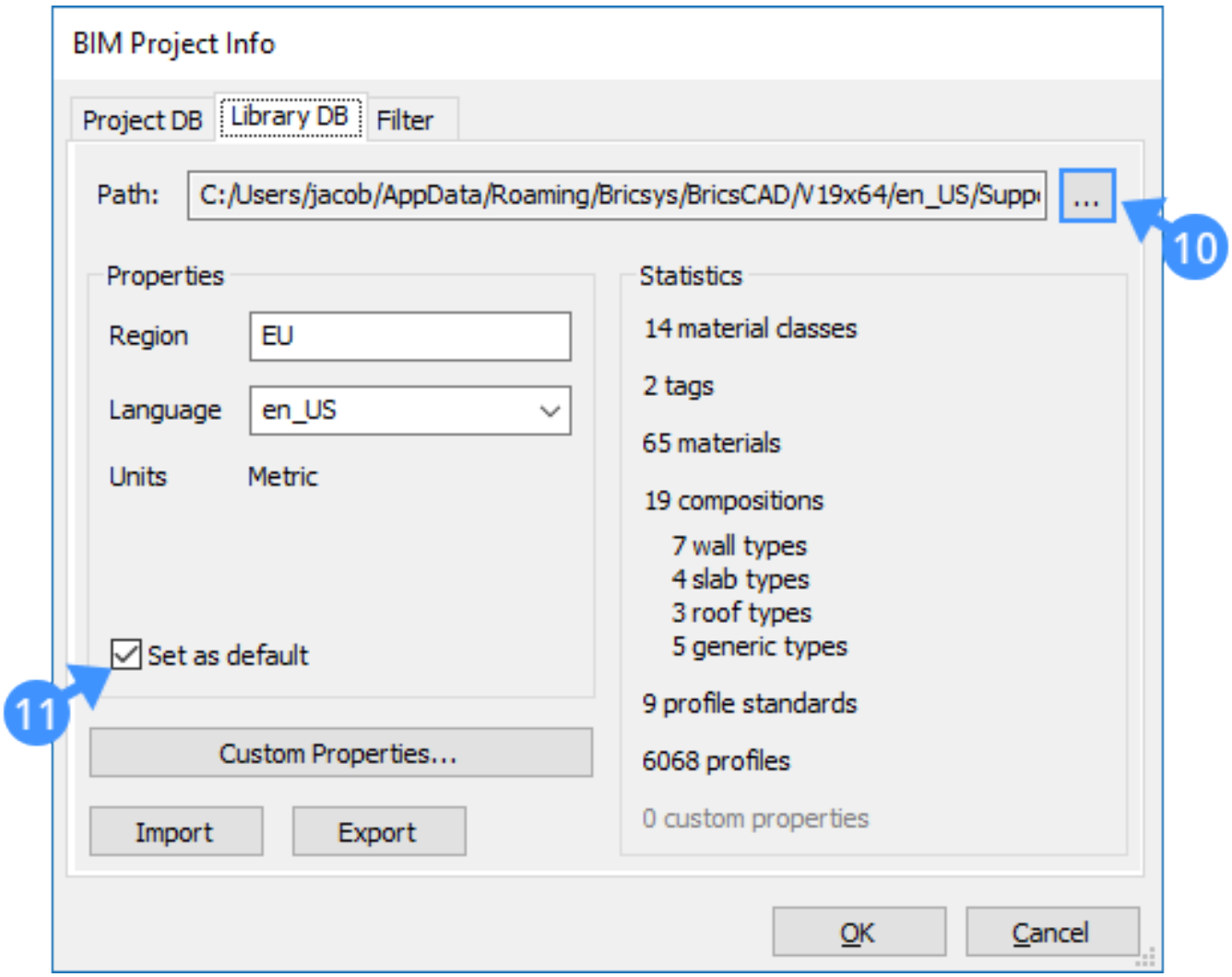
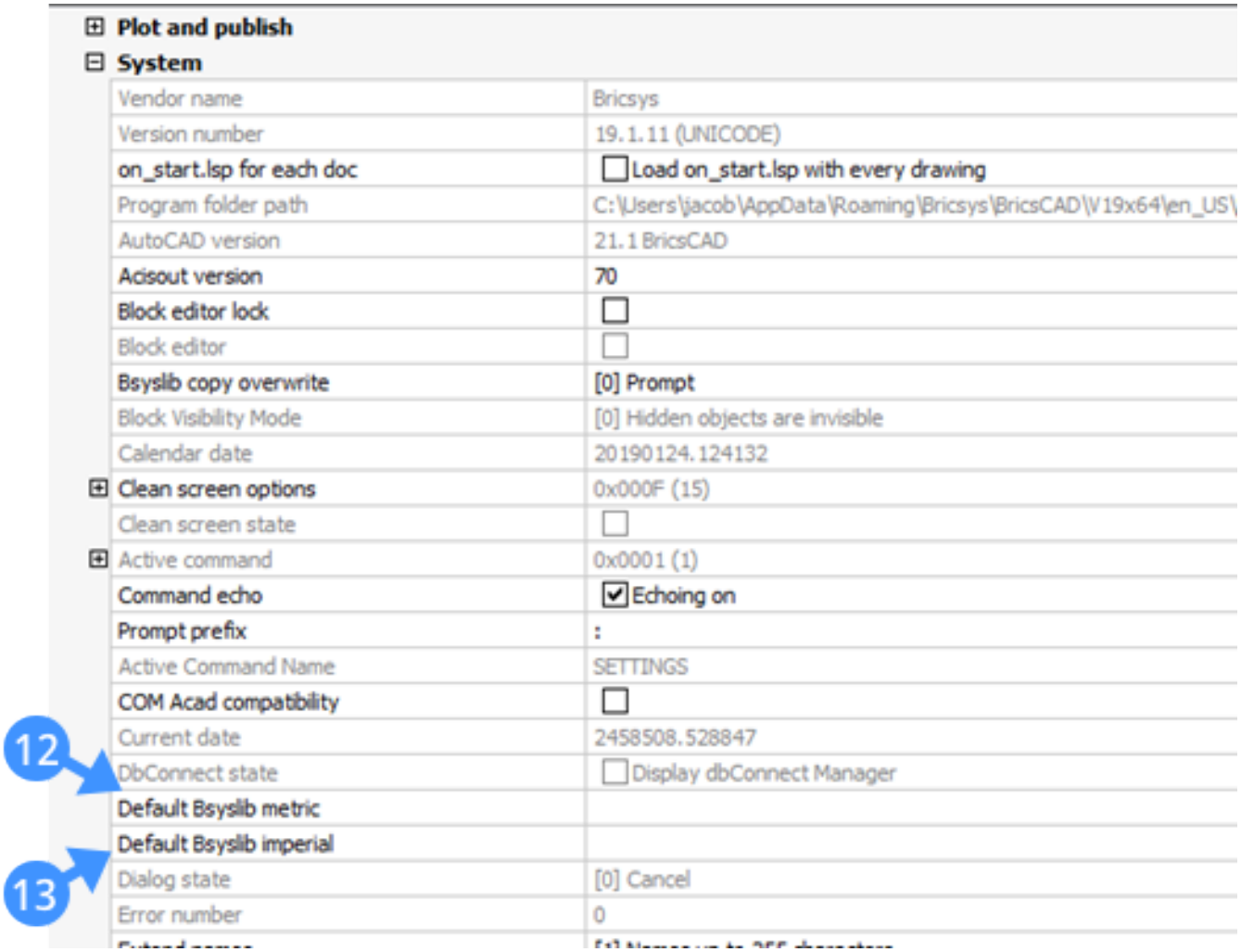
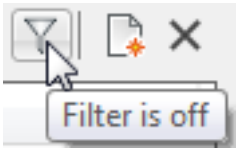

BIM Project Info
Project DB
Library DB
Filter
Path:
C:/Users/jacob/AppData/Roaming/Bricsys/BricsCAD/V19x64/en_US/Supp
...
10
Properties
Region
EU
Language
en_US
Units
Metric
Set as default
11
Custom Properties...
Import
Export
Statistics
14 material classes
2 tags
65 materials
19 compositions
7 wall types
4 slab types
3 roof types
5 generic types
9 profile standards
6068 profiles
0 custom properties
OK
Cancel

Figure 2.309


Plot and publish
System
Vendor name
Bricsys
Version number
19.1.11 (UNICODE)
on_start.lsp for each doc
Load on_start.lsp with every drawing
Program folder path
C:\Users\jacob\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\
AutoCAD version
21.1 BricsCAD
Acisout version
70
Block editor lock
Block editor
Bsyslib copy overwrite
[0] Prompt
Block Visibility Mode
[0] Hidden objects are invisible
Calendar date
20190124.124132
Clean screen options
0x000F (15)
Clean screen state
Active command
0x0001 (1)
Command echo
Echoing on
Prompt prefix
:
Active Command Name
SETTINGS
COM Acad compatibility
Current date
2458508.528847
DbConnect state
Display dbConnect Manager
Default Bsyslib metric
Default Bsyslib imperial
Dialog state
[0] Cancel
Error number
0
12
13

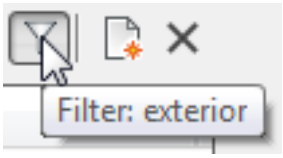
The project filter

To eachBuilding Material (2.6.10)orComposition (6.7.38), any number of tags can be attached. Any string can be entered as a tag. Each of the tags can be used as a filter on the database. There is a button at the top of each list ( ), to switch the filter on or off. If the filter is on, only Compositions and Building Materials that match at least one of the filter tags will be listed. The**Filter**button ( ) has three states:

- If the project filter is empty, the button is not active (**BIM Compositions**panel) or is not available (**Materials**and**Compositio**
- The project filter is defined, but not active. 

- The project filter is defined and active. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Build
  A tooltip shows the current project filter.

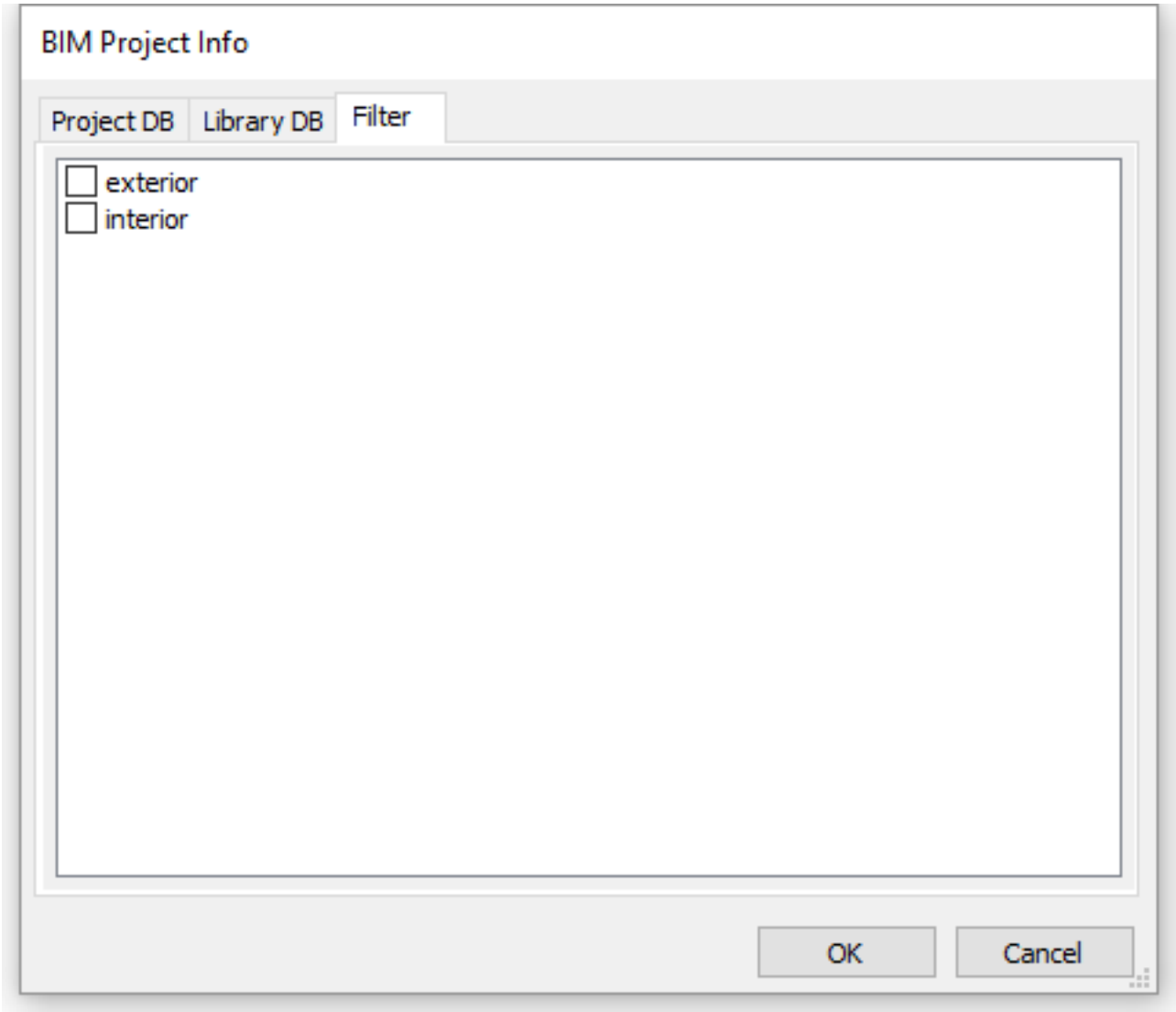
Figure 2.310



To set the project filter

1. Select the**Filter**tab page on the**BIM Project Info**dialog box.
2. Check the tags you want to use in the filter. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/

Figure 2.311


BIM Project Info
Project DB
Library DB
Filter
exterior
interior
OK
Cancel

2.6.6 BIM Room

Command: BimRoom (6.7.26) **Overview:** In this lesson, you learn how to create BIM Room entities from enclosed spaces. **Lesson Objectives** After completing this lesson, you will be able to:

- Define room areas.
- Change the display of a room.

Tutorial: How to define Room Click here (2.5.5) to watch

About: Create Room

You can create rooms from enclosed spaces. When the wall elements enclose a space in your model, you can use the **Create Room** tool to define a BIM Room. The 3D room geometries added to the current model are associated with room tags. The room tag is a block that contains a hatch and attributes. The default attributes of the room tag include a room name, room number, and the room area. The values of each attribute can be changed using the properties panel.  Menubar: **BIM > Room** Toolbar: **BIM > Room** Quad: **Model > Room** Keyboard: **BIMROOM**

Defining a BIM room

1. Select the **Room** tool in the **Model** command group of the Quad. **You are prompted:** Pick a point inside the room or <select> To define a BIM room, pick a point inside the room area (1) in your BIM model. When moving the cursor inside the room area to pick a point, the **Dynamic UCS** enables to define the bottom plane of the room. In this plane, an area enclosed by wall elements or any room bounding elements is found and a room tag (2) is placed in the center of the room. **Room Bounding elements:** walls, curtain walls, slabs, columns, and beams. **Note:** Room tags are based on template drawing, allowing the user to customize their appearance. The room tags can be moved using the grip point (3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMRoom.png})
   - Optionally, divide a large space in different rooms, to do so first separate the space using linear entities, then classify the linear entities as a room bounding elements.
2. Edit the room properties in the Properties Bar. The following illustration shows the properties of a room. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/RoomBIMProperties.png})

Building (4)	Shows the building. Click the field, then select a building.
Story (5)	Shows the story of a building. Click the field, then select a story.
Room representation (6)	Includes three different options of room representations. When the option is set to **None** only room tag displays in the model. **Hatch** option displays room tag and a solid hatch. **Solid** option displays the Room as a 3D solid in the model.
Interior elevations (7)	Controls the display of the Interior Elevations sections. If the interior elevations are created for a room, these elevations can be shown and hidden quickly using this property.

Figure 2.312

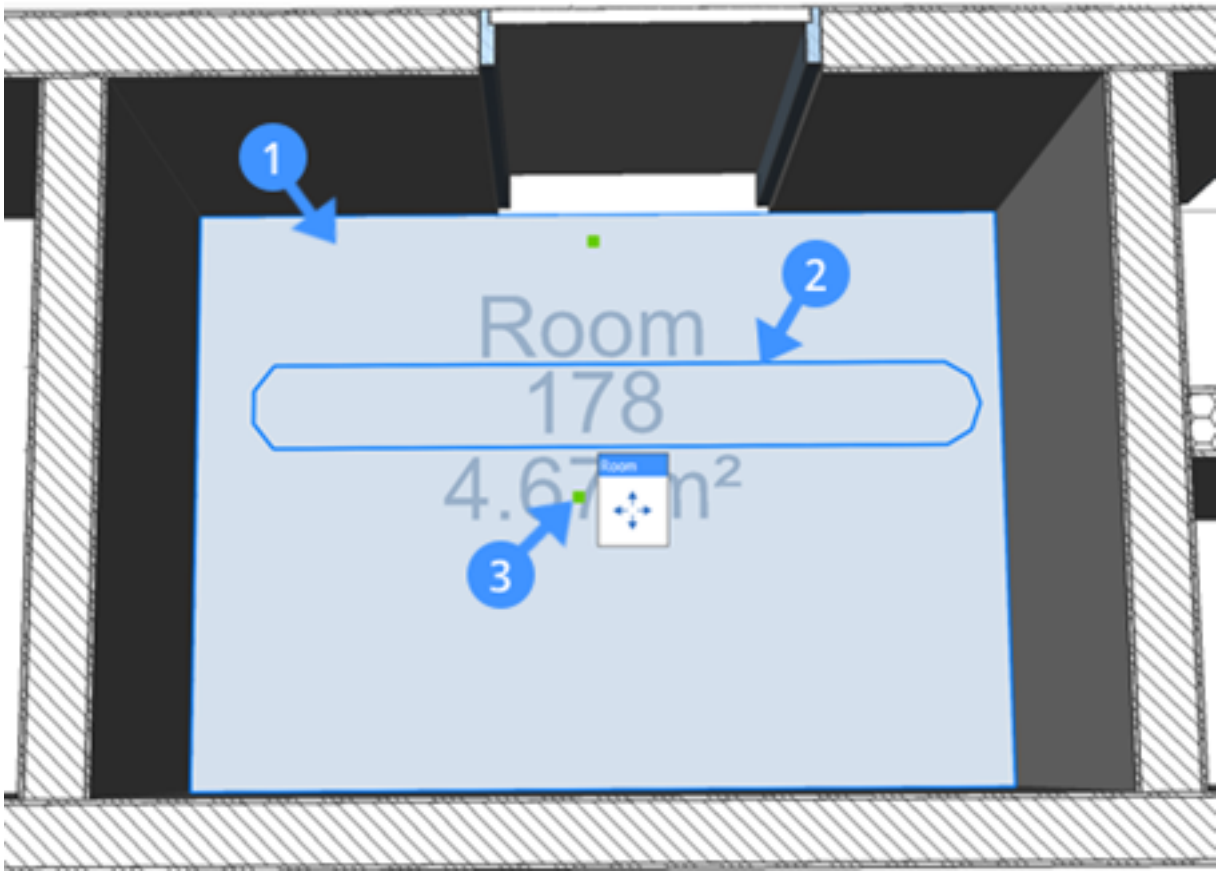
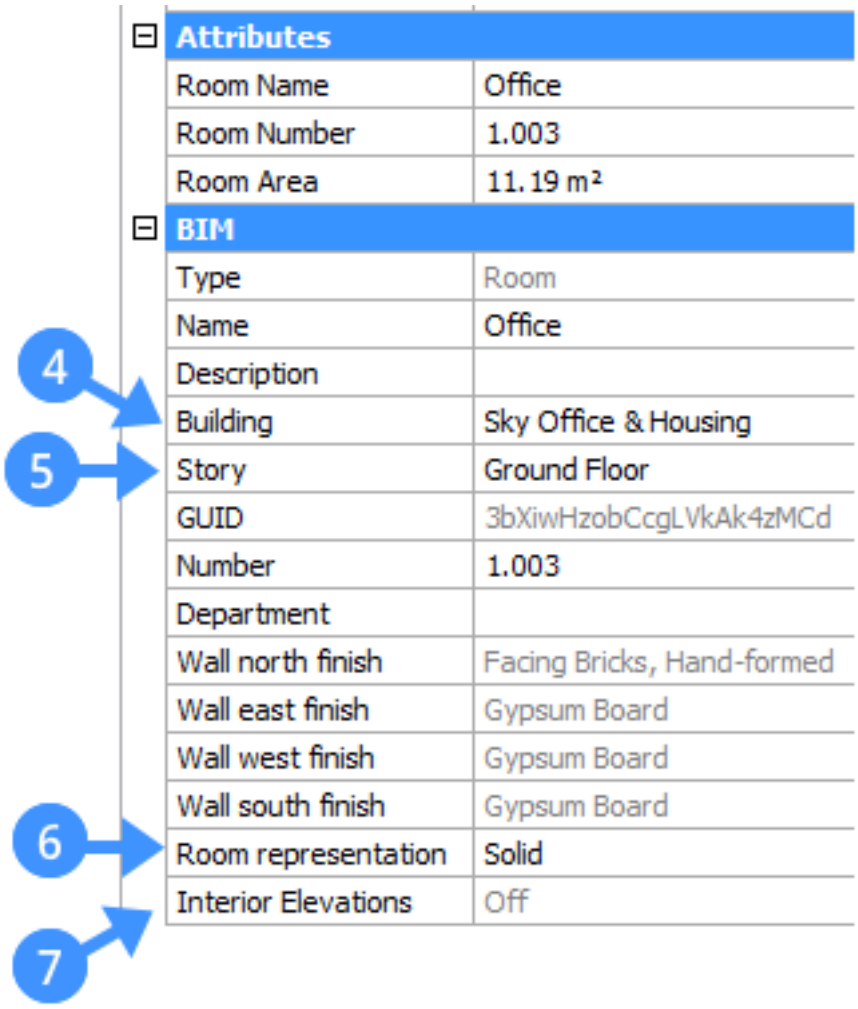


Figure 2.313


Attributes
Room Name
Office
Room Number
1.003
Room Area
11.19 m²
BIM
Type
Room
Name
Office
Description
Building
Sky Office & Housing
Story
Ground Floor
GUID
3bXiwHzobCcgLVkAk4zMCd
Number
1.003
Department
Wall north finish
Facing Bricks, Hand-formed
Wall east finish
Gypsum Board
Wall west finish
Gypsum Board
Wall south finish
Gypsum Board
Room representation
Solid
Interior Elevations
Off
4
5
6
7

2.6.7 Classification Codes

Overview: In BricsCAD, you can organize your BIM models with standard classification codes that are used in the construction industry. These codes can classify elements according to national or company standards.

- A set of pre-installed classification systems is available to attach to the BIM.
- Multiple classification systems can be used in the same BIM.

Tutorial: How to use Classification Codes Click here (2.5.11) to watch

About

Assigning classification codes organizes the BIM and increases the level of detail.For instance, they allow to organize library materials and project reports, estimate project costs and define product specifications. Currently, GUBIMClassCA, GUBIMClassES, Master Format, NLSfb, OmniClass, UniClass2015, Uniformat, and VMSW are available classification systems in BricsCAD.  Keyboard:**bimproperties** Toolbar:**BIM | Properties** Quad Tab: **BIM| Properties** Ribbon Tab | Panel:**Database | Properties**

Assigning classification codes to the BIM

1. Open the BIM Properties dialog box from the Quad (**BIM:**Model tab). **Note**: Quad should be in 'No Selection' state. A dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/

2. Click the Import (1) button at the bottom-left corner of the dialog box. A file dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/SelectxmlfiletoimportDialogBox.png})

3. Choose the classification system on the list (2), and click Open (3) to import it into the BIM. **Note:** If the file dialog does not show the Classification folder, you should manually enter this path in the address bar:C:\Users\<User>AppDa V19x64\en_US\Sup Support\Bim\Classification. **Available classification systems:**

GUBIMClassCA	Classification system for the BIM Users group of Catalonia.
GUBIMClassES	Unified classification systemof building elements designed for the AEC industry in Spain.
Master Format	Standards forcommercial and institutional building projects in the US and Canada.
NLSfb	Standards for the AEC industry in the Netherlands.
OmniClass	Classification system for the construction industry.
UniClass2015	Standards for all sectors of the UK construction industry.
Uniformat	Standards for classifying buildingspecifications, cost estimating, and cost analysis in the US and Canada.
VMSW	Standards for the AEC industry in Belgium.

4. The classification system is added to the namespace (4), the selected classification system is added as a property set (5), and the classification code (6) is attached to this property set. By default, the classification system is applied to all sorts of building elements (7). **Optional:** Deselect the categories of building elements to which the classification system is applied. To do so, check the box next to the category name(8) (all elements in this category will be excluded), or expand the list and check the box next to the element type only. Click the OK button(9) to finalize the procedure. **Or** import another classification system following the previous steps. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMPropertiesDia

5. Select the element to which you want to assign a classification code, and open the properties panel. The classification system (10) and code (11) should be available in the properties panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/OmniClassElement.png})

Figure 2.314

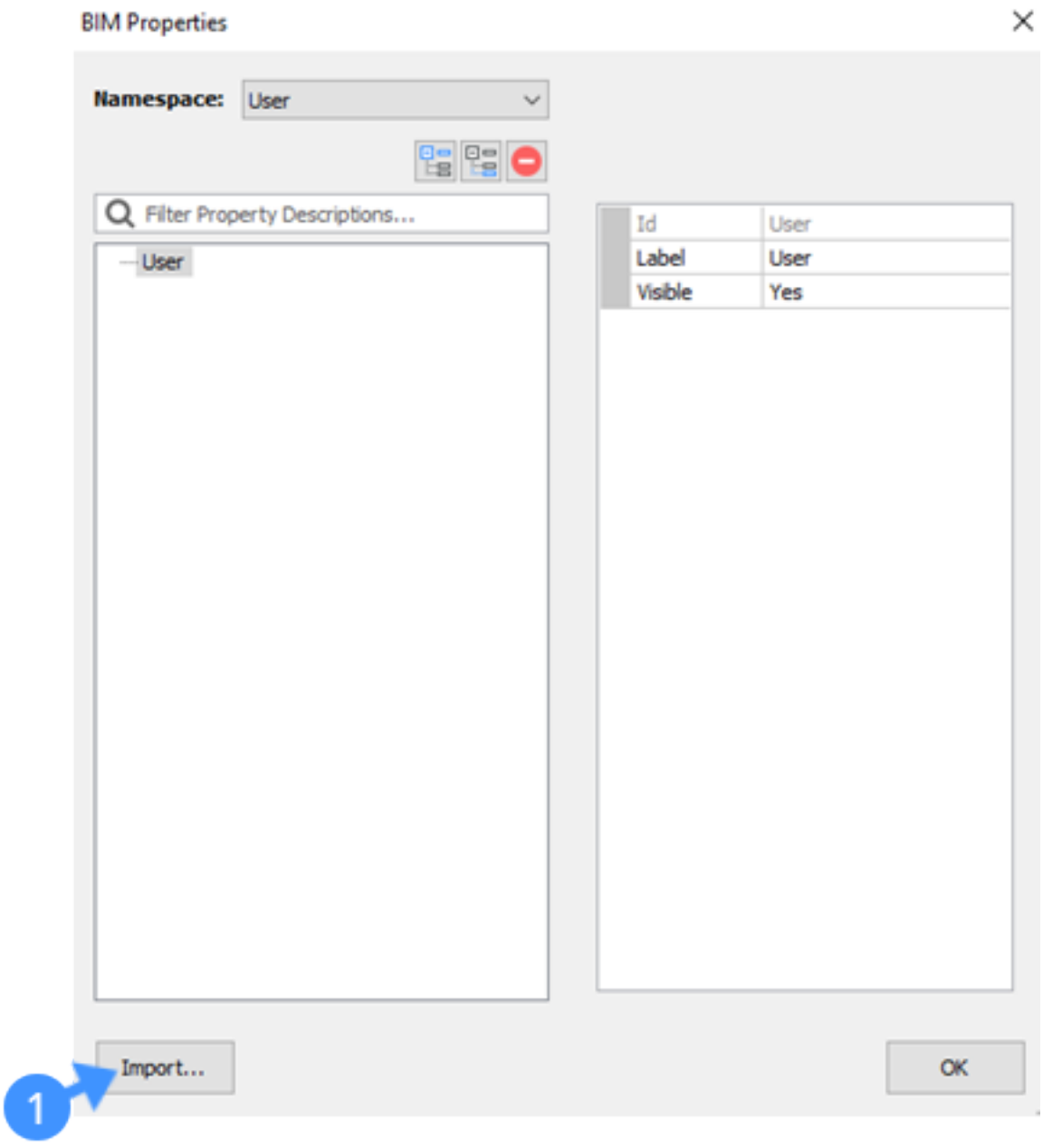
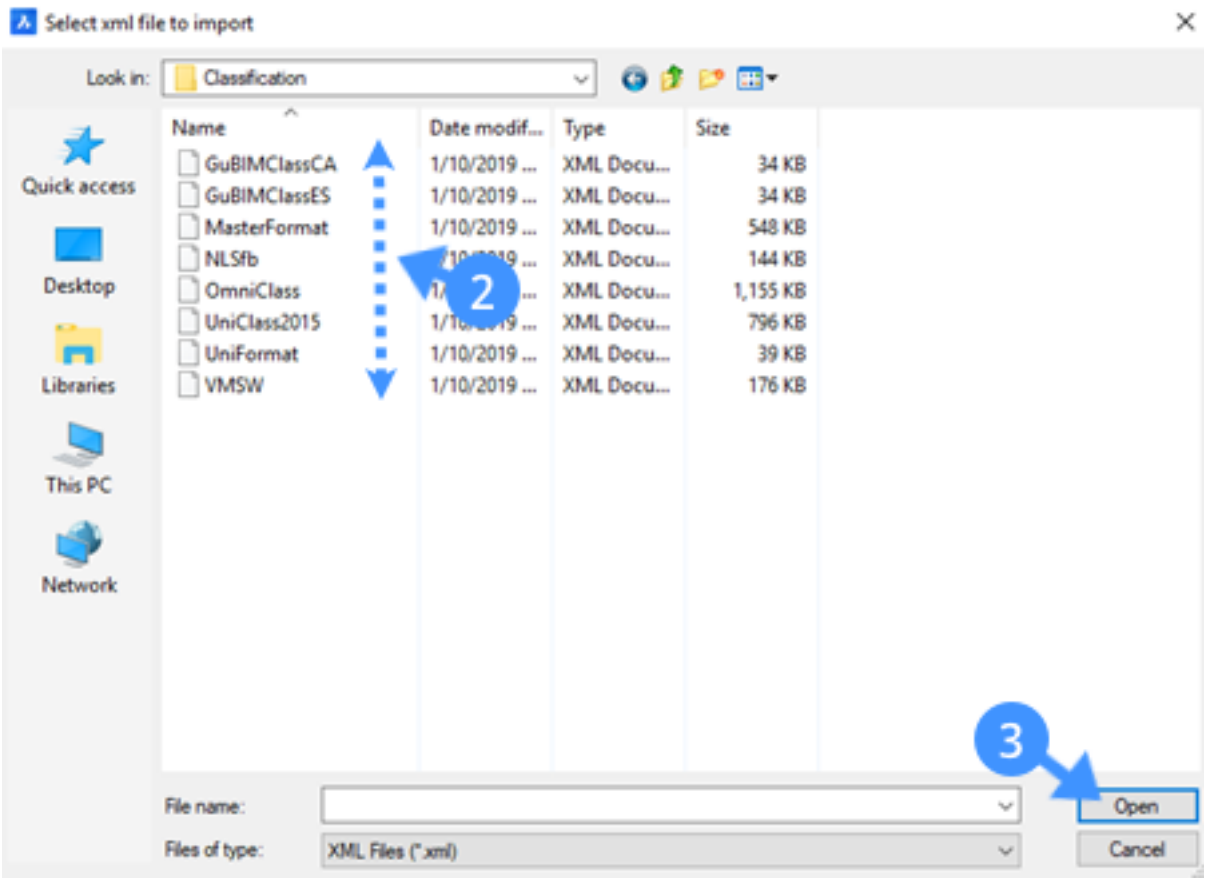

BIM Properties
Namespace:
User
Filter Property Descriptions...
User
Id
User
Label
User
Visible
Yes
Import...
OK
1

Figure 2.315



6. Click the arrow icon on the right-hand side (12), and select the classification code for the selected element in the context menu (13). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/Class
Note: Exporting the BIM to an IFC file contains the classification system for each element with assigned classification codes. These properties are exported with proper labels and descriptions. The following image shows the OmniClass Classification (14) in Solibri after exporting and importing BIM with the OmniClass Classification code from Bricsys to Solibri. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Build

Figure 2.316

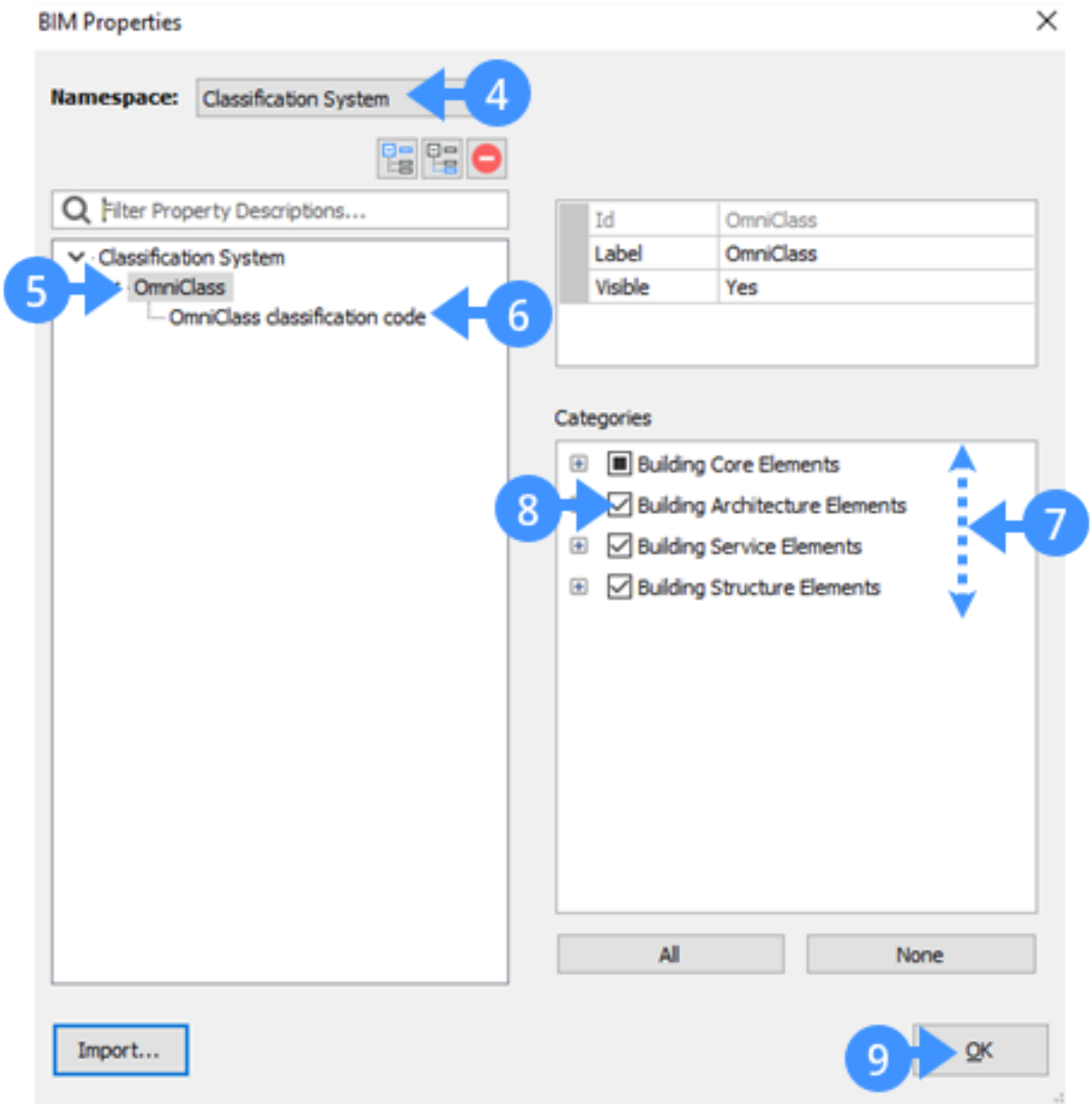

BIM Properties
Namespace:
Classification System
4
Filter Property Descriptions...
Classification System
OmniClass
OmniClass classification code
5
6
Id
OmniClass
Label
OmniClass
Visible
Yes
Categories
Building Core Elements
Building Architecture Elements
Building Service Elements
Building Structure Elements
8
7
All
None
Import...
9
OK

Figure 2.317

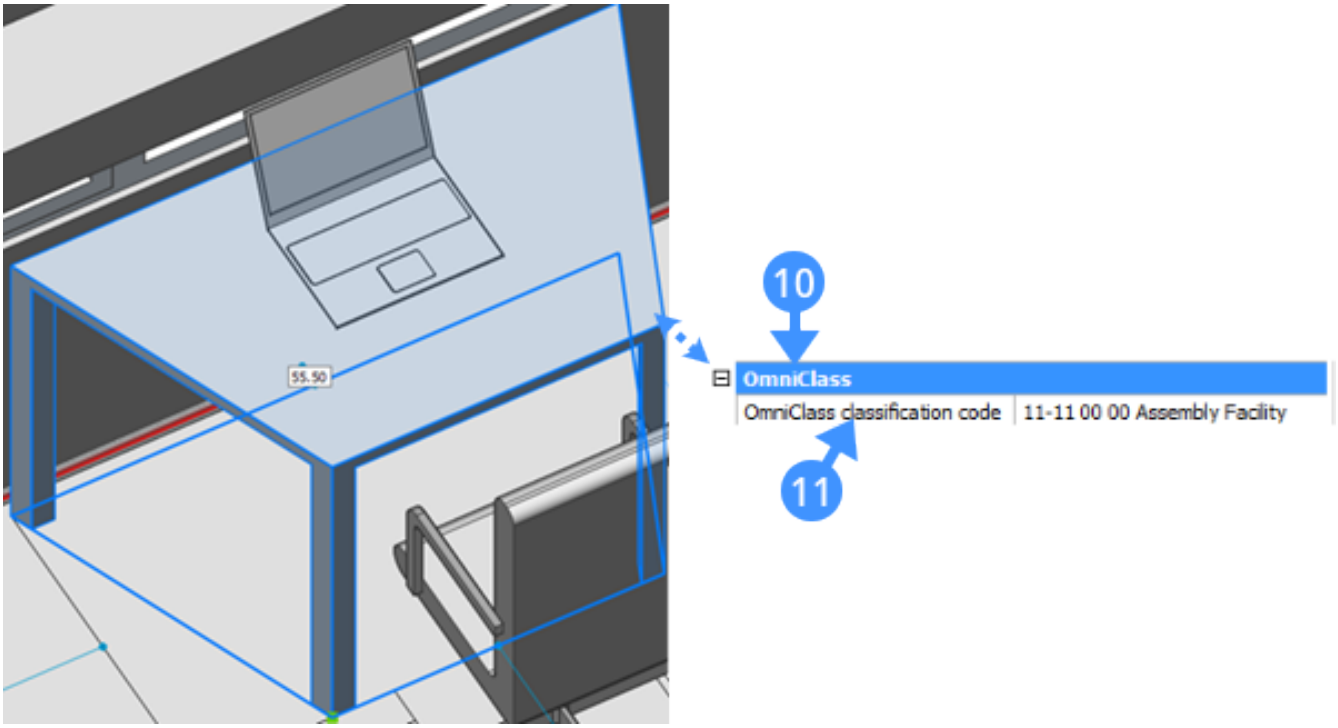

55.50
10
OmniClass
OmniClass classification code
11-11 00 00 Assembly Facility
11

Figure 2.318

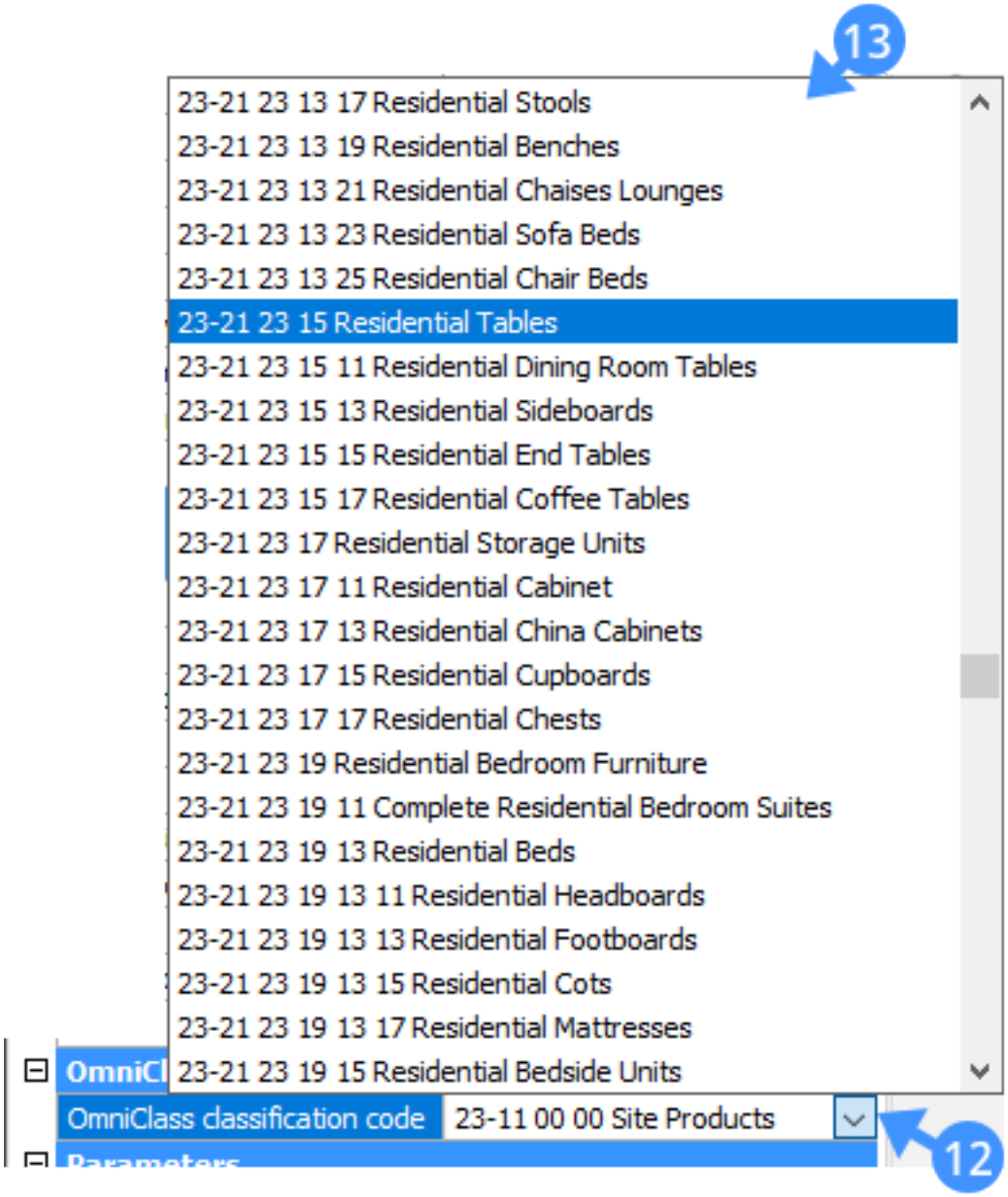
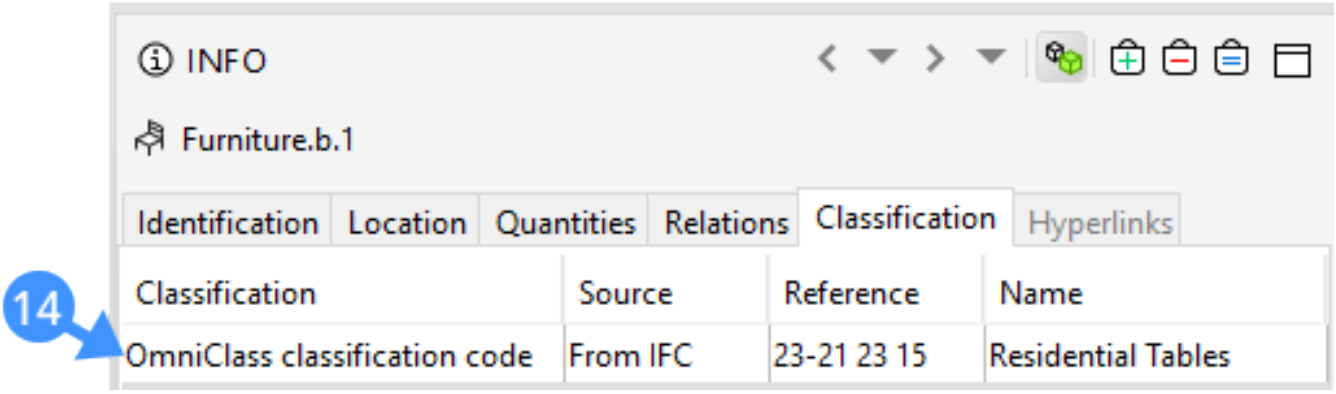


Figure 2.319



2.6.8 Custom Properties

Command: BimProperties (6.7.41) **Overview:** In this lesson, you learn how to organize properties in property sets and assign them to the building elements. **Lesson Objectives** After completing this lesson, you will be able to:

- Create, edit, and delete properties and property sets.
- Assign the customized property sets to the desired building elements.
- Specify values for the custom properties.

Tutorial: How to create and edit Custom Properties Click here (2.5.4) to watch

About

BricsCAD provides a way to assign your own properties to desired building elements in a single command. This is called assigning custom properties. The approach is first to create a custom property set, then assigning it to the building elements to which the property set is to be applied. The method is carried out by the BIMPROPERTIES command, which allows you to create, edit, and delete the custom property set in a dialog box. This dialog box also allows you to add properties to the currently selected property set. Important to note is that these custom properties are also included in IFC export.  Keyboard: **bimproperties** Ribbon: **Data > Properties**

Using the BIM properties dialog box

1. Open the BIM Properties dialog box by typing BimProperties in the command line. The dialog box displays for specifying and editing properties of BIM projects. The following illustration shows the dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMPropertiesDialogBox.png})

(1) Namespace	Groups property sets in namespaces.
(2) IFC 2X3 namespace	Shows the IFC properties.
(3) User namespace	Shows the custom properties.
(4) Add property set button	Adds a new property set.
(5) Add property button	Adds a property.
(6) Delete button	Removes the selected Property or Property Set.

2. To create a custom property set, select the **User** option in the namespace drop-list, then click the **Add Property Set** A new property set is added to the User space. To edit the property set do the following:

   - Type a name in the Id (7) field.
   - Type a name in the Label (8) field.

   The Label name appears in the properties tree. If the Label name is not defined, the Id name displays.

   - Select the building element categories (9) the property set applies to.

   When you expand one of the categories, a different type of building elements (10) displays, check the box next to its name to associate with the property set. The following illustration shows the BIM properties dialog box after inserting a new property set. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingI

3. To add a property to the currently selected property set, first, select a property set, then click the **Add Property** button to add a new property. To edit the property do the following:

   - Type a name in the **Id**
   - Type a name in the **Label**
   - Optionally, type a description in the **Description**
   - Select the property type. The options are **Boolean, Integer, Real,** and

   The following illustration shows the BIM properties dialog box after inserting a new property. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/AddingNewProperty.png})

4. To remove the property set/property, select a property set/property in the tree, then click the Delete button to remove.

Figure 2.320

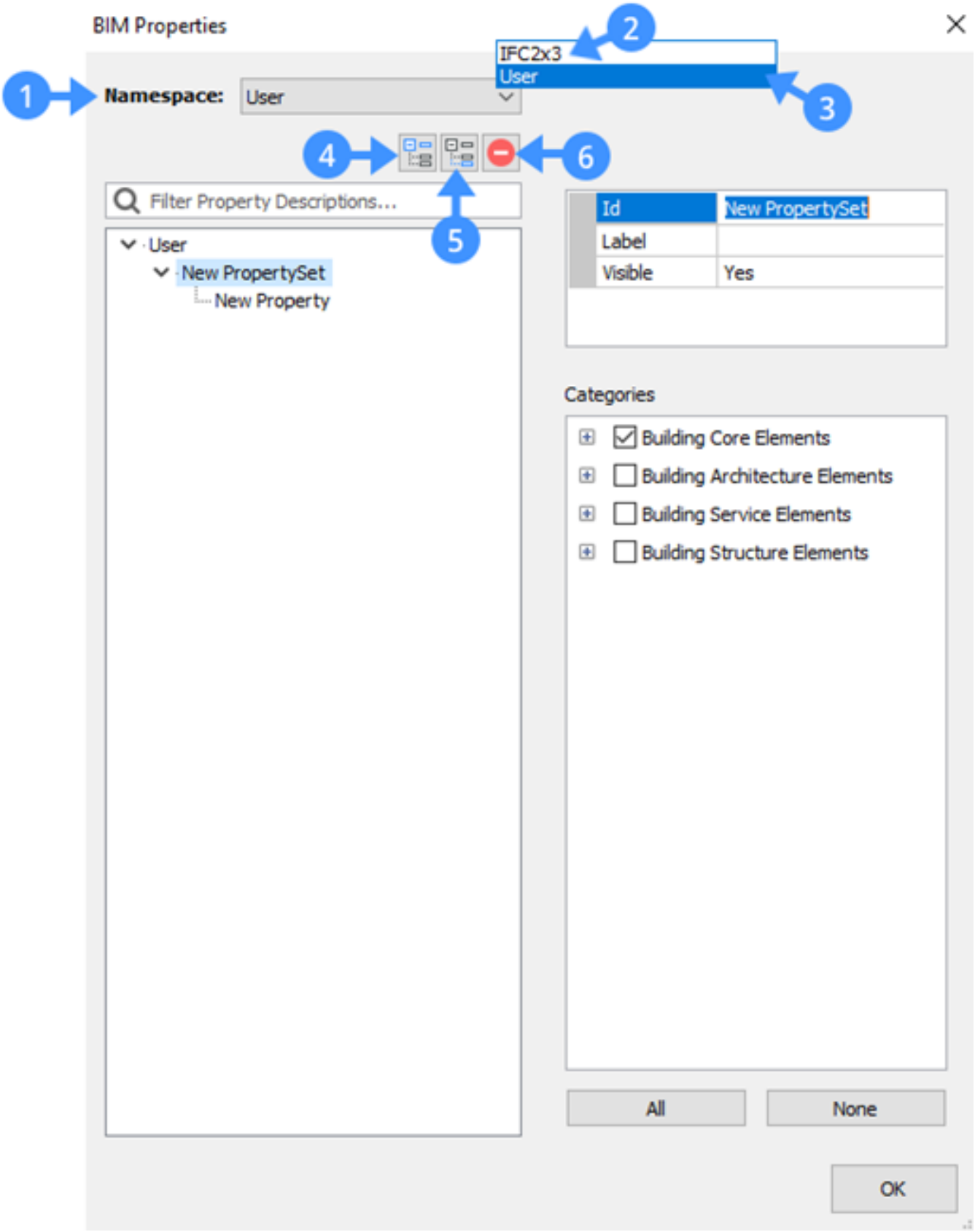

BIM Properties
1
Namespace:
User
2
IFC2x3
User
3
4
5
6
Filter Property Descriptions...
User
New PropertySet
New Property
Id
New PropertySet
Label
Visible
Yes
Categories
Building Core Elements
Building Architecture Elements
Building Service Elements
Building Structure Elements
All
None
OK

Figure 2.321

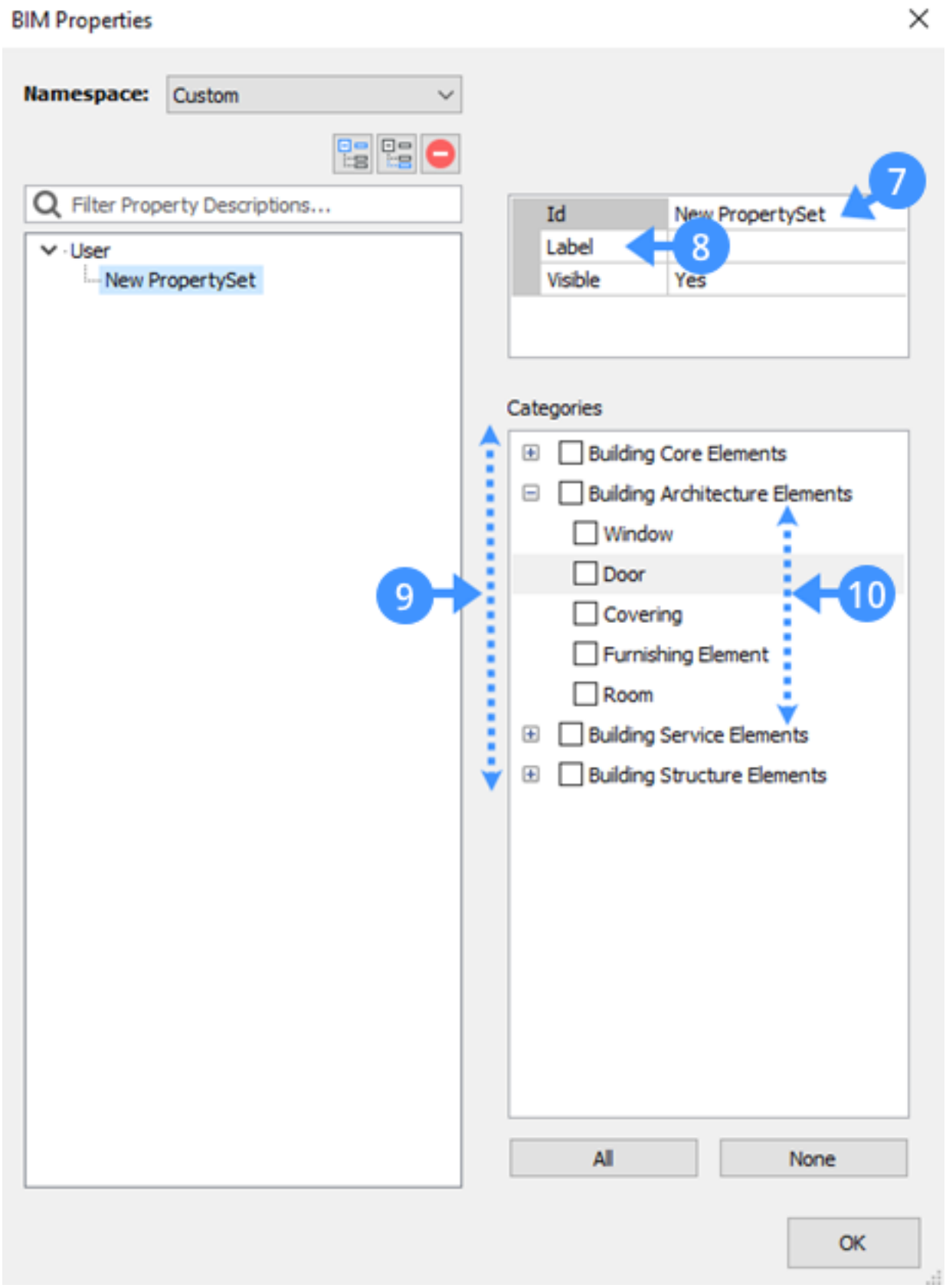

BIM Properties
Namespace:
Custom
Filter Property Descriptions...
User
New PropertySet
Id
New PropertySet
Label
Visible
Yes
7
8
Categories
Building Core Elements
Building Architecture Elements
Window
Door
Covering
Furnishing Element
Room
Building Service Elements
Building Structure Elements
9
10
All
None
OK

Figure 2.322

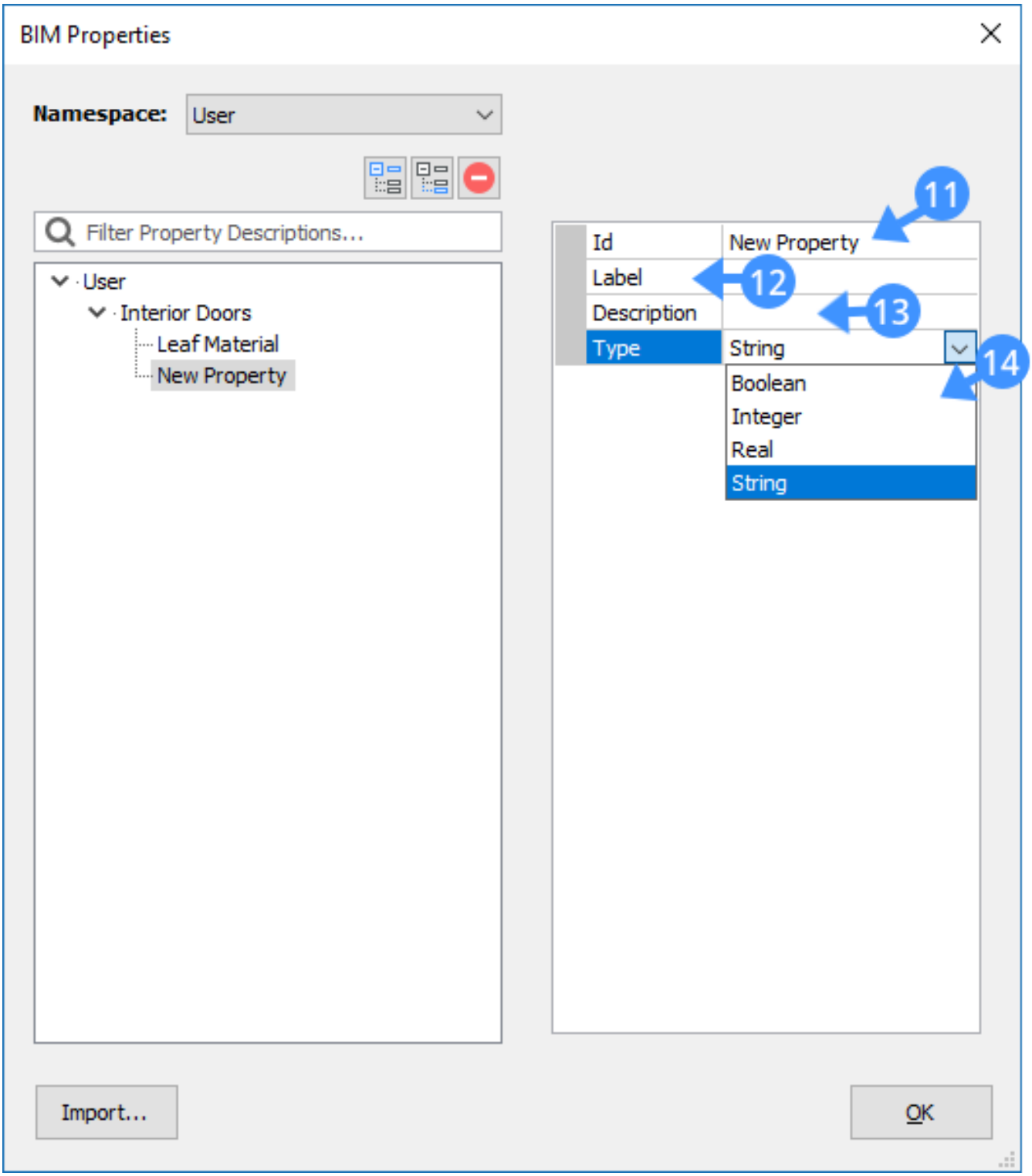

BIM Properties
Namespace:
User
Filter Property Descriptions...
User
Interior Doors
Leaf Material
New Property
Id
New Property
Label
Description
Type
String
Boolean
Integer
Real
String
11
12
13
14
Import...
OK

Figure 2.323



5. Press OK button to accept it. Now if you select a building element, which is associated with the property set, you will see its newly assigned properties (15) are placed in a new category (16) in the properties panel. To assign a value to your custom property, select it in the properties panel, then type a new value in its field (17). The following illustration shows the custom properties in the properties panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/AssignedCustomProperty.png})

2.6.9 The BIM Project Browser

Commands: DATAEXTRACTION (6.10.1), BIMSECTIONUPDATE (6.7.29), BIMSECTIONOPEN (6.7.28), The BIM Project Browser allows you to easily navigate through the entire content of a project: models, sections, drawings and schedules.

To open the Project Browser panel

1. Move the cursor over a toolbar, ribbon item or a dockable panel (1.12.7), then right click.
2. Choose **BIM Project Browser** in the context menu.

Creating a BIM project

When creating a BIM project you must select one or more model drawings.

1. Click **the Create Project...** button on the **Project Browser** panel. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h
2. (optional) Click the Path list button and select a parent folder. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel
3. Type a name in the **Name** field.
4. Select the model drawings to include. All drawings in the project folder and subfolders, in case a parent folder is selected, are listed.
5. Click the **OK** button. The select models are listed in the **Models** section of the **Project Browser**. Currently closed models are greyed out. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocu

Figure 2.324

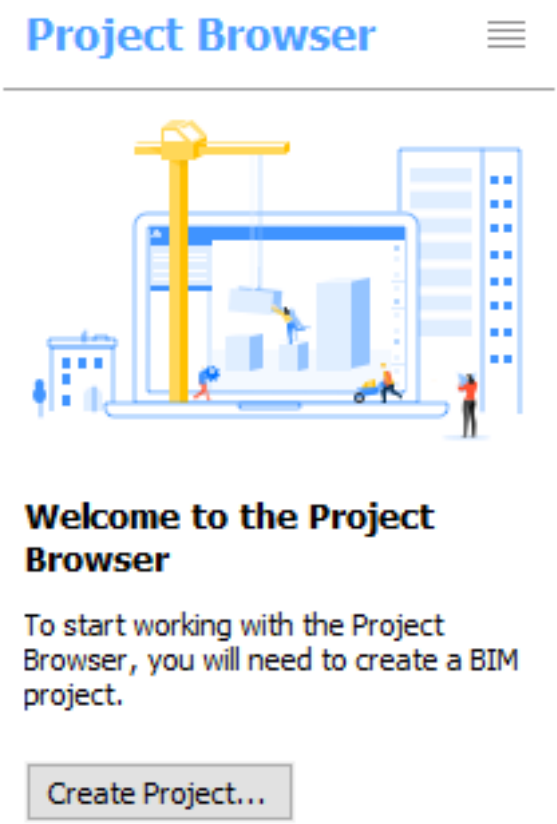


Figure 2.325

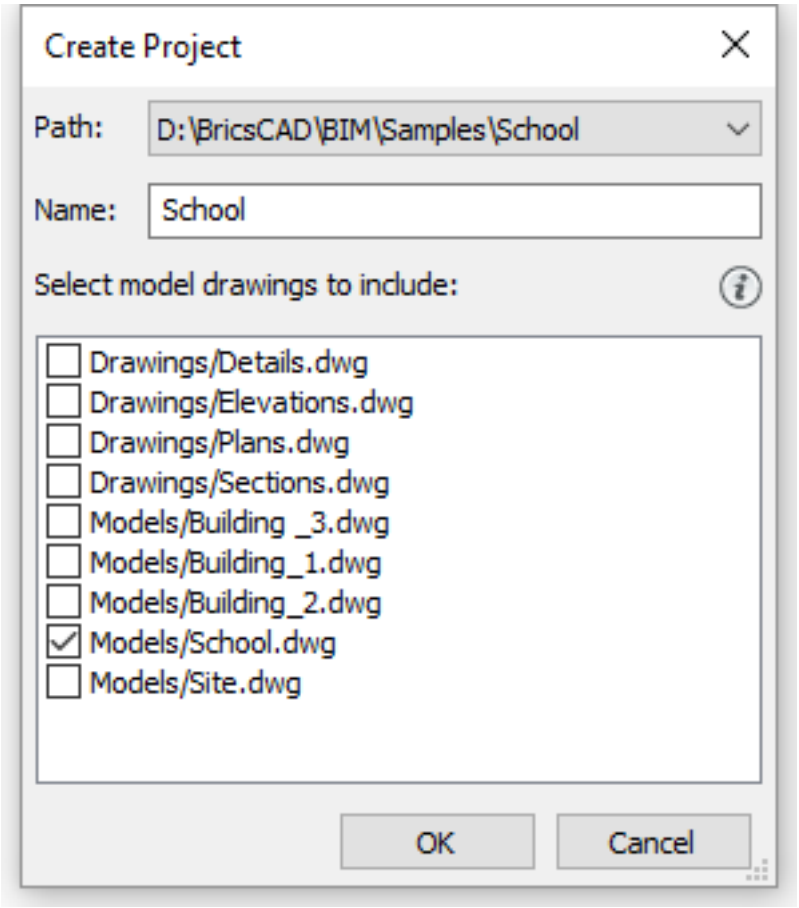
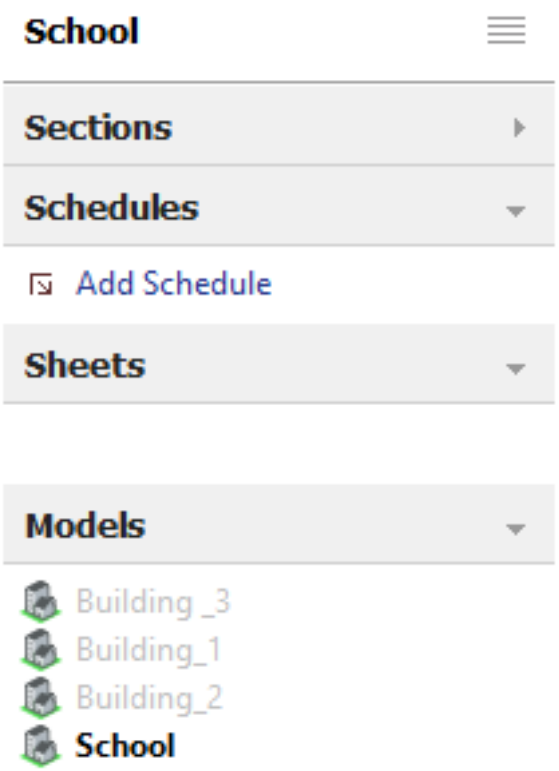


Figure 2.326



Editing a project

1. Click the **Menu** button (===ERROR WITH DOWNLOADED IMAGE===).
2. Choose **Project Setup...** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumenta

3. (option) Type a different name in the **Name** field.
4. (option) Click the checkboxes to add or remove model drawings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/

5. Click the **OK** button.
6. (option) Right click a model file, then choose **Remove from Project** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/ProjectBrowserModelContextRemove.png

Figure 2.327

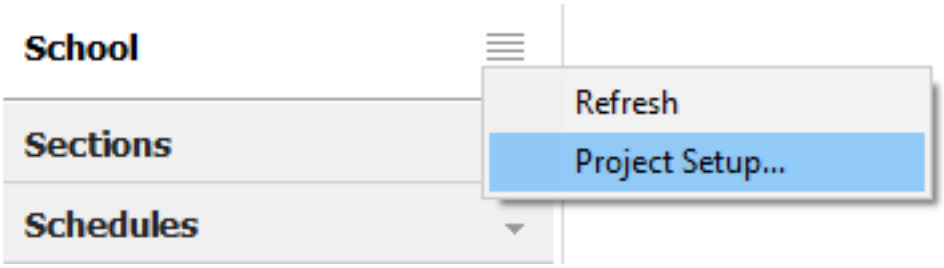


Figure 2.328

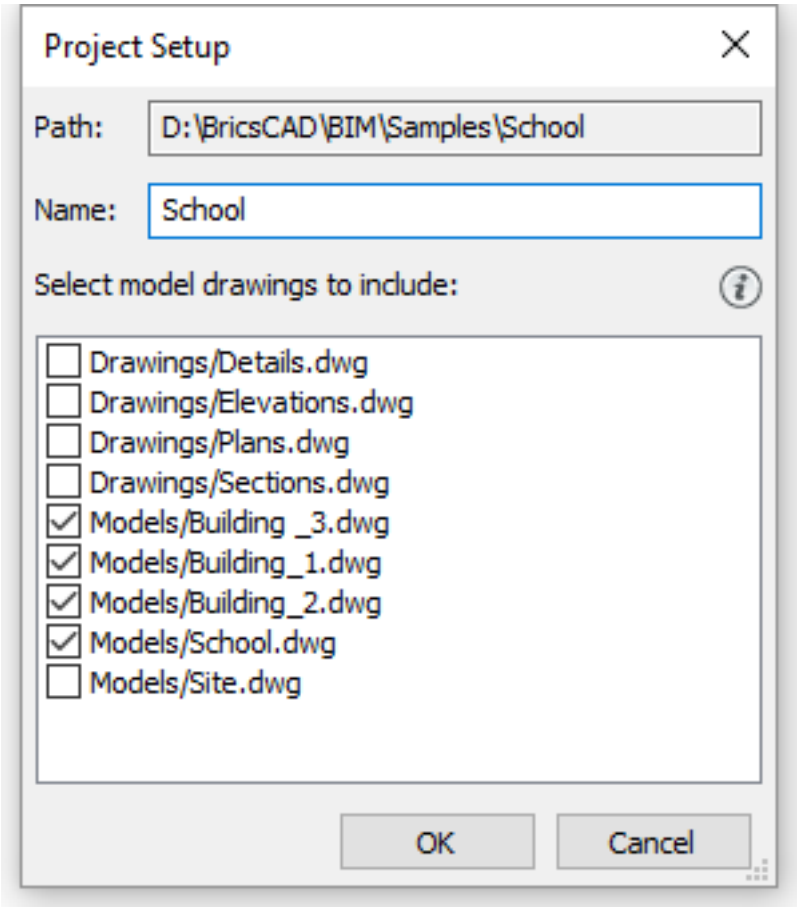
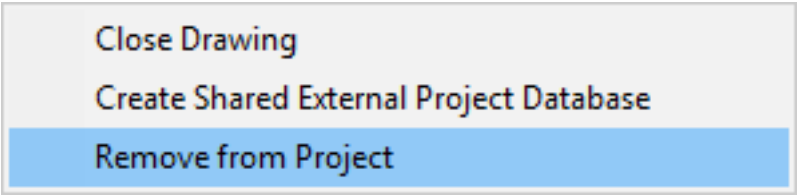


Figure 2.329



Creating Schedules

The **Add Schedule** item in the **Schedules** section of the project explorer uses a .dxd (data extraction definition) file to export data from a project model file in a Table (6.29.1) entity, which is saved in a project drawing.

1. Click the **Add Schedule** item in the **Schedules** section. The Create Schedule dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/CreateScheduleEmpty.pr
2. Type a name in the **Name** field.
3. Click the **Model** field, then click the **Browse** ( ) button. A **File** dialog displays.
4. Select a model file in the project folder, then double click or click the **Open** button.
5. Click the **Data Extraction File** field,then click the **Browse** ( ) button. A **File** dialog displays. By default the **.../Support/BIM/Schedules** folder is selected.
6. Select a **.dxd** file, then double click or click the **Open** button.
7. Select the **Result Drawing** field,then click the **Browse** ( ) button. A **File** dialog displays.
8. Select a drawing file in theproject folder, then double click or click the **Open** button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/CreateScheduleFull.png})
9. Click the **OK** button to create the schedule definition.
10. Select the schedule in the project browser, then right click and choose **Update** in the context menu to create the schedule in the **Result Drawing** file.
11. (option) Select the schedulein the project browser, then right click and choose **Display Schedule**in the context menu.

Figure 2.330

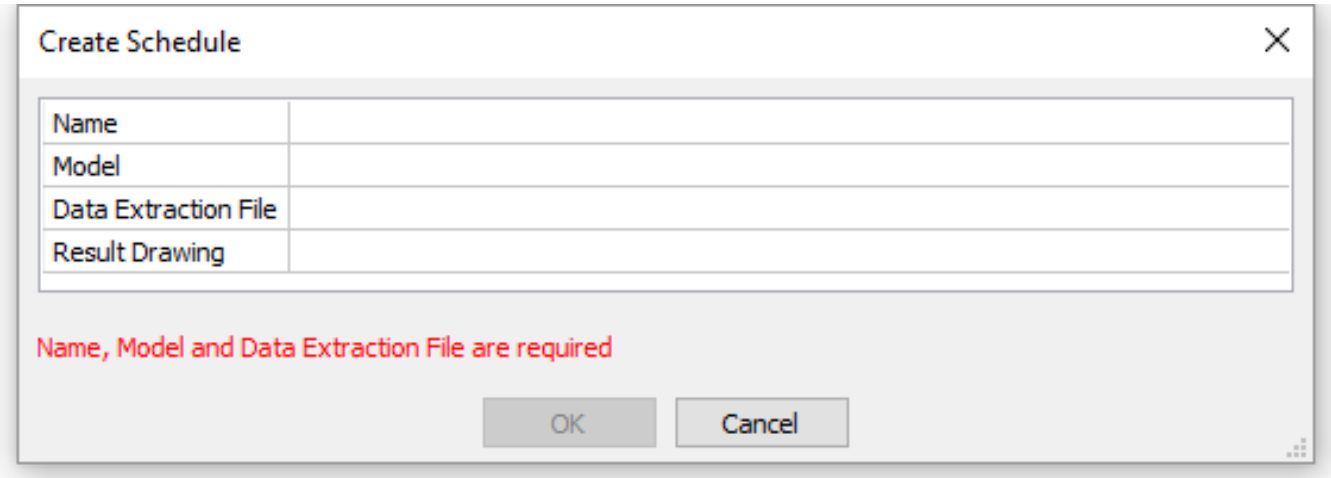
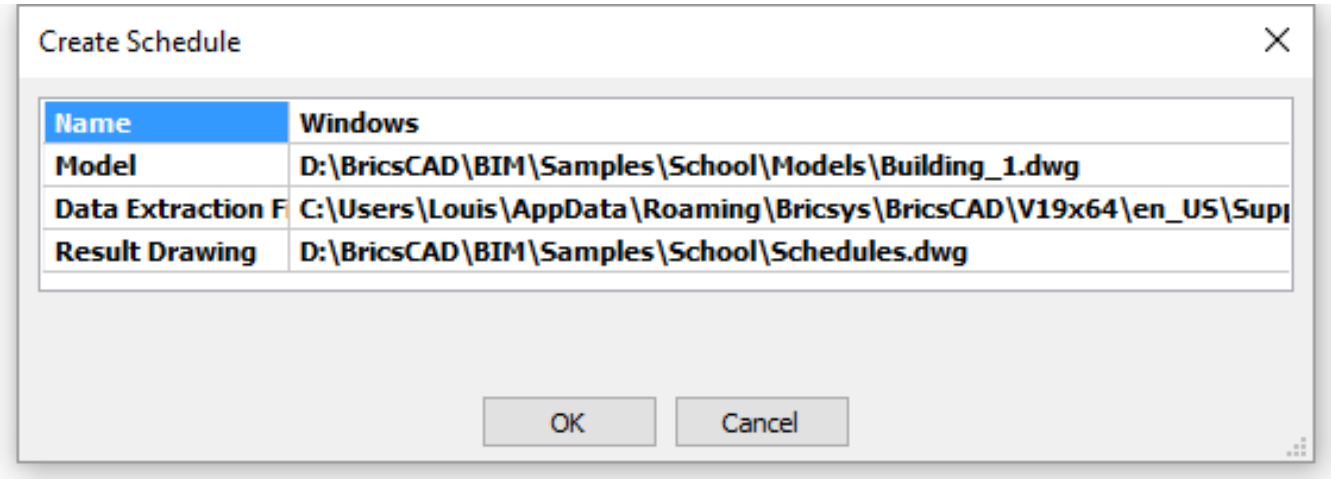


Figure 2.331



Working in the Project Browser

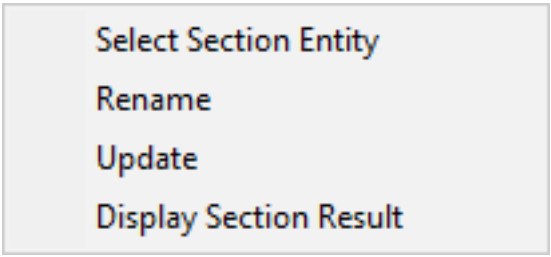
When you right click an item in the BIM Project Browser, a context menu displays.

Sections (\vref{fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/ProjectBrows

- **Select Section Entity**: Selects the section entity in the project model drawing.
- **Rename**: Renames the section entity.
- **Update**: Recalculates the section entity.
- **Display Section Result**: Displays the section result in the layout of a project drawing*.

*Opens the drawing when the drawing is not open yet.

Figure 2.332



Schedules: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/ProjectBro

- **Rename**: Renames the schedule
- **Update**: Recalculates the schedule content when the source data in the model file have been modified.
- **Display Schedule**: Displays the schedule in the result drawing*.
- **Remove...**: Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignD
  Select the items you want to remove:
  - **Schedule Entity**: removes the schedule definition in the project browser.
  - **Result Block**: deletes the table entity in the model space of the result drawing.
  - **Sheet View**: deletes the sheet view in the project sheet set (6.27.31).
  - **Named View**: deletes the named view (1.16.10) in the result drawing.

*Opens the drawing when the drawing is not open yet.

Figure 2.333

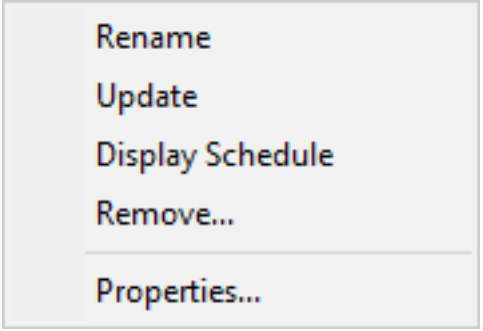
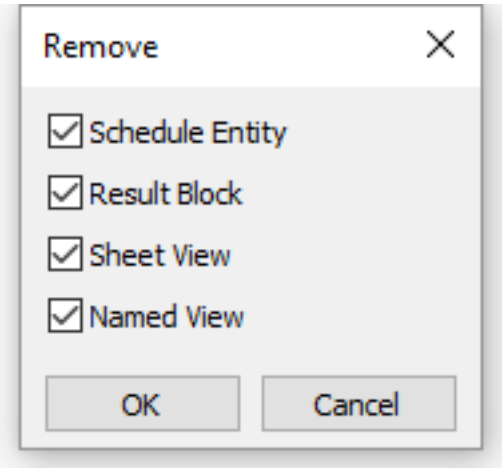


Figure 2.334



Sheets (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/ProjectBrowser

- **Open Sheet**: Displays the corresponding layout the project drawing*.
- **Rename**: Renames the layout name in the paperspace of the project drawing.
- **Remove...**: Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignD

*Opens the drawing when the drawing is not open yet.

Figure 2.335

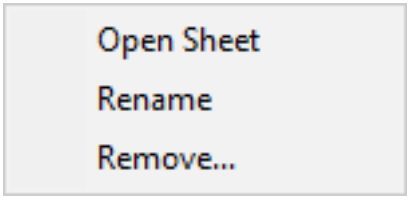
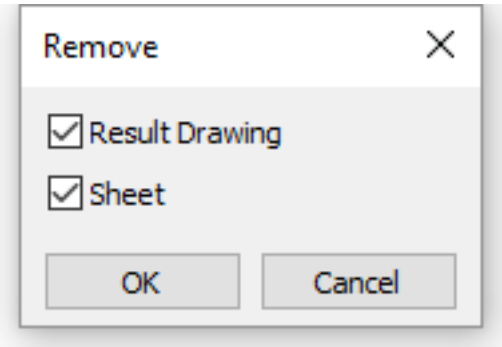


Figure 2.336



Sheet Views (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/ProjectBr

- **Display view**: Displays the layout in the project drawing* and zooms in on the corresponding viewport.
- **Rename**: Renames the sheet view title.
- **Update**: Updates the sheet view = executes the BimSectionUpdate (6.7.29) command for the related BIM section.
- **Remove...**: Displays a dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignD

  - **Result Block**: deletes the section block in the model space of the result drawing.
  - **Sheet View**: deletes the sheet view in the project sheet set (6.27.31).
  - **Named View**: deletes the named view (1.16.10) in the result drawing.

*Opens the drawing when the drawing is not open yet.

Figure 2.337

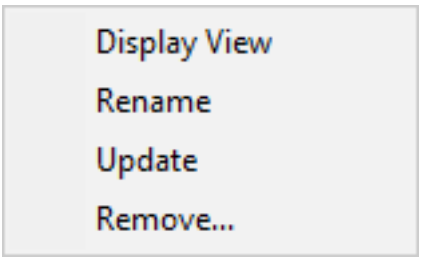
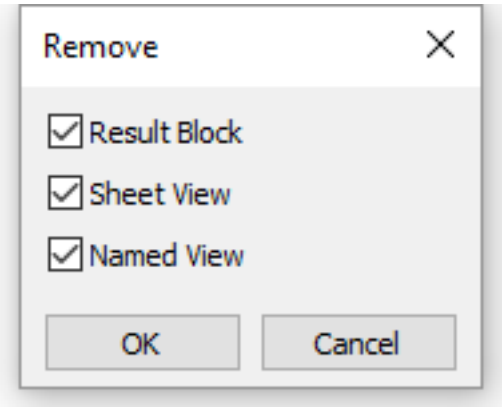


Figure 2.338



Model drawings Context menu of a currently open model: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS
Context menu of a currently closed model: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Desig

- **Close/Open Drawing**: Closes or opens the model.
- **Create Shared External Project Database**: Creates an external project database (see Bim Project Info (2.6.5)).
- **Remove from Project**: Removes the drawing from the project model drawings.

Figure 2.339

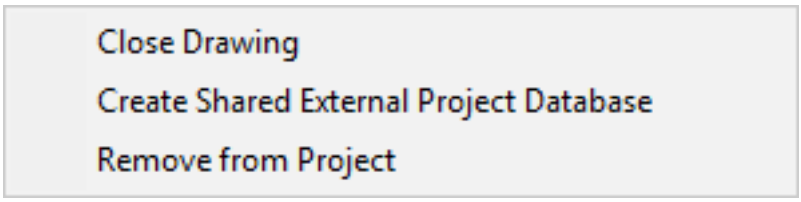
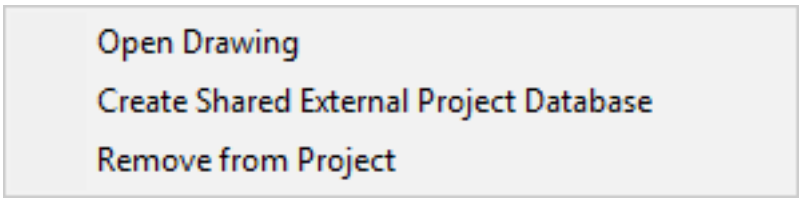


Figure 2.340


Open Drawing
Create Shared External Project Database
Remove from Project

2.6.10 Physical Materials

Command :BLMATERIALS (6.7.39) The **Physical Materials** dialog box allows to:

- Create new materials
- Edit materials
- Copy materials between the Project and Library databases
- Create single-ply compositions (2.5.1)
- Delete materials

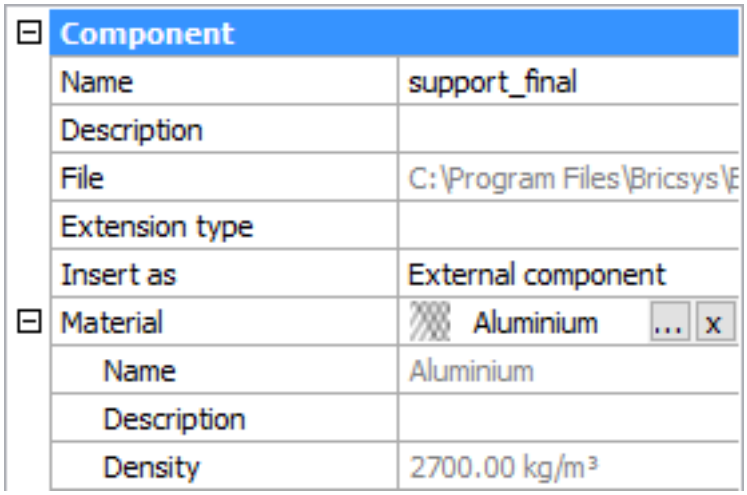
Tutorial: How to add and create Compositions Click here (2.5.1) to watch

To open the Physical Materials editor

Do one of the following:

- Type **blmaterials** in the command bar.
- Click the **Physical Materials** button (===ERROR WITH DOWNLOADED IMAGE===) on the Compositions panel (2.5.1).
- Click the **Browse** button (...) of the **Material** property of a component in the Mechanical Browser (3.2.2). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/ComponentMaterial.png})

Figure 2.341



Adding physical materials

Physical materials can be created in either the Project or the Library database. New material can be created from scratch or as a copy of existing material.

Adding physical material from scratch Click on either the **In Project** or the **In Library** material list. Do one of the following:

1. Click the **Add Material** button () on the **Physical Materials** dialog.
2. Right-click and choose **Add Material** in the context menu.

A material named **New** is added in the selected database.

1. Edit the newly created material.

Adding a physical material as a copy of an existing material Select a material in either the **In Project** or the **In Library** material list. Do one of the following:

1. Click the **Add Material** button () on the **Physical Materials** dialog.
2. Right-click and choose **Add Material** in the context menu.
3. A material named **<existing_material>_1** is added in selected database.

Edit the newly created material.

Editing a physical material

The various settings of a physical material definition are divided into three settings groups under three tabs on the Physical Materials (2.6.10) dialog:

- Identity: Name, Class and description of the material.
- Appearance: The BIM Materials layers the composition is composed of.
- Properties: Properties of building material such as **Cost,Manufacturer,Thickness,**...

Select the material in either the **In Project** or the **In Library** list on the **Physical Materials** dialog.

1. On the **Identity** tab page: Type a name for the material and select a **Class** from the drop-down list. (option) Type a description.
2. On the **Appearance** tab page:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMMaterialsEditAppearance.pn
Section settings:

- Click the **Hatch Type** button, then select the type of the hatch: **None,Solid,Linear,Predefined** or **Custom.**
- Do one of the following to select a predefined or custom hatch pattern
  - Click the **Hatch** preview, then select the pattern in the **Hatch Pattern** dialog box.
  - Click the **Hatch Pattern Name** button, then select the pattern in the list.
- Define the **Scale.** The final pattern scale in a section is the product of the **Scale** setting of the physical material and the value of the Hatch Scale property of the section plane entity. (2.12.1)
- Set the **Rotation** angle.
- In the **Layer** field, either type a name or click the arrow button and select a layer in the list.

3D settings: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMMaterialsEditIden

- Double click the preview image, then select material in the **Render Material** dialog box.

On the **Properties** tab page: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMMa

- Edit the **Tags** field: tags are separated by commas. Tags are case sensitive. Click the **Browse** button () to open the **Tags** dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BIMTa

Figure 2.342

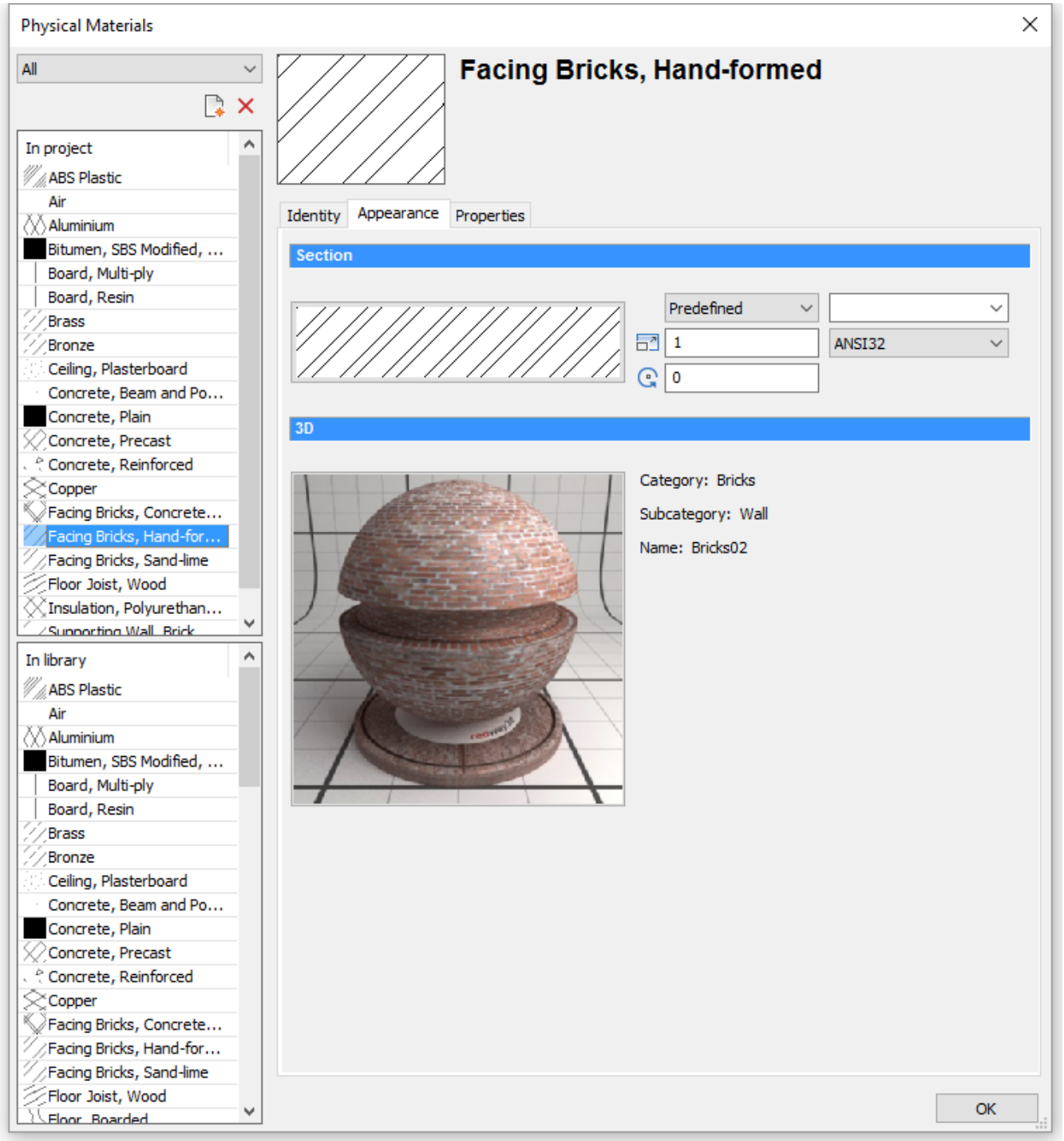

Physical Materials
All
In project
ABS Plastic
Air
Aluminium
Bitumen, SBS Modified, ...
Board, Multi-ply
Board, Resin
Brass
Bronze
Ceiling, Plasterboard
Concrete, Beam and Po...
Concrete, Plain
Concrete, Precast
Concrete, Reinforced
Copper
Facing Bricks, Concrete...
Facing Bricks, Hand-for...
Facing Bricks, Sand-lime
Floor Joist, Wood
Insulation, Polyurethan...
In library
ABS Plastic
Air
Aluminium
Bitumen, SBS Modified, ...
Board, Multi-ply
Board, Resin
Brass
Bronze
Ceiling, Plasterboard
Concrete, Beam and Po...
Concrete, Plain
Concrete, Precast
Concrete, Reinforced
Copper
Facing Bricks, Concrete...
Facing Bricks, Hand-for...
Facing Bricks, Sand-lime
Floor Joist, Wood
Facing Bricks, Hand-formed
Identity
Appearance
Properties
Section
Predefined
1
0
ANSI32
3D
Category: Bricks
Subcategory: Wall
Name: Bricks02
OK

Figure 2.343

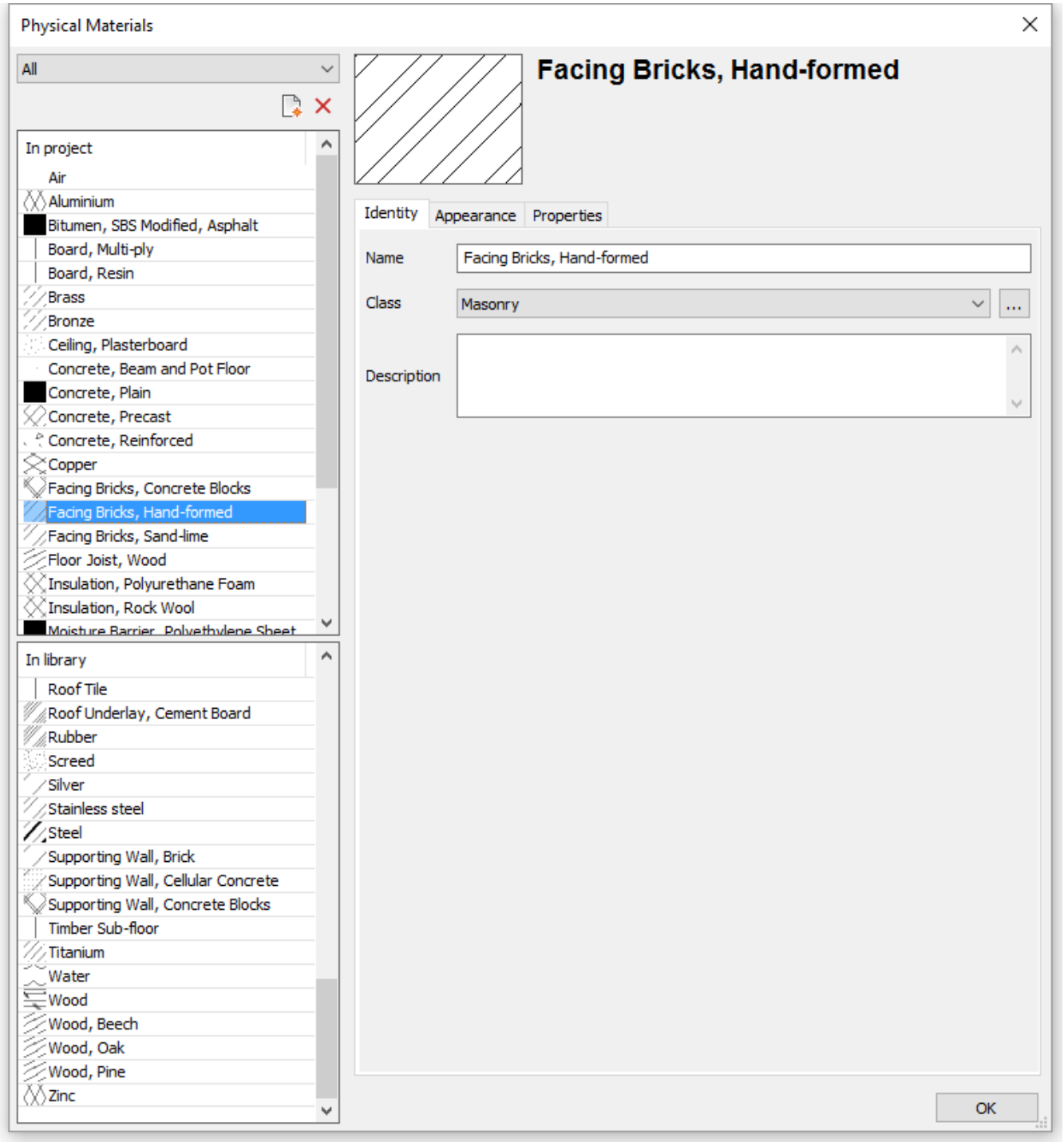

Physical Materials
All
In project
Air
Aluminium
Bitumen, SBS Modified, Asphalt
Board, Multi-ply
Board, Resin
Brass
Bronze
Ceiling, Plasterboard
Concrete, Beam and Pot Floor
Concrete, Plain
Concrete, Precast
Concrete, Reinforced
Copper
Facing Bricks, Concrete Blocks
Facing Bricks, Hand-formed
Facing Bricks, Sand-lime
Floor Joist, Wood
Insulation, Polyurethane Foam
Insulation, Rock Wool
In library
Roof Tile
Roof Underlay, Cement Board
Rubber
Screed
Silver
Stainless steel
Steel
Supporting Wall, Brick
Supporting Wall, Cellular Concrete
Supporting Wall, Concrete Blocks
Timber Sub-floor
Titanium
Water
Wood
Wood, Beech
Wood, Oak
Wood, Pine
Zinc
Facing Bricks, Hand-formed
Identity
Appearance
Properties
Name
Facing Bricks, Hand-formed
Class
Masonry
Description
OK

Figure 2.344

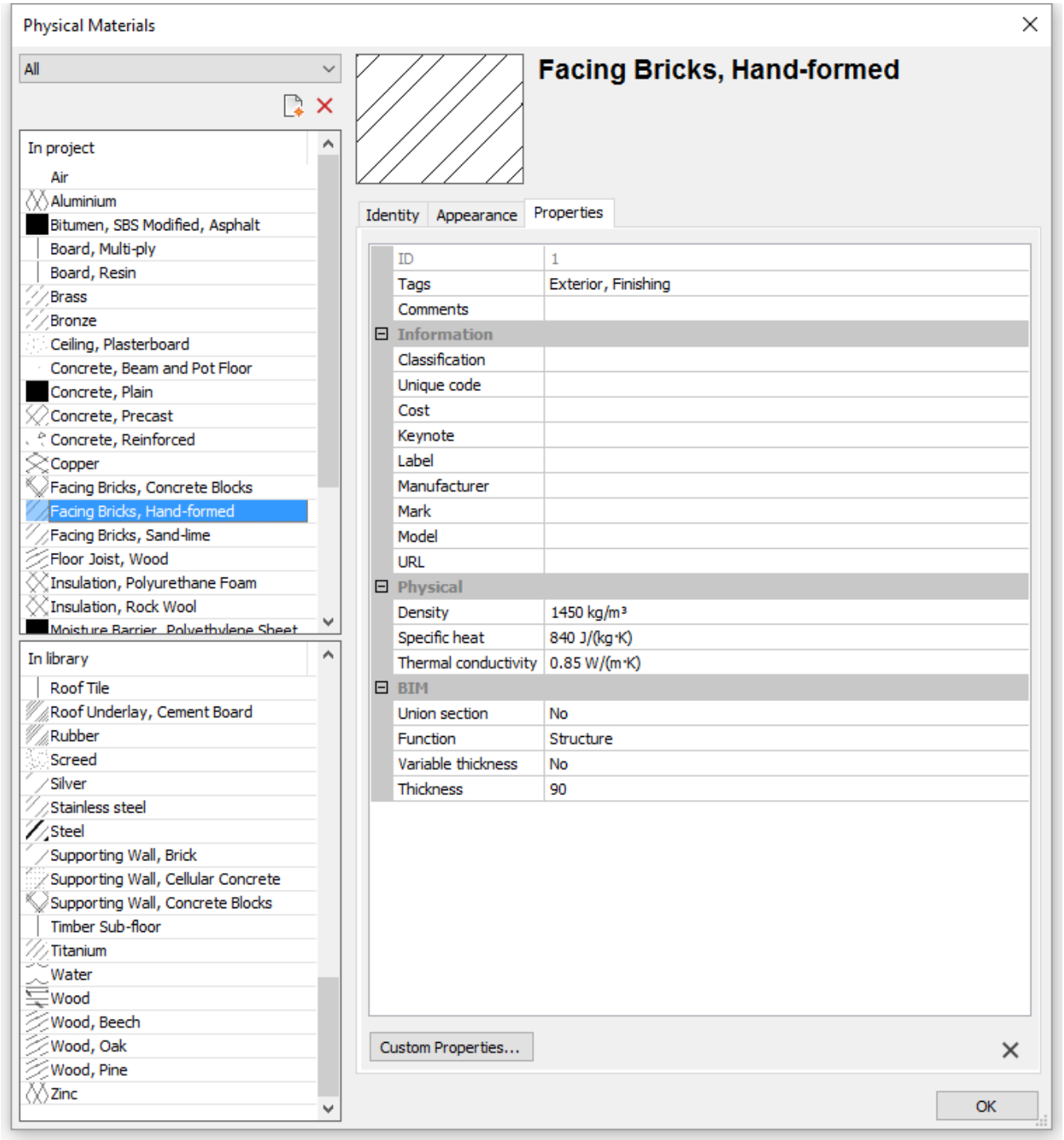
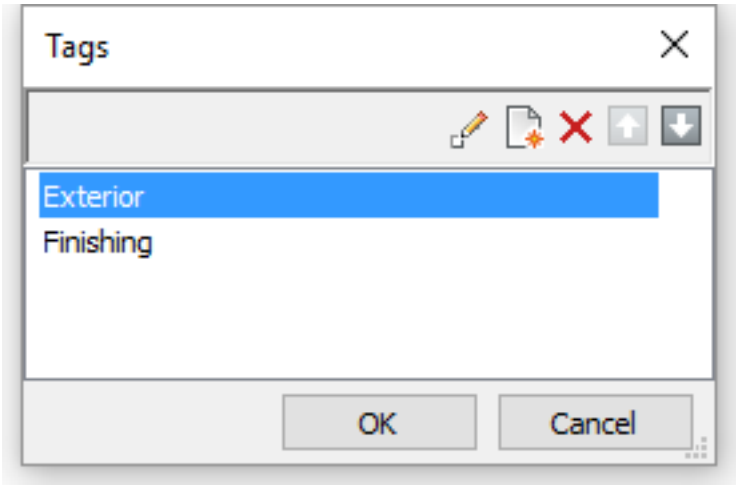

Physical Materials
All
In project
Air
Aluminium
Bitumen, SBS Modified, Asphalt
Board, Multi-ply
Board, Resin
Brass
Bronze
Ceiling, Plasterboard
Concrete, Beam and Pot Floor
Concrete, Plain
Concrete, Precast
Concrete, Reinforced
Copper
Facing Bricks, Concrete Blocks
Facing Bricks, Hand-formed
Facing Bricks, Sand-lime
Floor Joist, Wood
Insulation, Polyurethane Foam
Insulation, Rock Wool
In library
Roof Tile
Roof Underlay, Cement Board
Rubber
Screed
Silver
Stainless steel
Steel
Supporting Wall, Brick
Supporting Wall, Cellular Concrete
Supporting Wall, Concrete Blocks
Timber Sub-floor
Titanium
Water
Wood
Wood, Beech
Wood, Oak
Wood, Pine
Zinc
Facing Bricks, Hand-formed
Identity
Appearance
Properties
ID 1
Tags Exterior, Finishing
Comments
Information
Classification
Unique code
Cost
Keynote
Label
Manufacturer
Mark
Model
URL
Physical
Density 1450 kg/m³
Specific heat 840 J/(kg·K)
Thermal conductivity 0.85 W/(m·K)
BIM
Union section No
Function Structure
Variable thickness No
Thickness 90
Custom Properties...
OK

Figure 2.345



Click the **New Item** button ( ) to add a new tag at the bottom of the list. Select a tag in the list then click a tool button:

- : edit the selected tag
- : delete the tag
- : move the tag up
- : move the tag down
- **Optional**Expand the **Information** properties and fill out the necessary fields.
- **Optional** Expand the **Physical** properties and edit physical properties of the material.
- Expand the **BIM** properties.
- **Union Section:**Select **Yes** to remove connection lines in sections between different solids of this material. Choose **No** to keep the connection lines.
- Select a **Function** from the drop-down list.
- Choose **Yes** or **No** whether or not to allow a variable thickness.
- Type a default thickness in the **Thickness** field.

1. Click the **OK** button.

To copy materials between the Project and Library databases.

1. Select the material(s) in the source database. Press and hold the Ctrl-key to select multiple materials. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/BiMMaterialsSelection.png})

2. Drag the materials to the target database. The selection set is copied. If one or more materials already exist in the target database, an alert box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/E
Click the appropriate button to either overwrite, duplicate or skip the existing material. If the **Apply to all materials in this operation** option is checked, the option is applied to all existing materials. Duplicated materials have an index number added to their name: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/

Figure 2.346

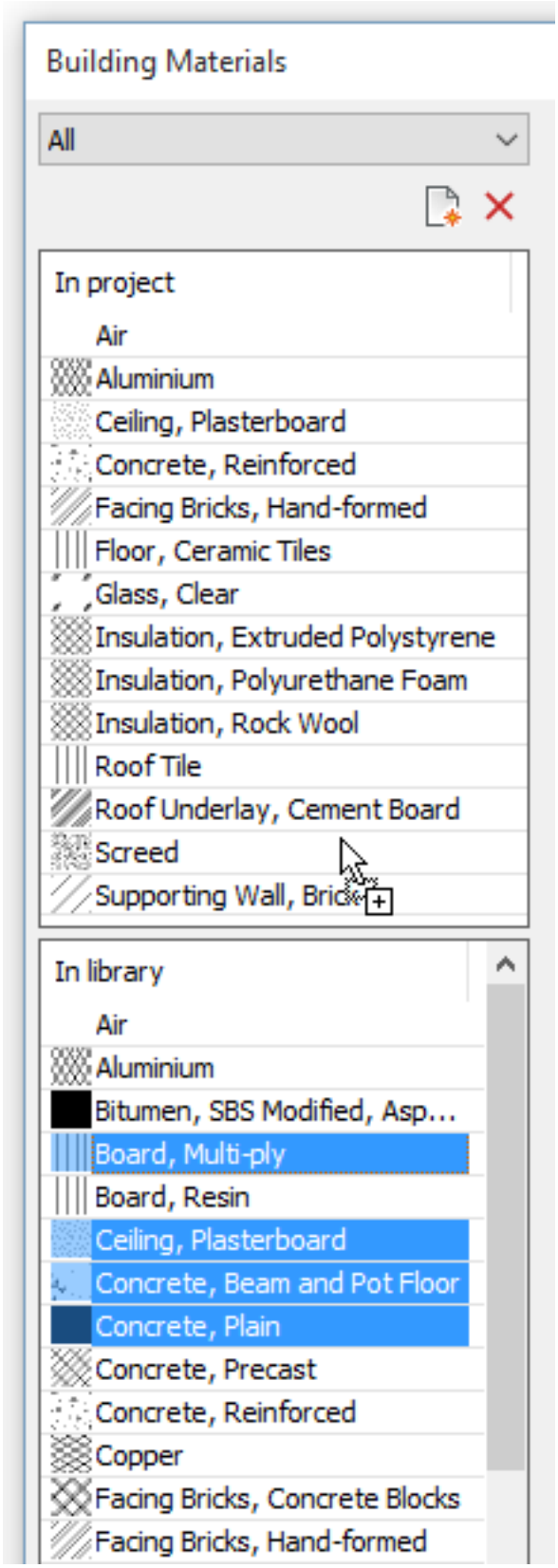

Building Materials
All
In project
Air
Aluminium
Ceiling, Plasterboard
Concrete, Reinforced
Facing Bricks, Hand-formed
Floor, Ceramic Tiles
Glass, Clear
Insulation, Extruded Polystyrene
Insulation, Polyurethane Foam
Insulation, Rock Wool
Roof Tile
Roof Underlay, Cement Board
Screed
Supporting Wall, Bric
In library
Air
Aluminium
Bitumen, SBS Modified, Asp...
Board, Multi-ply
Board, Resin
Ceiling, Plasterboard
Concrete, Beam and Pot Floor
Concrete, Plain
Concrete, Precast
Concrete, Reinforced
Copper
Facing Bricks, Concrete Blocks
Facing Bricks, Hand-formed

Figure 2.347

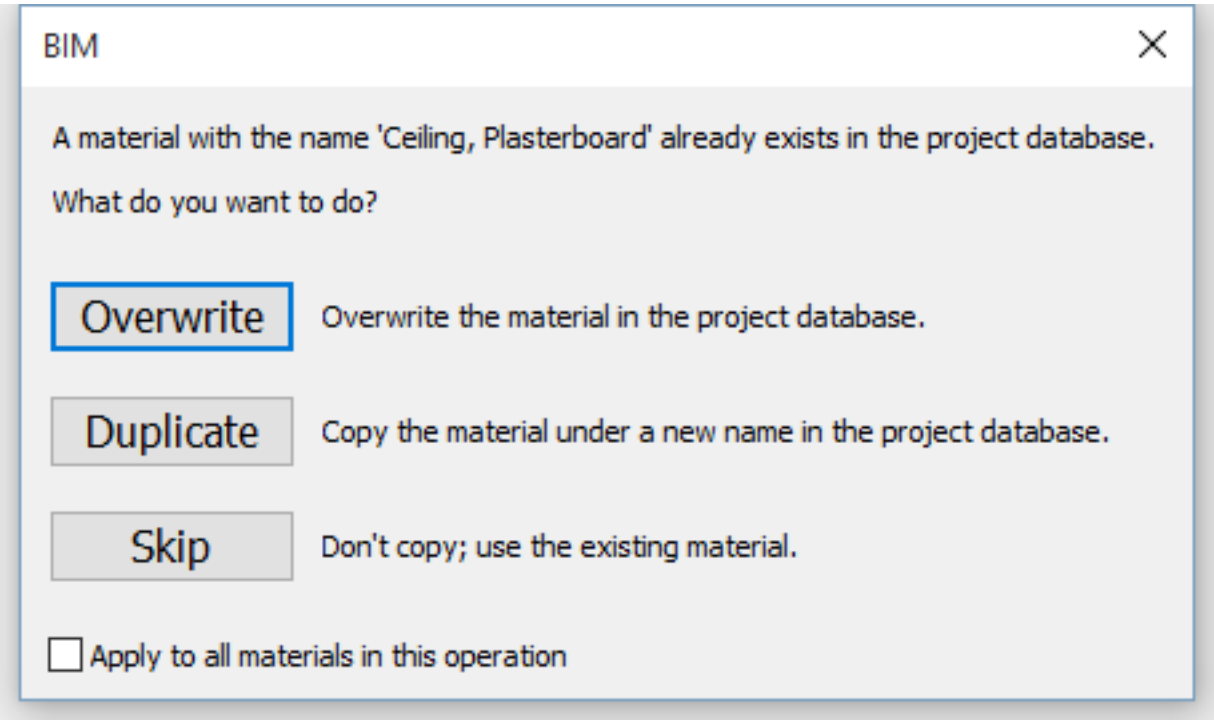
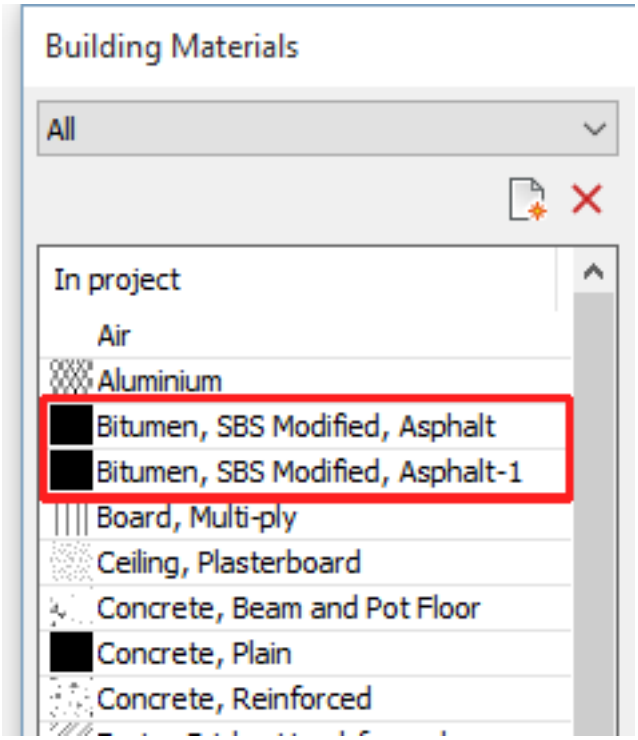


Figure 2.348



Deleting physical materials

1. On the **Physical Materials** dialog box, select the material(s) in the **Project** or **Library** database.
2. Click the **Delete** button.

Note	Materials that are used in a composition or a mechanical component cannot be deleted.

2.6.11 Spatial Locations

Command: BimAttachSpatialLocation (6.7.4) **Overview:** In this lesson, you learn how to create spatial locations and assign them to building elements. **Lesson Objectives** After completing this lesson, you will be able to:

- Create spatial locations from the Building & Story manager.
- Edit the site properties of your BIM model.
- Set up a building and story information about your building elements.

Tutorial: How to add Building & Story information Click here (2.5.2) to watch

About

The spatial locations allow you to specify a building and its stories in a project. In BricsCAD, a BIM model file can contain one site and multiple buildings and a building cancontainmultiple stories. When the entities are classified as a building element in the project, these classified entities will have a **Building** and a **Story** property. Every building element in the BIM model resides in a particular story within a particular building.  Menubar: **BIM > Spatial Locations** Toolbar: **BIM > Spatial Locations** Quad: **BIM > Spatial Locations** Keyboard: **BIMSPATIALLOCATIONS**

Assigning a building and story to a building element

1. Open the Building & Story manager using either the command line or Quad. The dialog box displays (1). Do the following to define the building and story:

   - Click the new building icon (2) to add a new building to the model. Fill out the properties grid (3) to specify a new name or value of the existing property.

   - Click the new story icon (4) to add a new story in the selected building. Fill out the properties grid to specify a new name or value of the existing property.

   Optional:click the delete icon (5) to remove the selected building or story. (\vref {fig:C:/Dev/Task-2448/images/staticbrics

2. Select building element(s) in the drawing area to associate them with the appropriate spatial locations. The properties of the selected building elements display in the Properties Bar. Do the following to assign the building and story to these building elements:

   - Expand the BIM section (6) in the properties panel
   - Click the Building field (7), then use the arrow (8) to select a building
   - Click the Story field (9), then use the arrow to select a story.

   The building elements are associated with the selected building and story. (\vref {fig:C:/Dev/Task-2448/images/staticbric

Figure 2.349

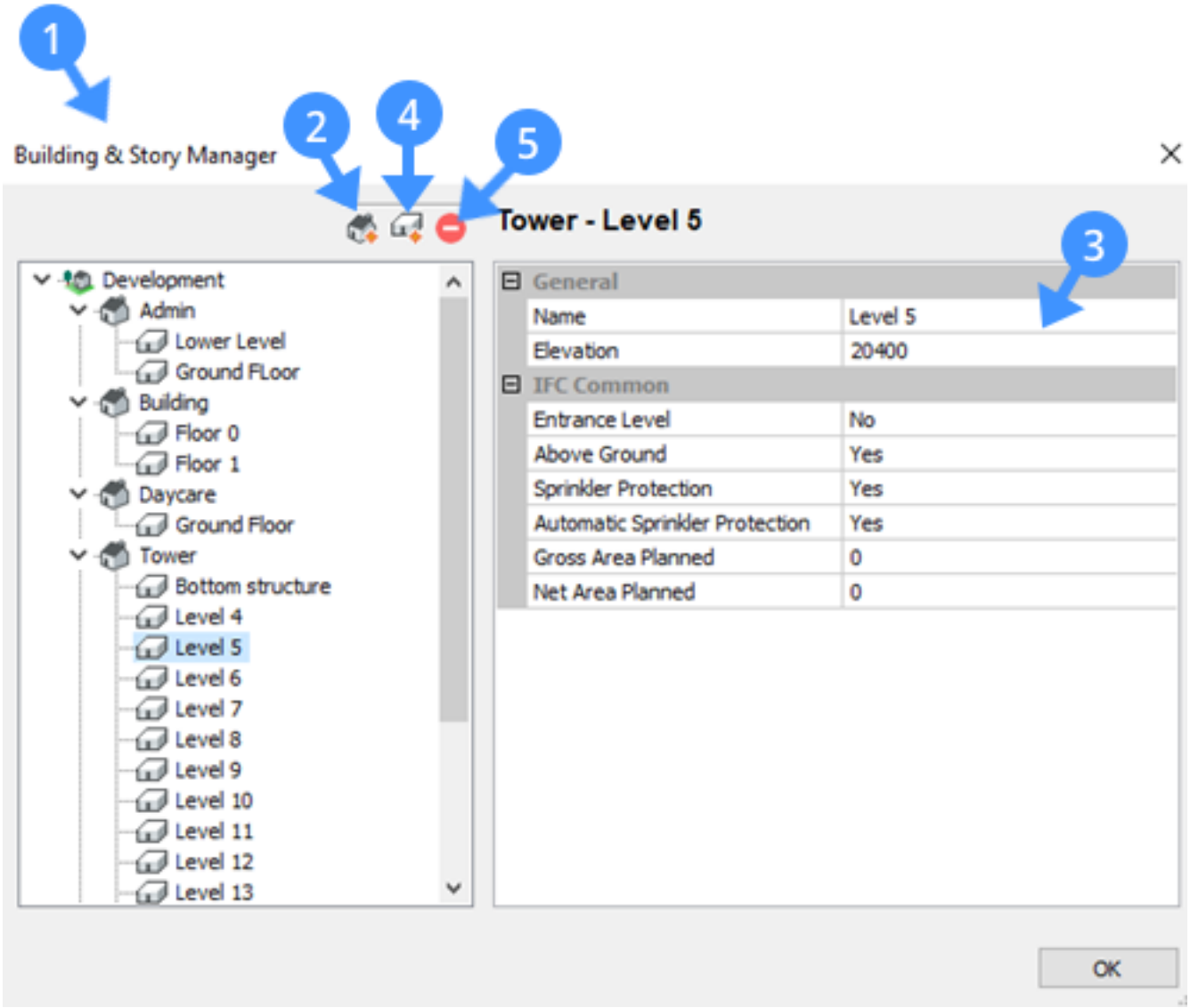
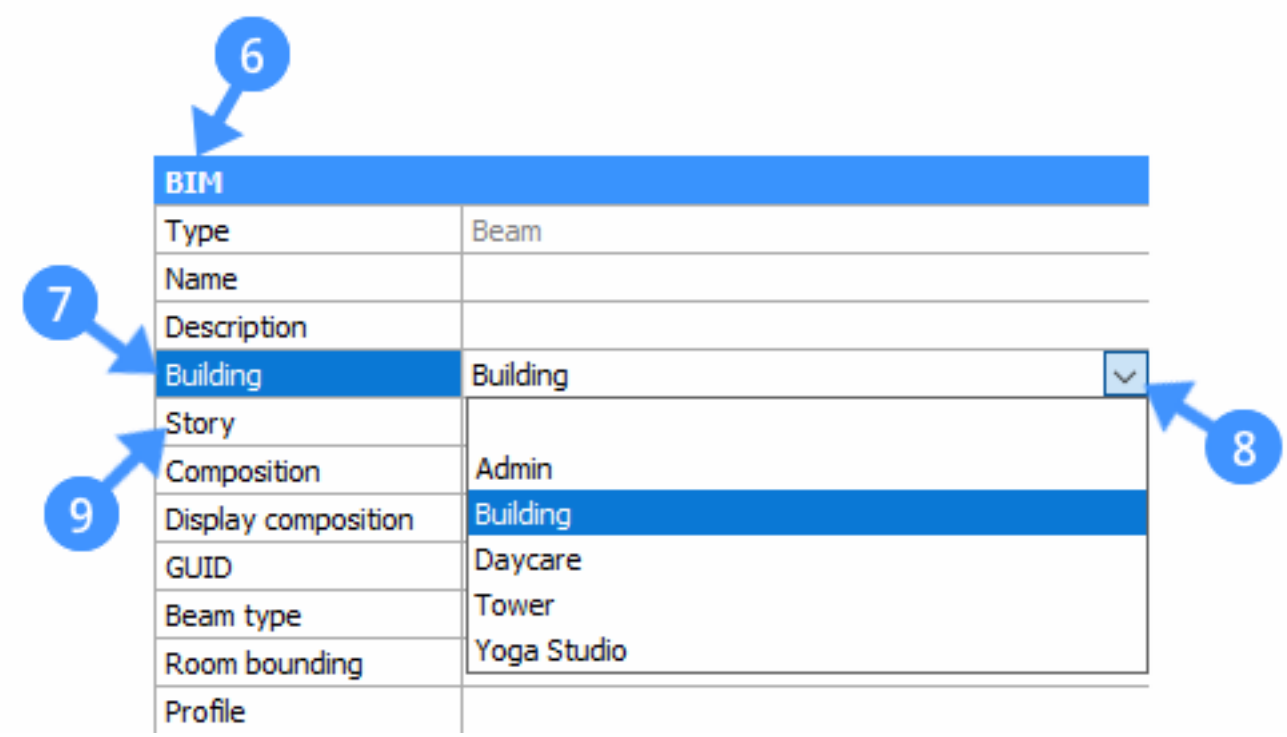

1
2
4
5
3
Building & Story Manager
Tower - Level 5
Development
Admin
Lower Level
Ground FLoor
Building
Floor 0
Floor 1
Daycare
Ground Floor
Tower
Bottom structure
Level 4
Level 5
Level 6
Level 7
Level 8
Level 9
Level 10
Level 11
Level 12
Level 13
General
Name
Level 5
Elevation
20400
IFC Common
Entrance Level
No
Above Ground
Yes
Sprinkler Protection
Yes
Automatic Sprinkler Protection
Yes
Gross Area Planned
0
Net Area Planned
0
OK

Figure 2.350



Viewing the building elements in the Structure browser

The Structure browser allows to view all of the entities in a drawing. By default, the structure of the building elements is organized in a particular form. Building elements are grouped by Building, Story, BIM type, and by composition respectively. This organization can be fully configured.

2.6.12 Structure Browser

Command: StructurePanel (6.27.50) **Overview:** In this lesson, you learn how to organize your entities in the structure tree, then view them on the Structure Browser. **Lesson Objectives** After completing this lesson, you will be able to:

- Configure the structure tree.
- Create a new configuration file.
- Filter, group and sort entities in the Structure Browser.

Tutorial: How to use Structure Browser Click here (2.5.7) to watch

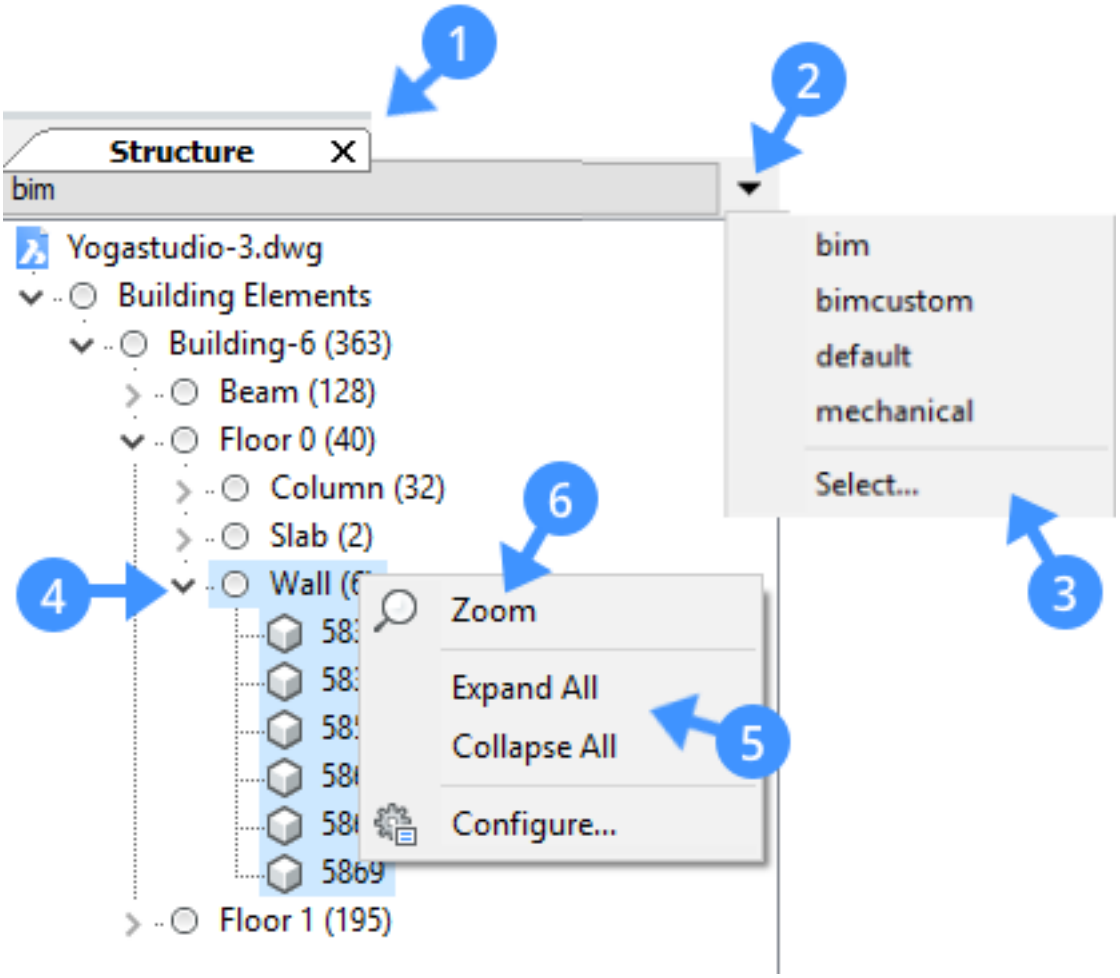
About

The structure browser is an interactive tree that displays the entities in the current model. Using the configurable structure tree on the Structure Browser the BIM model can be organized in a way that you want to view the elements. Once you configured the tree, you can easily save this structured tree as a .cst file. By default, the .cst files are stored in the **Support Folder**.  Place the cursor over a toolbar or ribbon panel, then right-click and choose **Structure** in the context menu. Or type **structurepanel** in the command bar, then press Enter.

Procedure: Using the structure browser

1. Open the structure browser. By default, the structure browser (1) displays on the left-hand side on your screen.
2. Click the down-arrow button (2) to load a configuration file in the list. The current list displays (3) on the left. Based on the selected file the BIM model will be organized in a tree-structure.
3. Click the expand/ collapse icon (4) of a node to expand or collapse the node. You can also expand or collapse all branches at once: right-click a node, then choose expand all or collapse all in the context menu (5). This context menu also provides you to zoom (6) into a specific entity in the model. The following illustration shows the Structure Browser. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/Struct

Figure 2.351


1
2
Structure
bim
Yogastudio-3.dwg
Building Elements
Building-6 (363)
Beam (128)
Floor 0 (40)
Column (32)
Slab (2)
Wall (6
58
58
58
58
58
5869
Floor 1 (195)
bim
bimcustom
default
mechanical
Select...
3
4
6
Zoom
Expand All
Collapse All
5
Configure...

Procedure: Managing the tree-structure

The tree structure of the Structure browser represents the hierarchical set of rules in a graphical form. Each rule has a filter property, a grouping property, and a sorting property. Organizing the properties in the Structure Browser enhances the performance of your project while querying the entities among others. When you have a tree for your BIM model you can establish a hierarchy across its building element types, buildings, stories, etc. You can use the hierarchy to manage the properties at various levels of the data display. Do the following to configure the tree-structure:

1. Right-click on the name of your drawing in the Structure Browser, then choose Configure in the context menu. The **Configure Structure Tree** (7) dialog box displays.

2. Select any existing rule in the structure tree, then click the **Add icon** (8) or right-click and choose Add Rule in the displayed context menu. This menu can also be used to delete and rename the existing rule. The new rule is added below the previously selected rule. Type a new name for the newly created rule by clicking its name field (9).

3. Select the **Group** (10) node of the rule, then use the Add icon or right-click and choose **Add Grouping Property** (11) in the context menu. The **Select Property** dialog box displays. Select a property on the list that you want to use to group the properties in your Rule setting. Then, click the OK button. The new property is added to the Group node. If you use the Filter node to add a property, the selected property will be added in this node.

4. To move an item in the tree, select the item, then click the **Move Up** (12) or **Move Down** (13)

5. Select the **Show/Skip** (14) tab to sort the entity types or exclude them on the Structure Browser.

6. Select the **Options** (15) tab to set the entities action in the model using the structure tree.

7. Do one of the following to save the configured tree:

   - Click the **OK** (16) button.
   - Click the **File** (17) menu in the dialog box and choose one of the available options on the list (18).

   The following illustration shows the Configure structure tree dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsy

Open (19)	Opens a file dialog to select another .cst configuration file.
Save (20)	Saves the current configuration and keep the file dialog open.
Save as (21)	Saves the current configuration under a different name and keep the file dialog open.

The following illustration shows the Select Property dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h

General (22)	Contains general properties.
Entities (23)	Contains various types of entity properties.
Extensions (24)	Contains various types of properties for BIM and Quantity extensions.

Figure 2.352

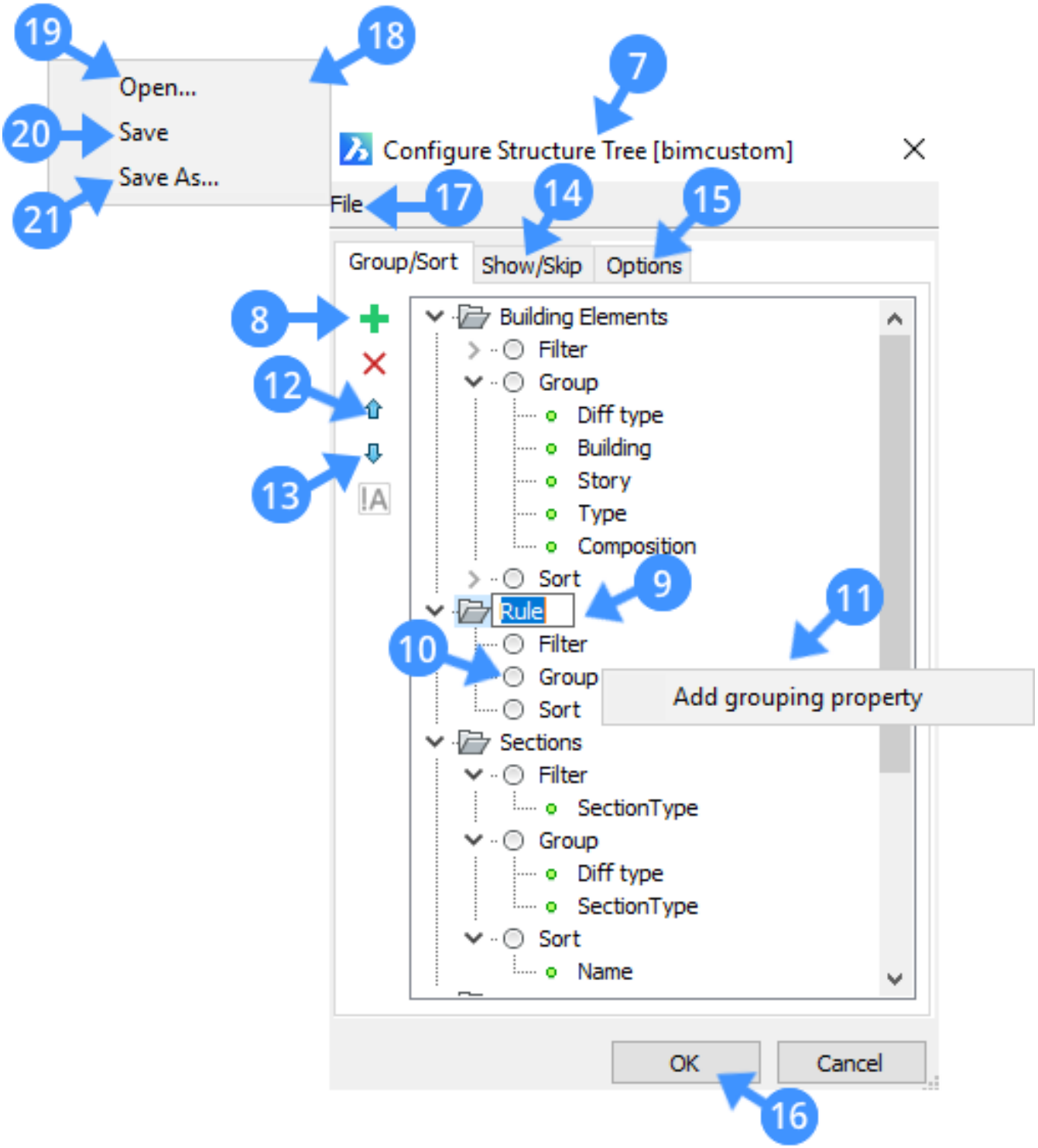
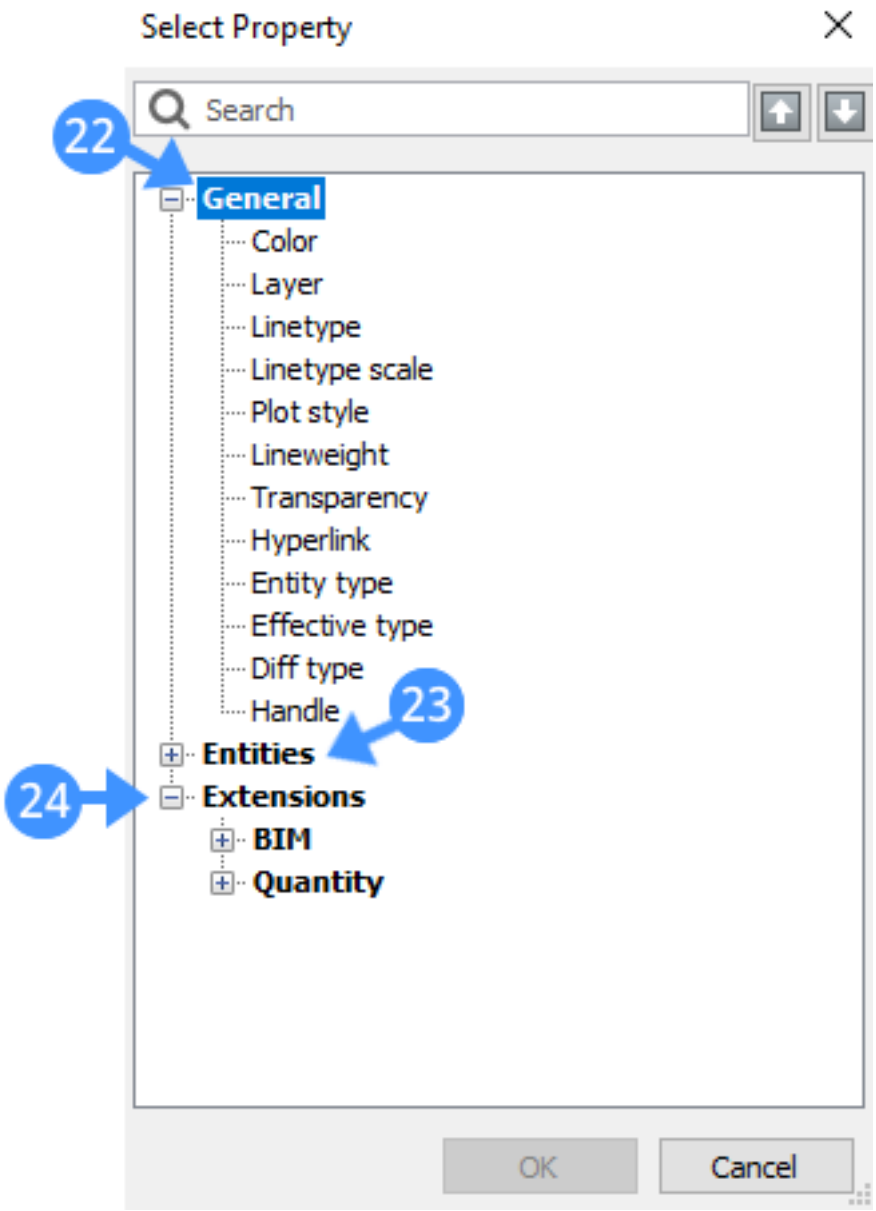

19
18
Open...
20
Save
Save As...
21
7
Configure Structure Tree [bimcustom]
File
17
14
15
Group/Sort
Show/Skip
Options
8
12
13
IA
Building Elements
Filter
Group
Diff type
Building
Story
Type
Composition
Sort
9
Rule
11
10
Filter
Group
Sort
Add grouping property
Sections
Filter
SectionType
Group
Diff type
SectionType
Sort
Name
OK
Cancel
16

Figure 2.353


Select Property
Search
22
General
Color
Layer
Linetype
Linetype scale
Plot style
Lineweight
Transparency
Hyperlink
Entity type
Effective type
Diff type
Handle
23
Entities
24
Extensions
BIM
Quantity
OK
Cancel

Procedure: Managing the tree-structure with custom properties

In BricsCAD, the user-defined/custom properties are available to add in the Select Property dialog using the GUI. The select property dialog provides a list of dynamic properties for the drawing based on the current workspace (1.13.10). Note that custom properties (2.6.8), through the select property dialog box, can be used to group the tree structure. Do the following to group the tree-structure using the existing custom properties:

1. Right-click on the name of your drawing in the Structure Browser, then choose Configure (25) in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/StructureBrowserCon
The **Configure Structure Tree dialog**box displays.

2. Select the **Group** node of the rule, and click the **Add rule or property** ( )icon. **Or** right-click on the Group node and choose **Add Grouping Property**in the context menu. The Select Property dialog box displays (see below image for the select property dialog). If you have custom properties, these properties will be shown under **Extensions** (26)and grouped by its namespace.

3. Click the expand/collapse icon of a node to expand and collapse the node. The following image shows the IFC 2X3 (27) andUser (28) namespaces in the Select Property dialog. For example, the User namespace is expanded to display the property set (29) and properties (30) that are in this node. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/SelectPropertyDialogBoxNamespace.png})

5. Select the property to use for grouping in the tree structure, and press OK. The property is added (31) to the group node.

6. Press OK (32) to use the propertyfor grouping entities in the Structure Browser. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/BuildingData/ConfigureStructureTreeDialogBoxUserproperty.png}

Figure 2.354

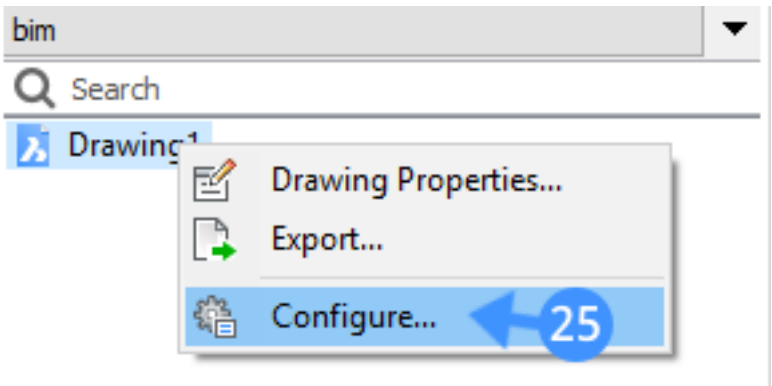


Figure 2.355

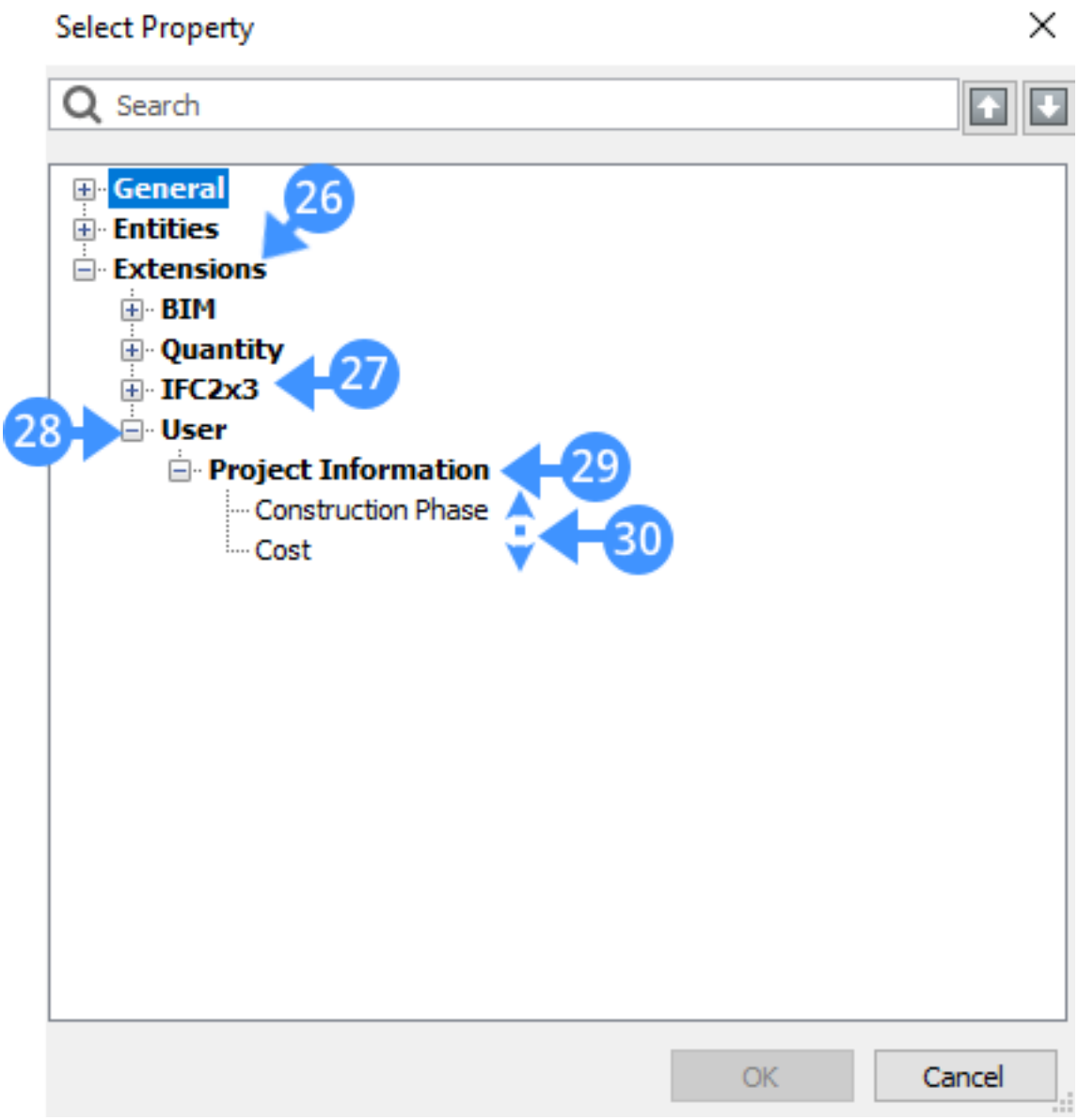
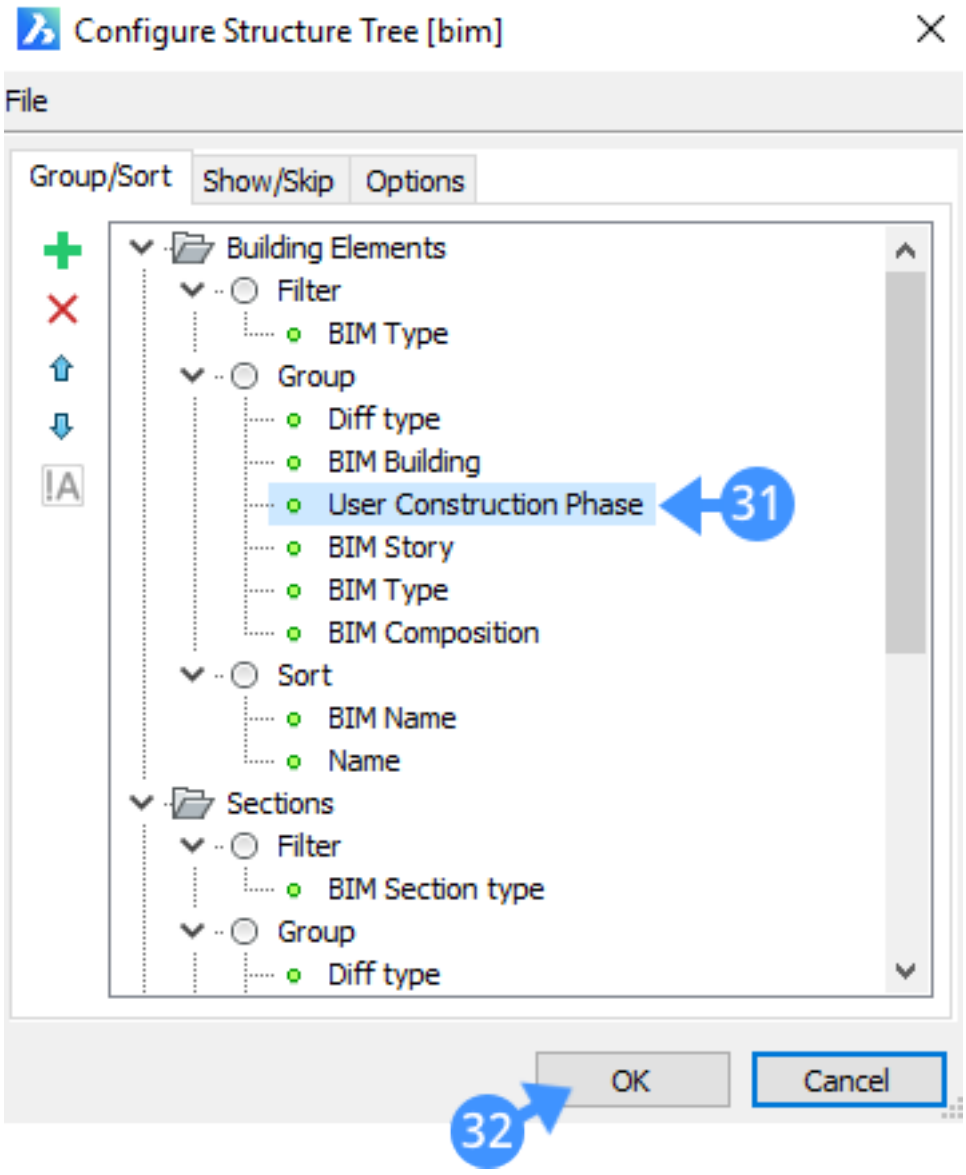

Select Property
Search
General
Entities
Extensions
BIM
Quantity
IFC2x3
User
Project Information
Construction Phase
Cost
26
27
28
29
30
OK
Cancel

Figure 2.356


Configure Structure Tree [bim]
File
Group/Sort
Show/Skip
Options
Building Elements
Filter
BIM Type
Group
Diff type
BIM Building
User Construction Phase
BIM Story
BIM Type
BIM Composition
Sort
BIM Name
Name
Sections
Filter
BIM Section type
Group
Diff type
31
OK
Cancel
32

2.7 Design Assistance

2.7.1 Advanced Modeling with Propagate

In this video series we will create a detailed model of a pitched roof structure, making extensive use of the Propagate tool and other direct modelling tools. Youtube We start off with the rough shape of our building; materials have already been applied to the different building elements. We'll turn on 'Display Composition' to be able to 'push-pull' the different composition plies. Using 'Connect with nearest' we can easily connect these plies to each other. We will create a detail locally, and then later Propagate this detail over the entire perimeter of our roof. This detail includes a gutter, some coverings, a concrete ring beam and a wooden beam. We will use 'Propagate Planar' to copy this detail. However, to make sure Propagate knows which detail to copy exactly, we will need to activate a section that cuts through all involved detail objects. For this reason, we will move this section plane slightly further into the model, to make sure it includes everything we need. We are prompted to the select the base solids. These are the solids that define the situation where this particular detail should be applied. In this case it's the floor slab, the roof slab and the wall. After pressing 'Enter' we are prompted to the select the detail objects. These are the objects that we wish to propagate or copy throughout the model, in this case the wooden beam, the concrete ring beam, the two covering elements and the gutter. After pressing Enter, a 2D detail is constructed. It is an extrusion of this detail that will be applied over the perimeter of the roof. Let's press Enter. 'Propagate' has now scanned the entire drawing for locations where this detail can be applied. A green check mark means that this detail will be applied when pressing Enter. The blue question mark means that the detail will not be applied, in this case because the original detail was only modelled locally; we will click it so it is replaced by an adapted extrusion, to make sure the detail is applied over the entire length of the wall. After pressing Enter, this detail is applied over the entire perimeter of our roof. Youtube Let's create a new section to design the connection of the roof with the end walls; we want the facing brick to protrude further into the roof structure. We will use 'Propagate Planar' to copy this detail to the other connections. The base solids are the roof slab and the wall. There are no detail objects in this case. We'll press Enter to accept the constructed 2D detail, and press Enter again to accept the three suggestions. We see now that the four corners of the roof are not what they are supposed to be; We can solve this detail once, and again use 'Propagate' to copy it to the other corners as well. We'll temporarily hide the gutter and these covering elements. We'll make smart use of 'push-pull' and other direct modelling tools to design this corner correctly. We've made sure that the insulation layers of both walls and the roof are now connected properly. We'll finish this corner by creating ends for the gutter and the covering elements. Now that we designed this corner correctly, we want to copy this situation to the other three corners. We will use the 'Propagate' command for this again. The base solids are the two walls, the roof slab, the gutter and the covering elements. These solids define the locations where this detail can be applied. The detail solids are the end caps of the gutter and the covering elements. These objects will be copied to the other locations. We will propagate this detail as a copy by entering 'C' in the command line or by clicking 'As Copy' in the context menu. The three other corners are suggested to copy this detail to. We'll accept by pressing Enter. Youtube Let's create a new layer and move the four roof slabs to this layer, so we can easily toggle their visibility. Note that we can change their layer using the rollover tips. We'll create the rafters by extruding this boundary and slicing the resulting solid. For this next step, we will temporarily turn off the subtract behaviour of 'push-pull' by going to the settings and typing in 'pushpull'. If we now push-pull this rafter so that it goes through the insulation layer of the wall, we see that this volume is not subtracted from the wall. This is an interference that we will solve later. Now that we've specified where the rafter ends, we want to copy this to the other side as well. We'll do this with 'Propagate Linear'. The base solids are the wall and the rafter. We want to specify where the rafter ends, so we will use its end face as detail object. By pressing Enter we see that the rafter on the other side of the roof now also has the correct geometry. We will copy these two rafters to the other end of the roof, again by using 'Propagate Linear'. You could view these two rafters as elements connecting the two concrete ring beams. Thus, our base solids are the ring beams, and our detail objects are the two rafters themselves. We'll choose to copy this detail as a copy, and accept the highlighted suggestion. Note that a small exclamation mark appears to indicate that an interference will be introduced when applying this suggestion. Next, we'll copy these rafters and repeat this copy, so that the entire length of the building is covered. Finally we will slice the rafters to make an L-shaped roof. Youtube Let's create some more wooden trusses. We'll use the 'Linear Solid' tool and select the profile in the model by clicking one of the rafters. Using the Manipulator we can easily shift this member so that it connects properly to the rafter and the beam. Remember: you can call the Manipulator by holding down the left-mouse button slightly longer when selecting an entity. We want this collar tie to be cut off diagonally, like we did on the other side. Instead of using 'slice' we can again use 'Propagate Linear'. The base solids are the collar tie and the rafter; there are no detail objects in this case. We will also use 'Propagate Linear' to copy these wooden members to the rest of our building. You could view these members as a connection between the beam resting on the ceiling slab, and the rafter. Thus, our base solids are this beam and the rafter, whereas the detail objects are the members we just modelled. 'Propagate' provides a suggestion for every location where a rafter meets this horizontal beam. Notice again the exclamation marks, indicating the interferences between the objects that will be copied, and the already existing entities. We'll press Enter to accept this. The same technique can be used to copy this member connecting the two rafters. We will now create a ridge beam and again propagate this beam so it is placed between every two pairs of rafters. Finally we will clean up the interferences. We want our rafters and wooden members to go through the insulation layer of the wall, so we will subtract these volumes from the wall. Now we'll use 'Propagate' again to copy the connection of these elements. Our base solids are the wall, the rafter and the wooden beam. There are no detail objects in this case. Note that we still have to manually solve the

2.7.2 How to automate classification with ”BIMIFY” in BricsCAD BIM

https://www.youtube-nocookie.com/embed/k3f1m-FWD_0 This video shows how BIMIFY can turn your conceptual model into a BIM model. Automatically detect IFC classifications, spatial locations, rooms and more. Back to Tutorial Overview

2.7.3 How to use Propagate Edges

https://www.youtube-nocookie.com/embed/kAZnvUNxrgI This video shows how Propagate Edges can be used to easily copy the design of a wall cap to all wall edges of the model. Back to Tutorial Overview

2.7.4 How to use Propagate Linear

https://www.youtube-nocookie.com/embed/ICDBTDb0xH8 This video shows how Propagate Linear can be used to quickly and easily copy detailed connections of steel members to other applicable locations in the model. Back to Tutorial Overview

2.7.5 How to use Propagate Pattern

https://www.youtube-nocookie.com/embed/rY_B4-JeTiU We can use Propagate to copy objects over a certain pattern. In this example we'll create a pattern of skylights in this roof. We'll launch Propagate Pattern. First, we'll select the base solid, which is the roof in this case. Next we'll select the detail. We want to create a pattern of skylights, so we'll select this skylight as our detail object. Hovering over the suggestion icon, we can see there are two options. The first option is to propagate to similar locations. In this case this will be the four corners of the slab. The second option is to create a grid on top of this slab, by adding rows and columns. We can explode this suggestion so we can individually toggle the different suggestions. In this case we'll use this to create a seemingly random pattern. Finally we'll press Enter to accept. If a Grid is drawn on top of the base solid, then this grid will be detected and a third option will automatically be presented: create a pattern according to this grid. Again, the two options that were available in the previous example are also available here. Now a third option is available: propagate along this Grid. We can again explode to have control over the individual suggestions. Back to Tutorial Overview

2.7.6 How to use Propagate Planar

https://www.youtube-nocookie.com/embed/Lmqh0-jreNI This tutorial shows how Propagate Planar can be used to easily copy a detailed wall-slab connection to other applicable locations in the model. Back to Tutorial Overview

2.7.7 How to use Propagate Corner

https://www.youtube-nocookie.com/embed/k_1k-7Tjwro This video shows how Propagate Corner can be used to fix modelling issues that were created in an earlier stage of the modelling process. Back to Tutorial Overview

2.8 Design Assistance

2.8.1 BIMIFY

Overview: In this lesson, you learn how to use BIMIFY to automatically classify elements, recognize profiles for linear solids that are available in the profile library, assign spatial locations, create BIM rooms, detect external walls and create model elevations and floor sections. Lesson Objectives After completing this lesson, you will be able to automatically:

- Classify 3D solids, or block references
- Attach BIM Profiles for linear solids
- Assign spatial locations to building elements
- Create BIM Rooms
- Detect whether walls are external or not
- Add elevation views (front, left, right, back) and a plan section per floor

Tutorial: How to automate classification with "BIMIFY" in BricsCAD BIM Click here (2.7.2) to watch

About BIMIFY

BIMIFY analyzes the solids and block references and runs automatic classification and spatial location assignment for an entire BIM model. When the new buildings and stories are required, BIMIFY creates them to meet the model needs. BIMIFY detects room entities and places them automatically in the model. It also detects whether walls are external or internal by indicating this information in the properties panel under the IFC Common property set and "Is External" property. When Is External property is ON, it means that the wall is categorized as an exterior wall.BIMIFY will not change the "Is External" property of a wall when it's already set to On by the user. Moreover, if the solids are auto-classified as column, beam or member and their profile match a definition in the profile library, the profile name will be automatically added to the entity's properties as meta-data. More importantly, BIMIFY can be used on models at different stages. For example, if you have a few classified entities and unclassified entities in your model, you can use the BIMIFY to automatically classify the unclassified ones. The recently classified entities are also associated with the appropriate spatial locations. BIMIFY always re-uses the information that has been already chosen by the user. For example, if you have a few classified entities and unclassified entities in your model, BIMIFY automatically classifies the unclassified ones and skips the already classified ones in your model.

Access



Menu: **BIM > Bimify** Quad: **BIM > Bimify** Toolbar: **BIM > BIMIFY** Keyboard: **BIMIFY**

Procedure:Using BIMIFY in your BIM model

1. Choose BIMIFY from the Quad or type BIMIFY in the command line.

A dialog box displays: ===ERROR WITH DOWNLOADED IMAGE===

Classification	Classifies 3D Solids and Block References automatically
Structural/MEP Profiles	Assignscolumns, beams, members or flow segments profiles that are available in BricsCAD profile library
Buildings, Stories, Rooms, and External Walls	Assigns spatial locations such as buildings and floors, detects rooms, identifies which walls are internal and external
Sections	Creates elevation views and a plan section per floor

Ticking a checkbox allows to use a specific task or sub-task of BIMIFY. For example, clicking the classification checkbox only permits you to use the BIMIFY for auto-classification of solids and block references. **Note:** Here you can decide whether solids and block references should be automatically classified together or not. To disable the auto-classification option for solids simply click the related checkbox in the dialog box or vice versa. By default, all checkboxes are ticked. Ticking the Buildings, Stories, Rooms, and External Walls checkbox automatically includes the Classification option since this information is needed to find the other elements.

Figure 2.357

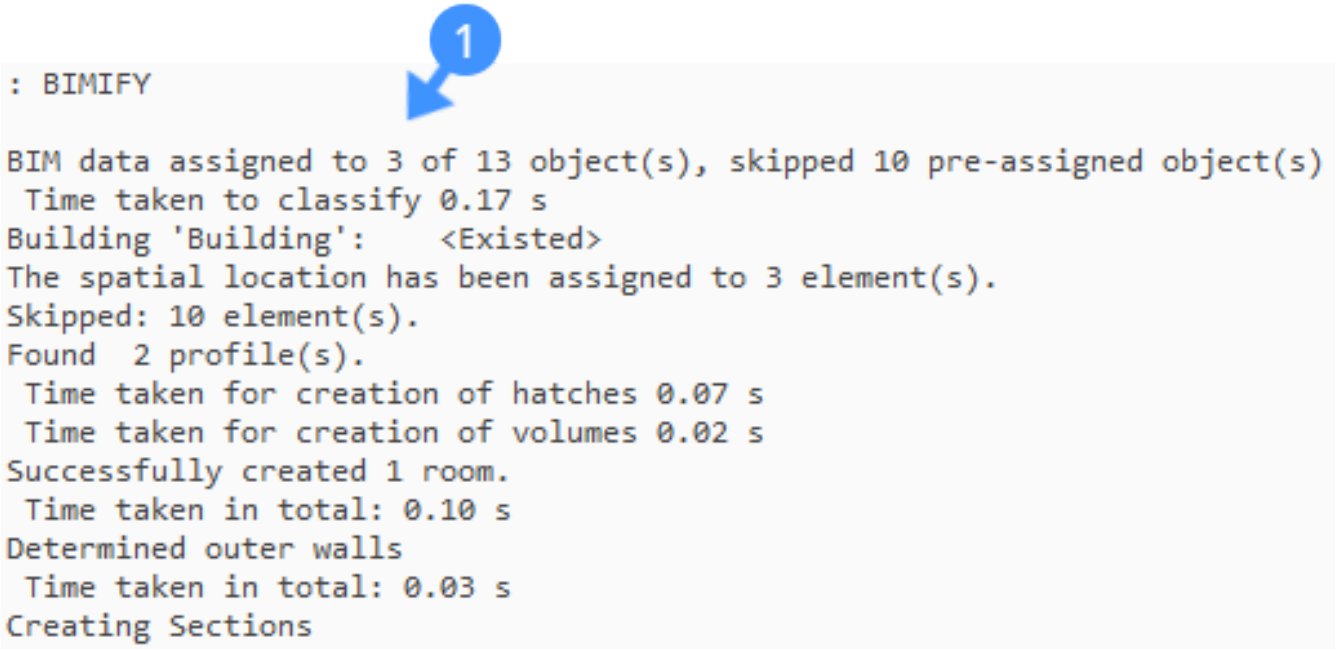
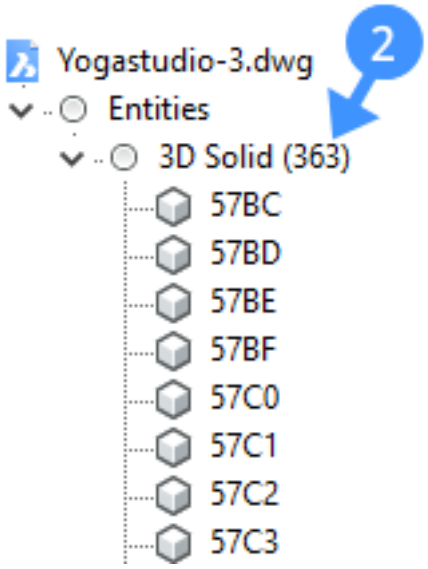


Figure 2.358



2. When the BIMIFY process is completed, an overview of the result (1) is provided in the Prompt History panel.

Figure 2.359

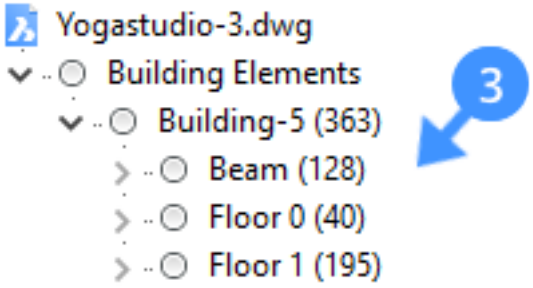


Figure 2.360

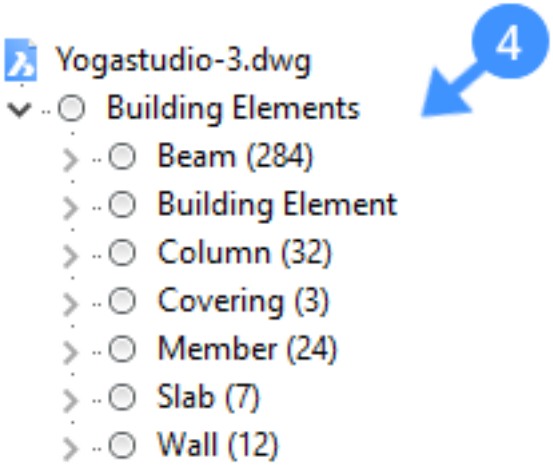


Figure 2.361

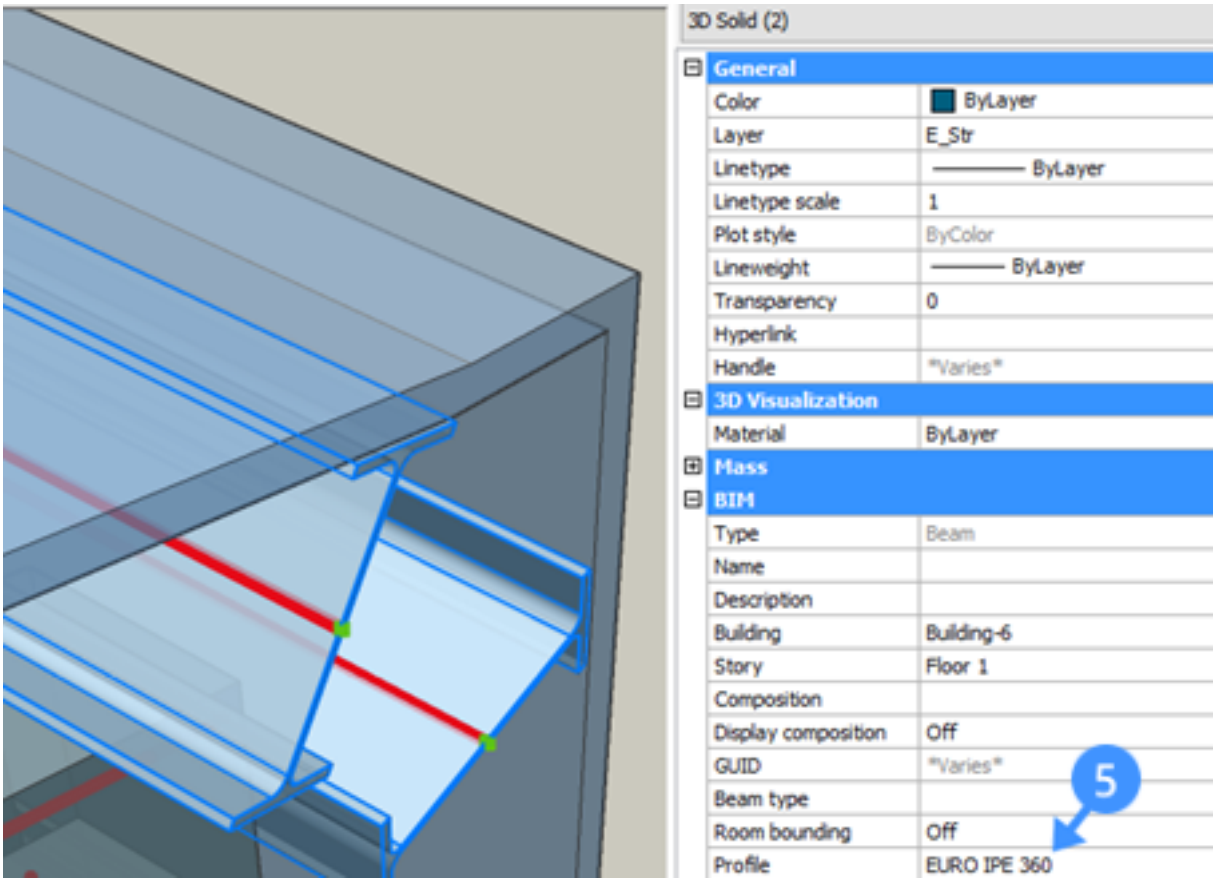
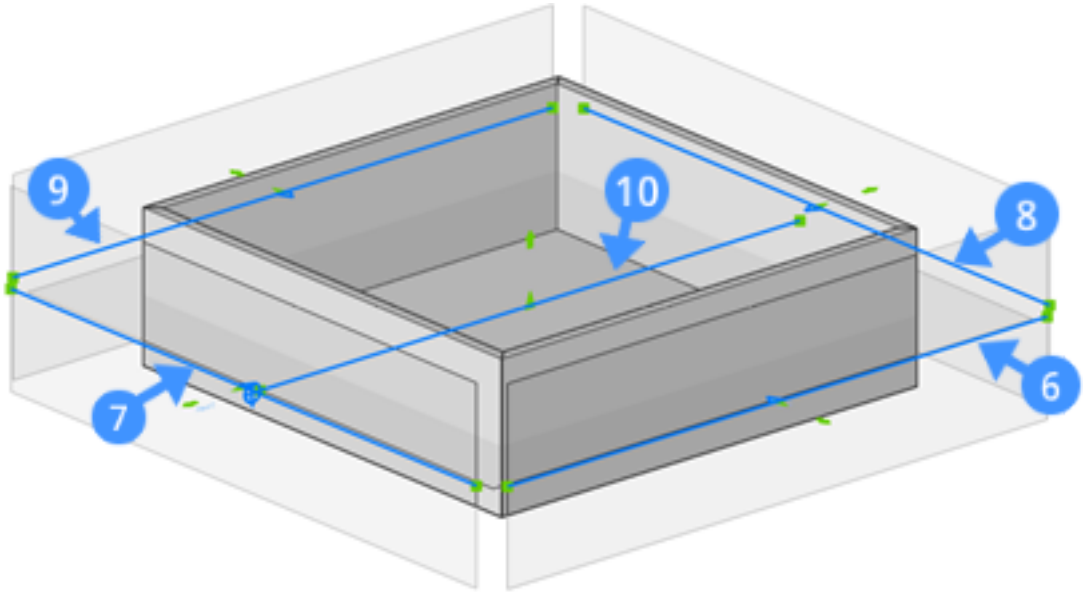
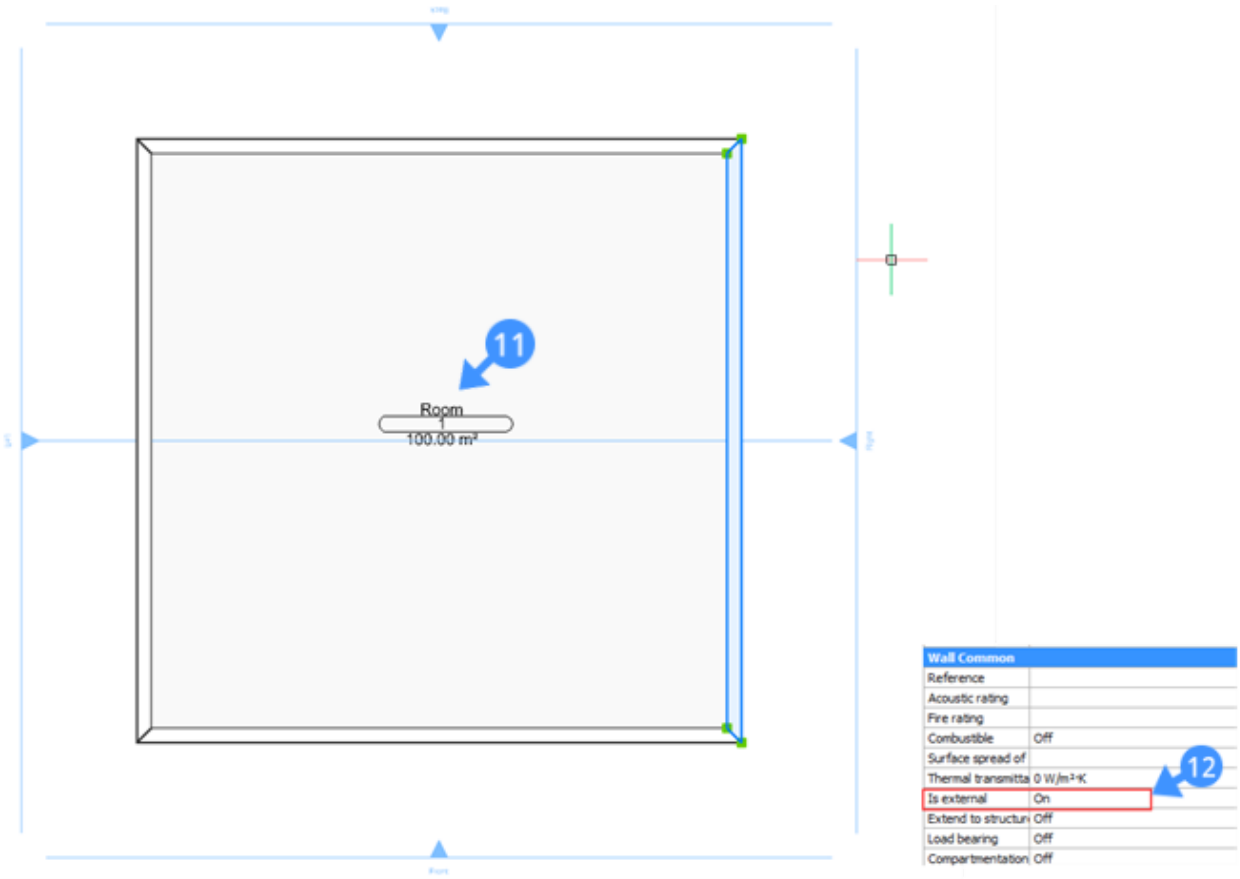


Figure 2.362



There will be no prompts or dialog box at the end of the process. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/artic
prompthistory.png}) **Optional** When you model more building elements in your project you can use the BIMIFY later to classify them automatically. BIMIFY checks whether all entities are classified and have appropriate spatial locations. If spatial locations have already been created, these are used to locate the entities. If the entities are required new spatial locations, BIMIFY also creates new ones. The following illustrations show the structure tree that displays the contents of the building model. In the beginning, the structure tree only had 3D solids (2), however after using BIMIFY, the building elements are displayed in their building and stories (3) and the building element types (4) since the automatic classification through BIMIFY has been made for these elements. **Note:** The structure browser can be customized to control the content of the tree. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360021819133/BIMIFY-StructureBrowser1.png}) (\vref {fig:C:/Dev/Tas
2448/images/helpbricsyscom/hc/articleattachments/360020900894/BIMIFY-StructureBrowser2.png}) (\vref {fig:C:/Dev/Tas
2448/images/helpbricsyscom/hc/articleattachments/360020900974/BIMIFY-StructureBrowser3.png}) The following illustration shows beams profile data (5) in the properties panel. BIMIFY recognized the profile of these model elements as structural members and detected the axis location of the element. (\vref {fig:C:/Dev/Task-2448/images/helpbricsys
ProfileRecognition.png}) The following illustration shows the elevation views and floor plan on a simple building after using BIMIFY. This building contains front (6), left (7), right (8), and back (9) views as well as a floor plan (10). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360020906054/BIMIFY-Sections.png}) For this simple building, one room and external walls are detected. The image below depicts the BIM room entity (11) and "Is External" (12) property under the Wall Common properties set. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360020906974/BIMIFYRoomIsExternal.png})

Figure 2.363


11
Room
100.00 m²
Wall Common
Reference
Acoustic rating
Fire rating
Combustible Off
Surface spread of
Thermal transmitta 0 W/m²·K
Is external On
Extend to structur Off
Load bearing Off
Compartmentation Off
12

2.8.2 Using Propagate

Command: BIMPROPAGATE (6.7.20)

About

PROPAGATE can be used to copy details throughout your model. It maps details related to selected base solids to similar base solids in the model. Anytime a certain detail is needed in multiple locations in the model (e.g. a connection between steel members, a wall-slab connection, a roof edge detail, ...) chances are the quickest way to solve it is by using PROPAGATE.  Quad: **Model > Propagate** Keyboard: **BIMPROPAGATE** Menubar:**BIM > Propagate**

The Five Flavours

Before getting into the PROPAGATE workflow it is important to note that there are five variants of PROPAGATE, each working on a limited set of situations. These five variants (or flavours) were designed to guide new users through the process of PROPAGATE. The tool has so many different uses that it can seem overwhelming to a new user; these variants help keep the tool more clear and concise, as they are designed for more specific use cases. However, they are still distillates from original PROPAGATE command, so every problem you can solve with a particular flavour you can **also** solve using the standard PROPAGATE command. A more advanced user will even be able to solve problems that can not be solved by any of the particular flavours. The PROPAGATE variants are:

-  BIMPROPAGATEPLANAR (2.8.6): Propagation of connections between planar elements such as walls, slabs and roofs.
-  BIMPROPAGATELINEAR (6.7.23): Propagation of connections between linear elements such as beams, columns, pipes and ducts, and maximum one planar element.
-  BIMPROPAGATEPATTERN (6.7.24): Propagation of a single element or group of elements on a flat surface to multiple locations and grids; can be used to propagate light fixtures, light switches, windows, air diffusers, columns on a grid
- ===NO VALID URL FOR IMAGE=== : Propagation along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc.
-  BIMPROPAGATECORNER (6.7.21) Propagates details connected to multiple planar base solids coming together in one node.

General Procedure

The general workflow of PROPAGATE consists of two major steps: defining the detail volume, and choosing where to apply this detail.

Defining the detail volume

Figure 2.364

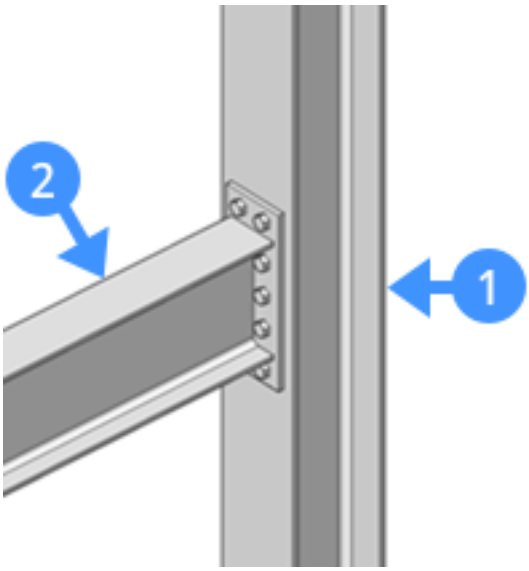
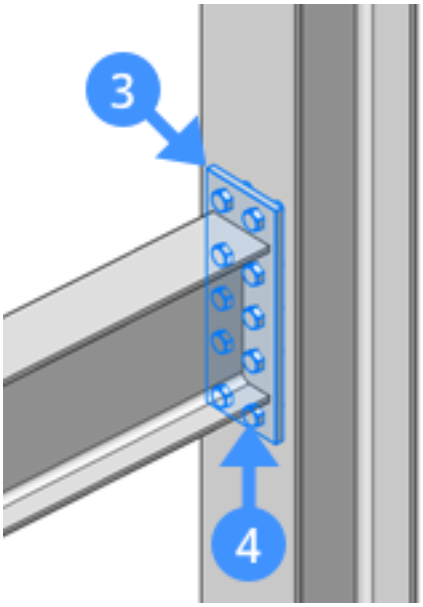


Figure 2.365



You are prompted:Select base solids: The first step of every propagation is specifying the **base solids**. These are the solids that define the situation where this particular detail should be applied. For instance, the image below shows a column-beam connection using an end plate and several bolts. This detail was modelled for this particular beam connecting with this particular column, and now we want to propagate this detail for every situation where a column and beam of these types come together. Thus, the base solids in this case will be the column (1) and the beam (2). Later in the process, PROPAGATE scans the model for situations where these base solids occur, and tries to map the detail onto these locations. **Note**: this is an example of a situation that can be solved by BIMPROPAGATELINEAR (6.7.23). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/Propa
Press Enter to stop selecting base solids. The second step is specifying the **detail objects or sub entities**. These are the objects that you wish to propagate or copy throughout the model. They can be solids, block references or even faces (e.g. a hole in the base solid can be propagated by selecting its faces). You are prompted:**Select detail objects or detail sub entities (optional):** In the example at hand the detail objects will be the end plate (3) and the bolts (4). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagateDetailObjects.
Note that in some cases detail objects are not even required. Let's say we want to propagate the connection between the floor slab and the wall shown in the image below. Our base solids are then the floor slab (5) and the wall (6), and no detail objects are required. In this case you can skip this step by pressing Enter or right click. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagateNoDetailSolids.png})
If we were to model extra solids in this connection (e.g. an insulation block or a skirting board), these extra solids would then be detail objects. **Note**: this is an example of a situation that can be solved by BIMPROPAGATEPLANAR (6.7.25). Once the **base solids** and the **detail objects** are specified, a detail volume is defined automatically. The visual style (6.31.13) is temporarily set to X-ray, and the view is automatically zoomed in onto the detail volume. Everything that is inside this volume and is either a detail object or part of a base solid will be copied to the other locations when executing propagate. A detail volume can be either a **3D volume** or a **2D slice through a linear detail**. In our example of the column-beam connection, the detail volume will be as shown in the image below. This is

Figure 2.366

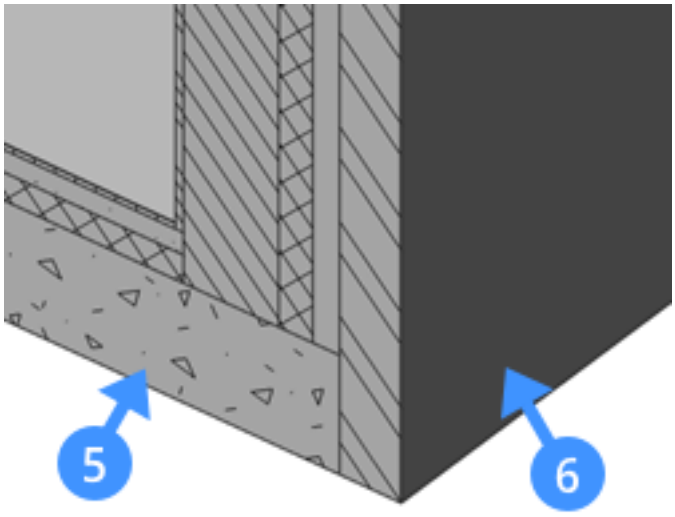


Figure 2.367

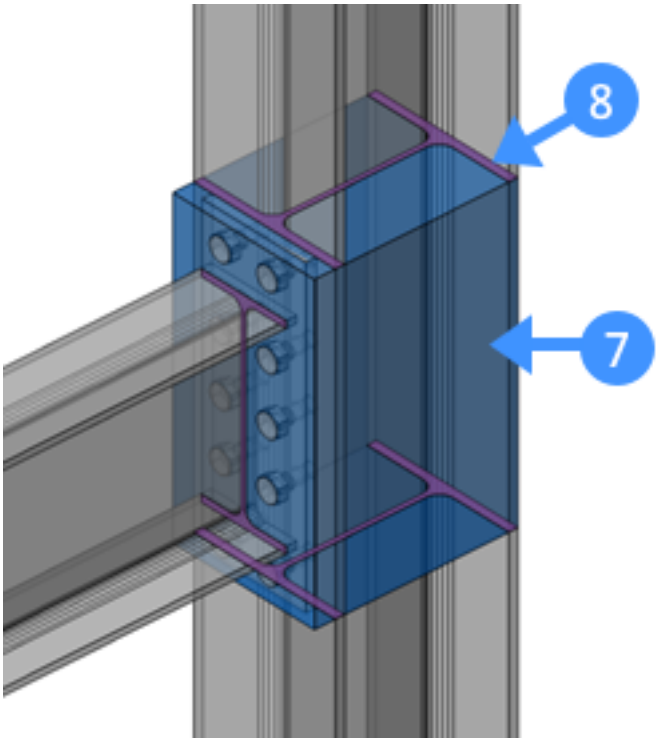


Figure 2.368

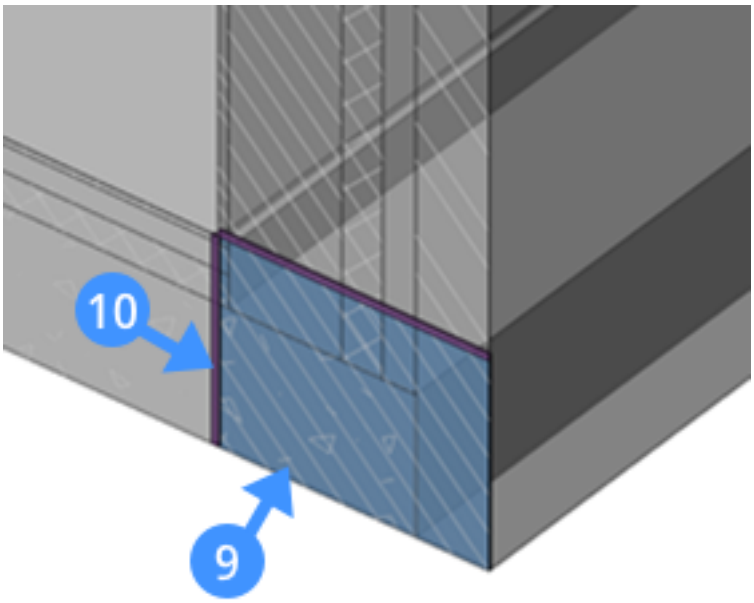
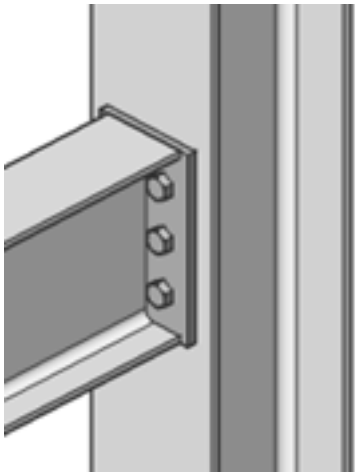


Figure 2.369



- If you want to replace an already existing detail with a new detail that is **smaller** than the original one. For example, we want to replace the previously proposed column-beam connection by the following: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagateSmallerDetail.png} In this case, we will have to inflate the detail volume (11) so that it is larger than the original detail: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagateInflatedVolume.pn

- You might want to include a larger part of the base solid, for example to connect with target solids that are further apart. In the image below, we want to propagate the connection between the **red** and the **yellow** beam, so that the **yellow** and the **green** beem have a similar connection. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he To achieve this, we need to inflate the detail volume so that it overlaps the end faces of the green and the yellow beam. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagateIn

Once the detail volume includes everything you want to propagate, you can go over to the second step: choosing where to apply the detail. For 3D detail volumes there is one last choice to make: Propagate the detail as a Block or as a Copy. If you choose to propagate the detail as a block, then it will group all detail objects into a new Block Definition and copy around references to this Block Definition.

Figure 2.370

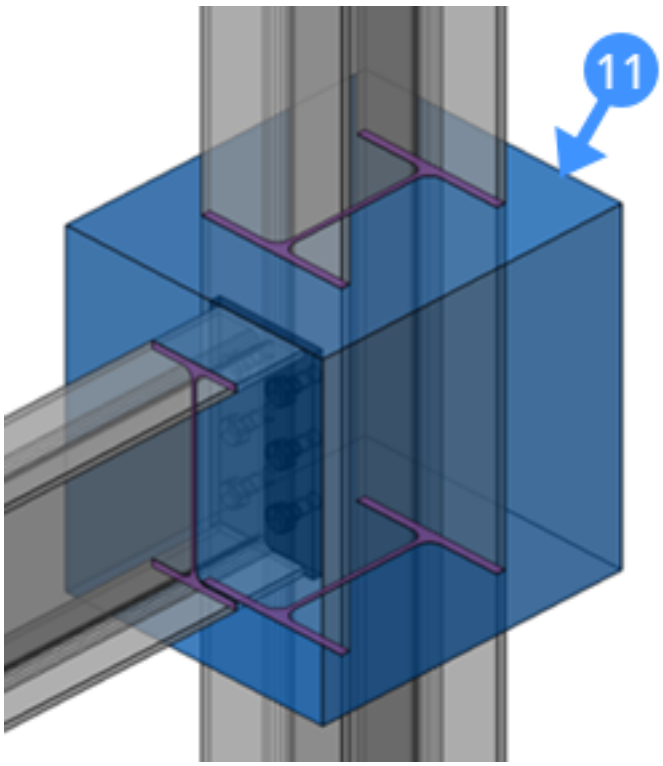


Figure 2.371

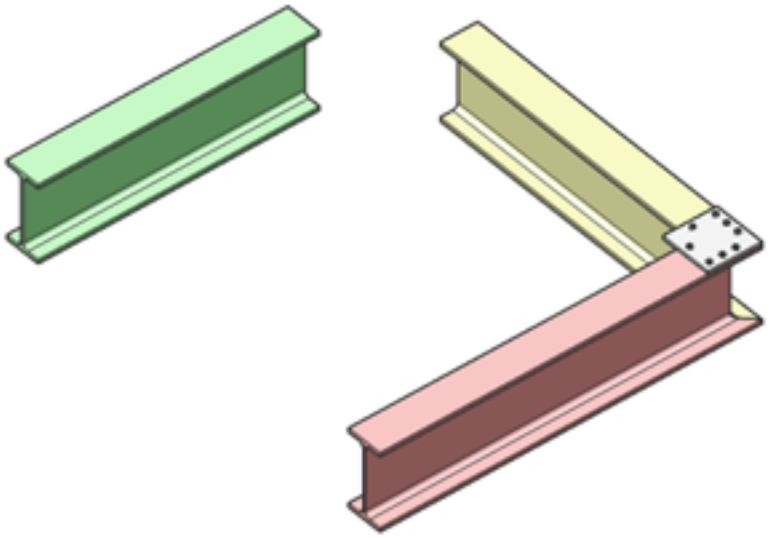
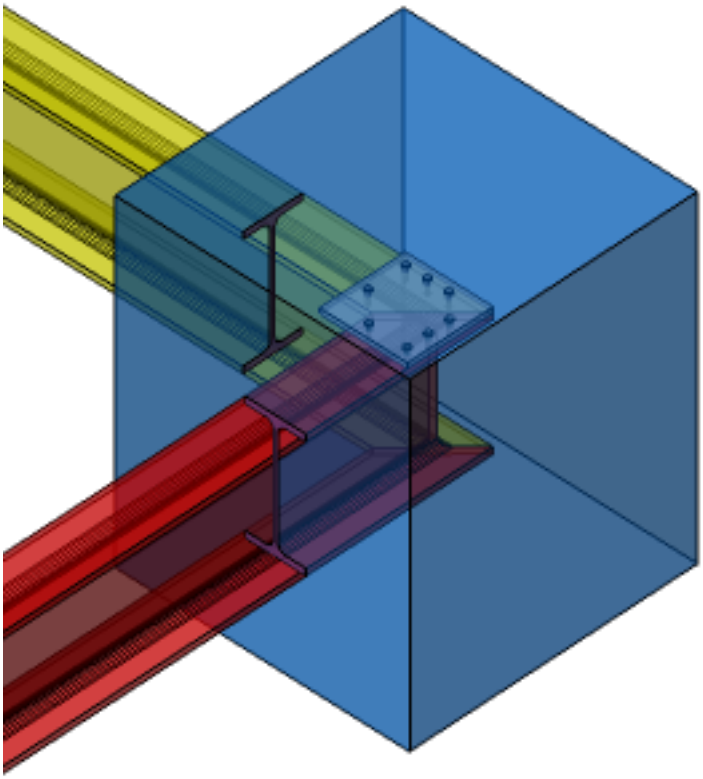


Figure 2.372



Choosing where to apply the detail You are prompted: **Choose next action [Apply all/Cancel] <Apply all>:** In this step the view zooms out showing all the possible suggestions on screen. Each suggestion is a location where this detail could be applied, and is indicated by one of the following icons (or widgets):

	The detail will be applied
	The detail will not be applied; it was manually toggled off by the user
	The detail will not be applied, because of one of several possible reasons. Hovering over the icon will provide more information
	The detail will be applied, but interferences were detected
	The detail will be applied, but there are several alternatives. Hovering over the icon will allow you to cycle between alternatives

Hovering over a widget reveals some more information. Clicking a widget changes its status: you can toggle suggestions individually by clicking their respective widgets. Pressing Enter will then apply all the widgets as they are shown at that point: suggestions with a green checkmark will be applied, others will not be applied. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagateAllWidgets.png})

A suggestion also shows a preview of what the detail will look like. In the images below, you can both see the widgets (12) and the previews (13). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssis

Widgets can also be **selected** using a window selection. A selected widget is highlighted by a blue border. Once one or more widgets are selected, pressing Enter will no longer apply **all** the suggestions, but **only those that are**

Figure 2.373

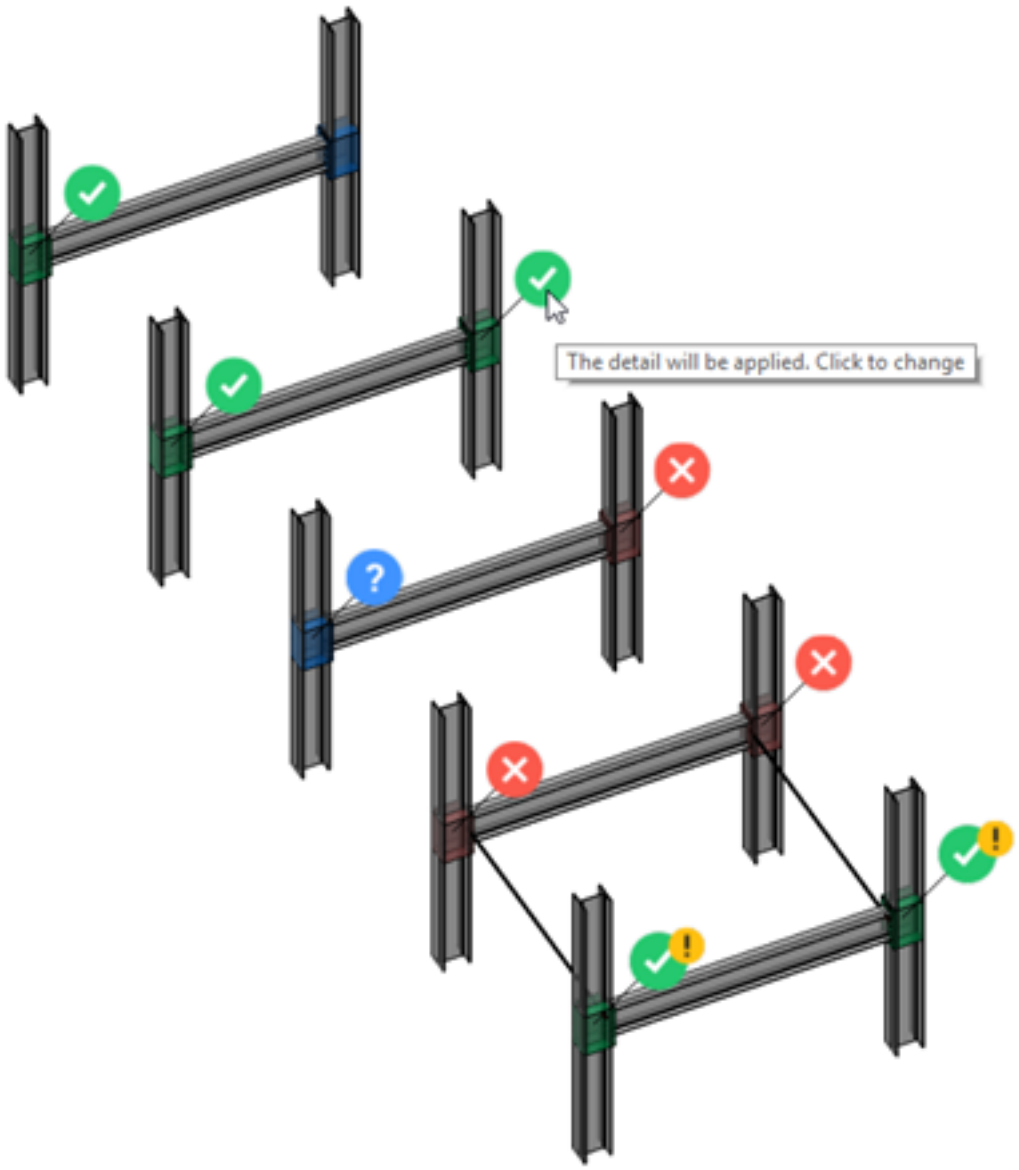


Figure 2.374

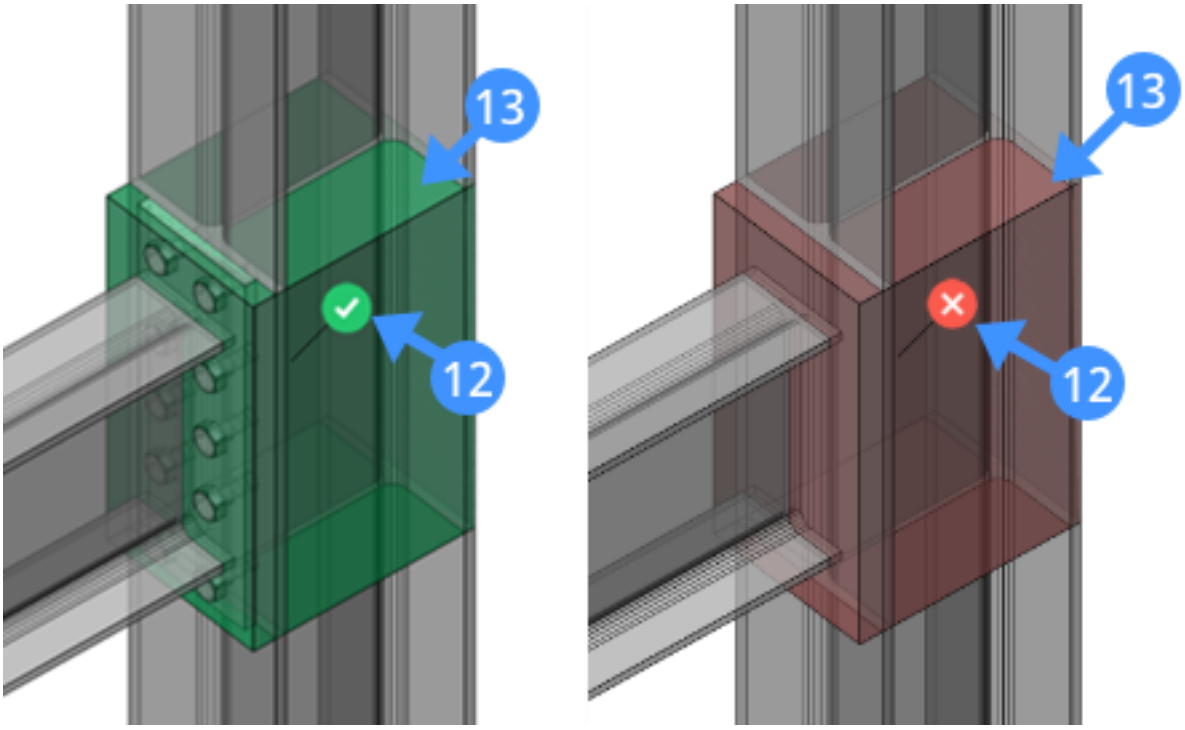


Figure 2.375

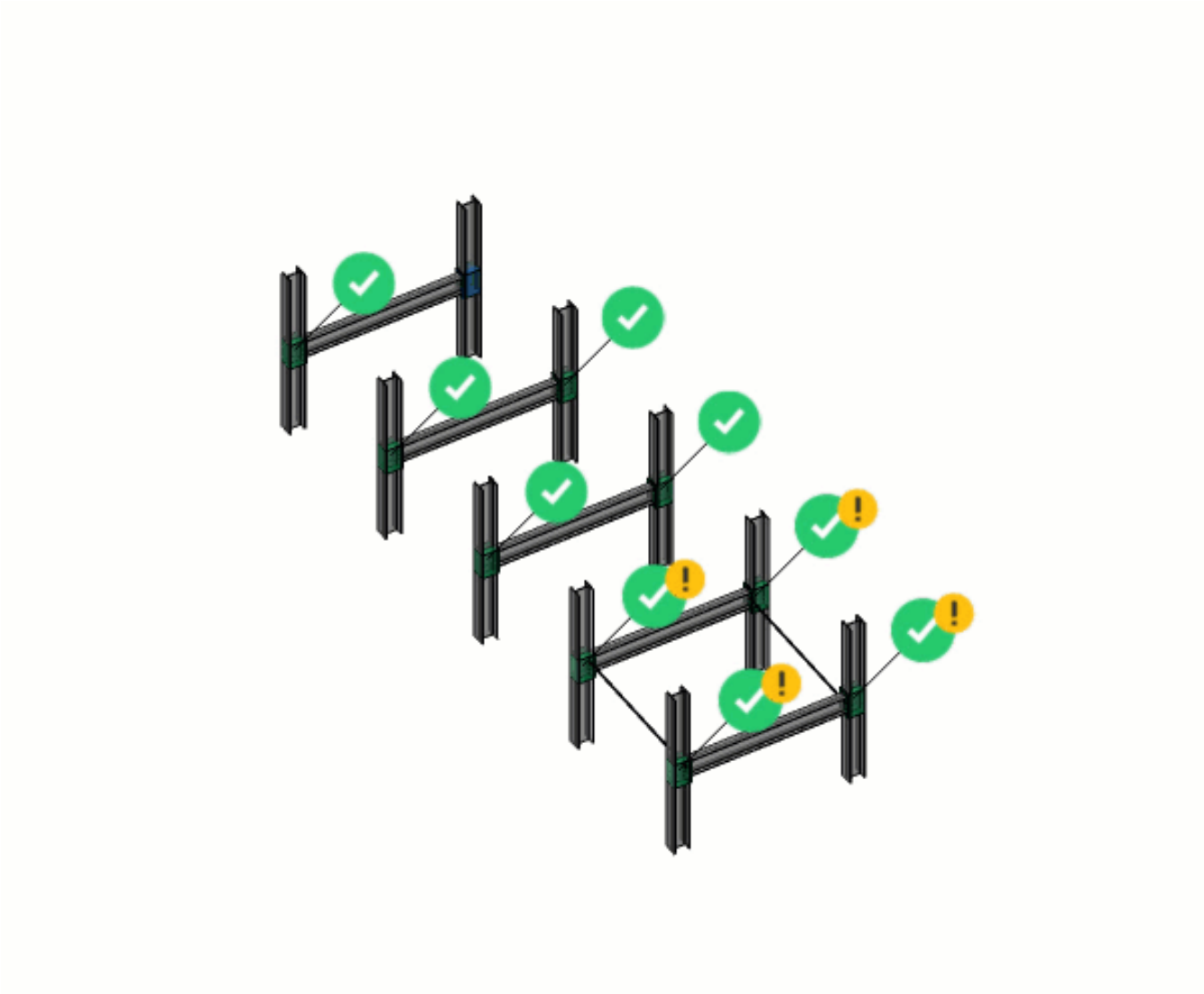
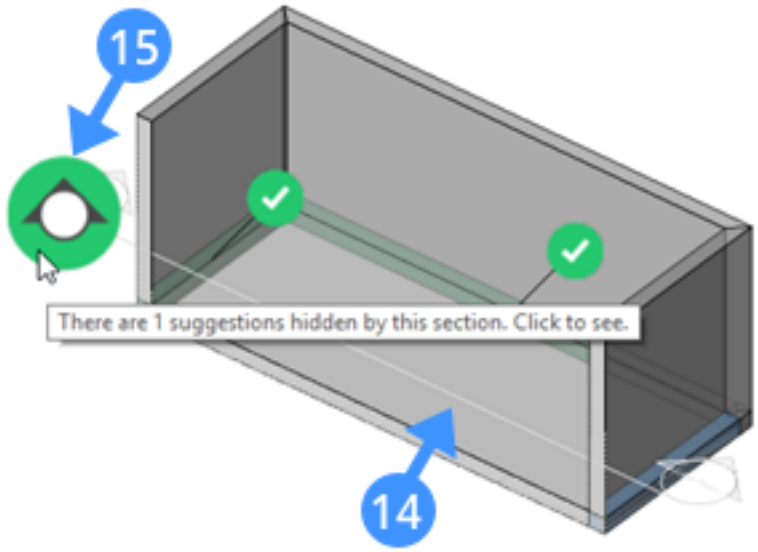


Figure 2.376


15
There are 1 suggestions hidden by this section. Click to see.
14

2.8.3 Propagate Edges

About: BIM Propagate Edges

BIMPROPAGATEEDGES is one of the five Propagate flavours. It can be used to copy details along edges of planar solids. A common example is a wall cap along the edges of a flat roof. For a more in-depth explanation of the general principles of Propagate, read thisarticle on Propagate (2.8.2).

Tutorial: How to use Propagate Edges Click here (2.7.3) to watch

Procedure: Propagating a wall cap

Consider the following situation of a one-storey building with a flat roof. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom
Let's create a vertical section plane to cut through our building and take a look at where the wall (1) and the roof slab (2) come together. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/Propagate
We want to create a capping (3) on top of this wall to get the following result: (\vref {fig:C:/Dev/Task-2448/images/staticbricsys
We can start by creating this locally on top of one wall, either by using direct modelling tools or inserting a block from somewhere else and exploding it. In our section it might look like this: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h
Now we can use Propagate to copy this detail over to all applicable locations. Let's deactivate the section and launch BIMPROPAGATEEDGES. **You are prompted:Select planar solid to which edge detail is related.** Select the wall on top of which the capping was modelled, and press Enter. **You are prompted:Select detail objects to be included in the edge detail (solids or faces).** Select the objects that make up the capping, and press Enter. **You are prompted:Propagate detail as an extrusion of this 2D Section? [Yes/No/Inflate first] <Yes>:** Inflating the detail is not necessary in this case. For more information on inflating, read thisarticle on Propagate (2.8.2). Choosing 'No' will exit the command. Press Enter to accept. **You are prompted:Propagate to edges with similar orientation only? [Yes/No] <Yes>:** Choosing **Yes** will only apply this detail to edges with a similar orientation, in this case wall edges that point upwards. Press Enter to accept. BIMPROPAGATEEDGES now scans the entire drawing for similar locations. When this is done, the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/Propag
You are prompted: Choose next action [Apply all/Cancel] <Apply all>: A green check mark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information. In this case it says **"Location of original detail. Click to replace original detail by adapted extrusion".** Because we modelled the wall cap only locally, we should replace the original detail by an adapted extrusion so that it covers the entire length of the wall. Clicking the widget will turn it into a green check mark. Press Enter to complete the command. Now all four walls should have this wall cap. Also the corners should be solved automatically.

Figure 2.377

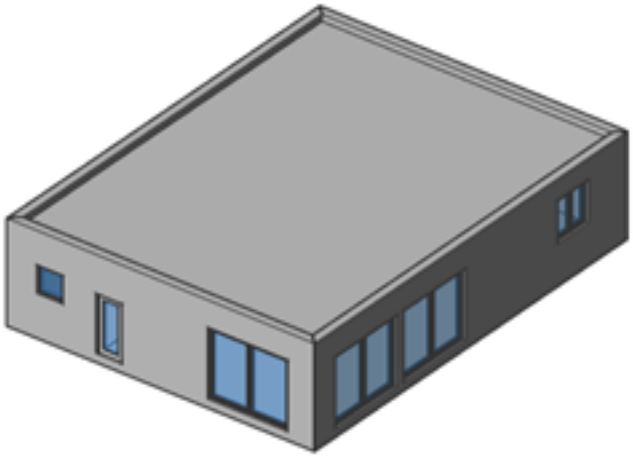


Figure 2.378

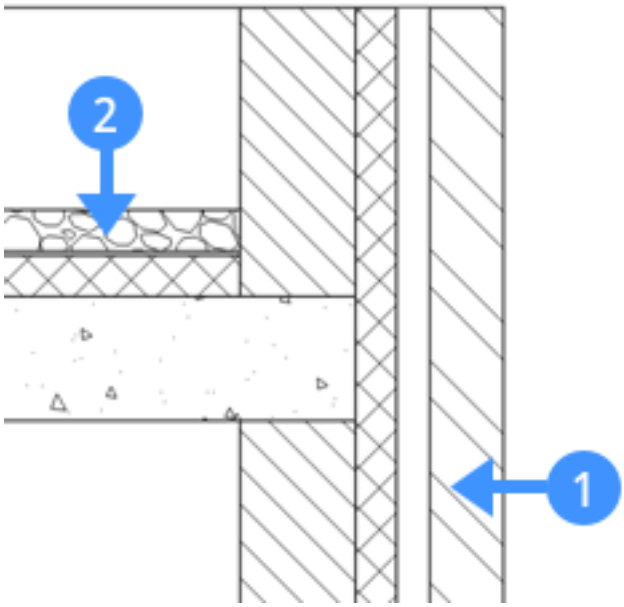


Figure 2.379

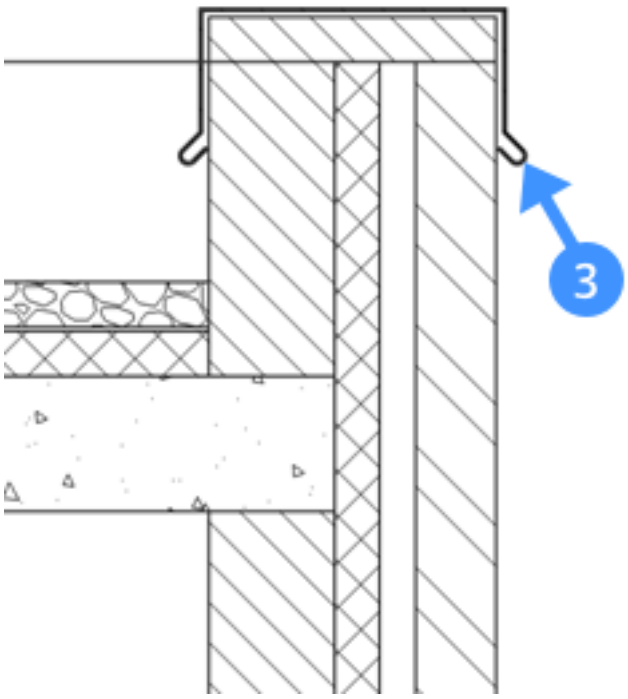


Figure 2.380

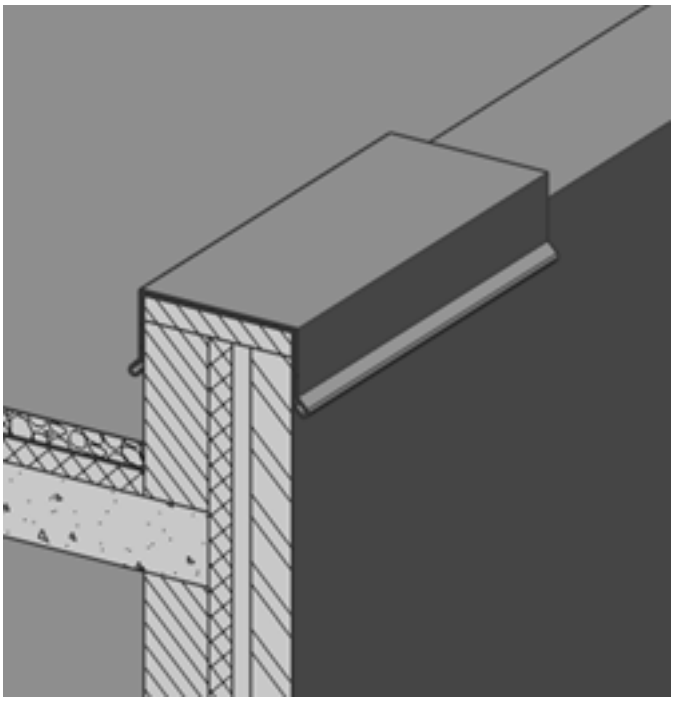
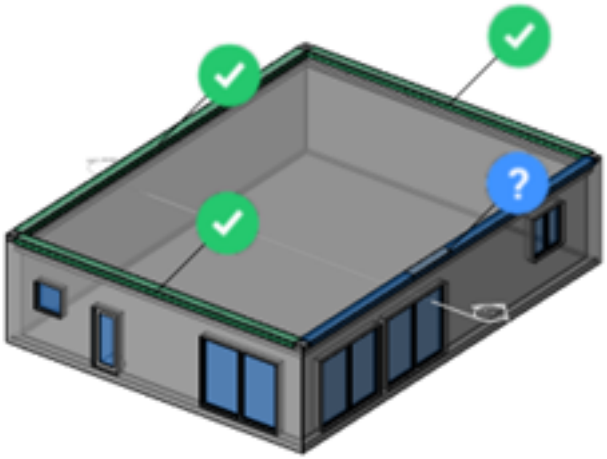


Figure 2.381



2.8.4 Propagate Linear

About

BIMPROPAGATELINEAR is one of the five Propagate flavours. It can be used to copy detailed connections between two or more linear solids. Common examples are column-beam connections or column-slab connections. For a more in-depth explanation of the general principles of Propagate, read this article on Propagate (2.8.2).

Tutorial: How to use Propagate Linear Click here (2.7.4) to watch

Procedure: Propagating a column-beam connection

Consider the following situation of four columns and two beams. Two of the connections have been designed with an and plate and bolts trough the column flanges (1). One connection has been designed with a diagonal member for additional rotational stiffness (2). The last connection has not been designed yet (3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagateLinearSituation.png}) (\vref {fig:C:/Dev/Ta 2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagateLinearConnections.png}) We can now use Propagate Linear to copy one of the details to the other locations. In this example we will copy connection 2 to all four locations. We will launch BIMPROPAGATELINEAR. **You are prompted:Select linear or planar solids to connect. At least one should be linear.** Select the column and the beam that are connected by the diagonal member, and press Enter. **You are prompted:Select detail objects (solids, block refs or faces) to be included in the connection (optional).** Select the diagonal member and press Enter. A detail volume is created that completely encompasses the diagonal member and parts of the column and beam. **You are prompted:Propagate this 3D detail? [as Block/as Copy/No/Inflate first] <As block>:** You can choose to propagate this detail as a block to keep the file smaller and more manageable, or as a copy. Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate (2.8.2). Choosing 'No' will exit the command. Press Enter to accept. BIMPROPAGATELINEAR now scans the entire drawing for similar locations. When this is done, the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagateLinearSuggestions.png}) **You are prompted:Choose next action [Apply all/Cancel] <Apply all>:** A green check mark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information. In this case it says "**Existing details detected. Suggestion will not be applied. Click to change.**" We want to replace the bolted connection by this diagonal member, so we click both the blue widgets so they turn into green check marks. If we now press Enter, all connections should be replaced by the diagonal member.

Figure 2.382

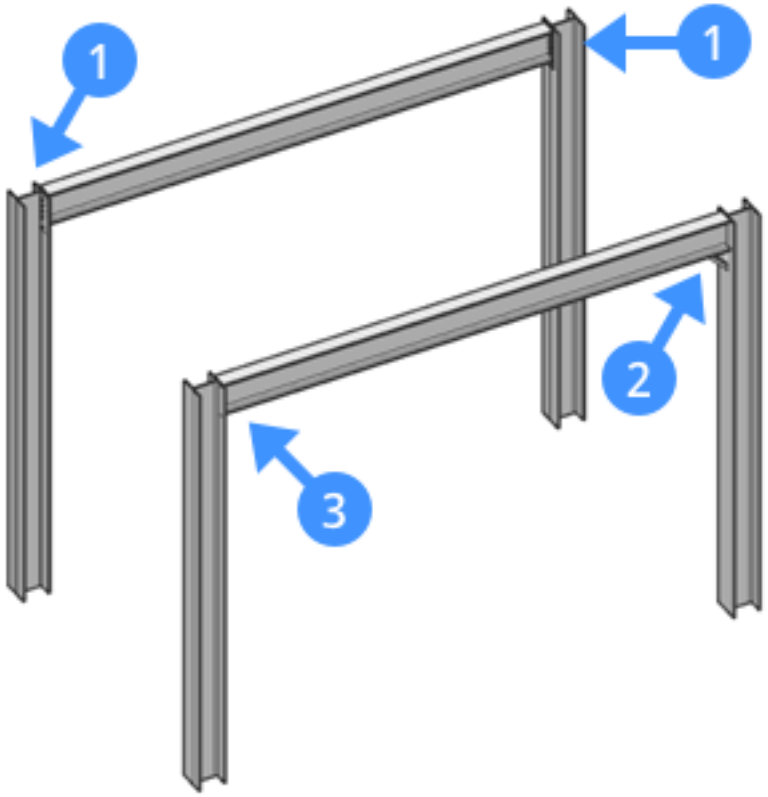


Figure 2.383

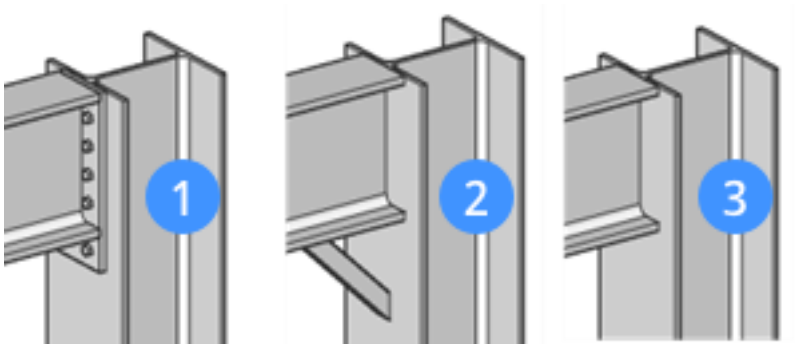
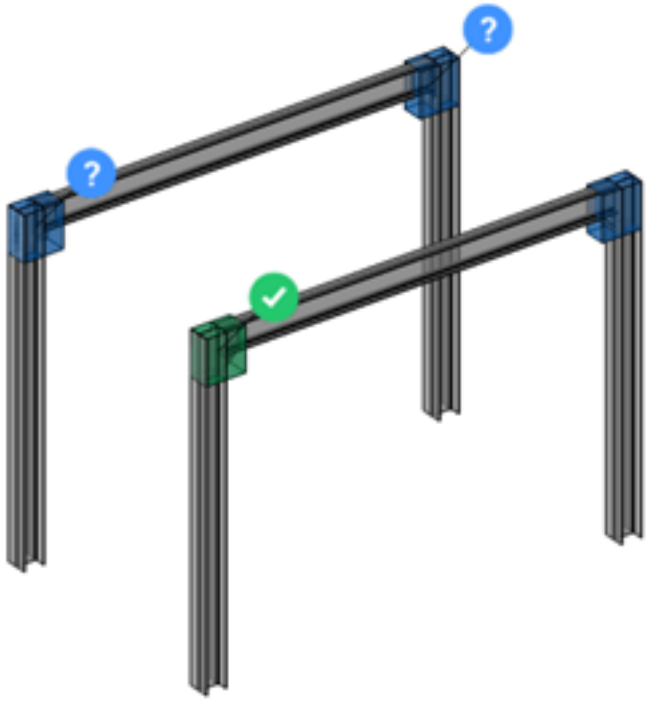


Figure 2.384



2.8.5 Propagate Pattern

About: BIM Propagate Pattern

BIMPROPAGATEPATTERN is one of the five Propagate flavours. It can be used to copy objects to similar locations or over a certain pattern or grid on top of a planar solid. Common examples are columns on a grid or windows in a wall. For a more in-depth explanation of the general principles of Propagate, read thisarticle on Propagate (2.8.2).

Tutorial: How to use Propagate Edges Click here (2.7.5) to watch

Procedure: Propagating columns on top of a slab

Consider the following situation: a column (1) is placed on top of a slab (2). There is also a grid drawn on top of this slab (3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagatePatternSit
We want to place this column on specific axis intersections. We could manually copy the column, but Propagate allows us to do this faster. Let's launch BIMPROPAGATEPATTERN. **You are prompted:Select planar solid to which detail is related.** Select the slab, and press Enter. **You are prompted:Select detail objects to be propagated (solids or faces).** Select the column, and press Enter. A 3D detail volume is created. **You are prompted: Propagate this 3D detail? [as Block/as Copy/No/Inflate first] <As block>:** Inflating the detail is not necessary in this case. For more information on inflating, read thisarticle on Propagate (2.8.2). Choosing 'No' will exit the command. Press Enter to accept. A widget appears. Hovering over this widget reveals a flyout menu with several options: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagatePatternOptions.png})

- Bim Grid: this is the grid that was drawn on top of the slab. Choosing this option and pressing Enter will copy this column to every axis intersection of this grid.
- Similar Location: this will copy the column to similar locations on this slab; in this case the four corners of the slab.
- Grid: this allows you to create your own grid of columns on top of this slab. You can choose to add or remove rows and columns. These rows and columns will always be evenly spaced.

We'll choose the Bim Grid option. Clicking this will expand the menu further; we now have the option to Explode. This will create a separate widget for each instance of this column, giving you more control over which suggestions you wish to apply (in this case which axis intersection you wish to place a column). We can now manually turn off suggestions: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagatePatternE
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagatePatternResult.png}

Figure 2.385

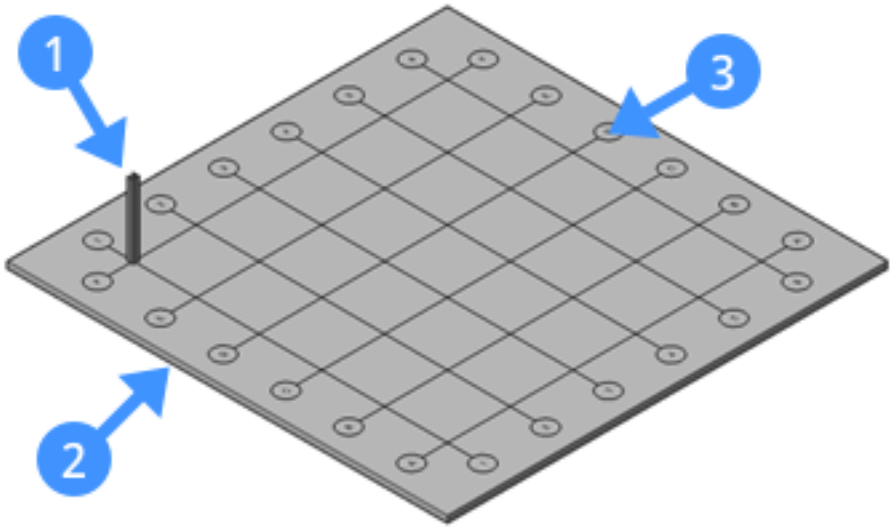


Figure 2.386

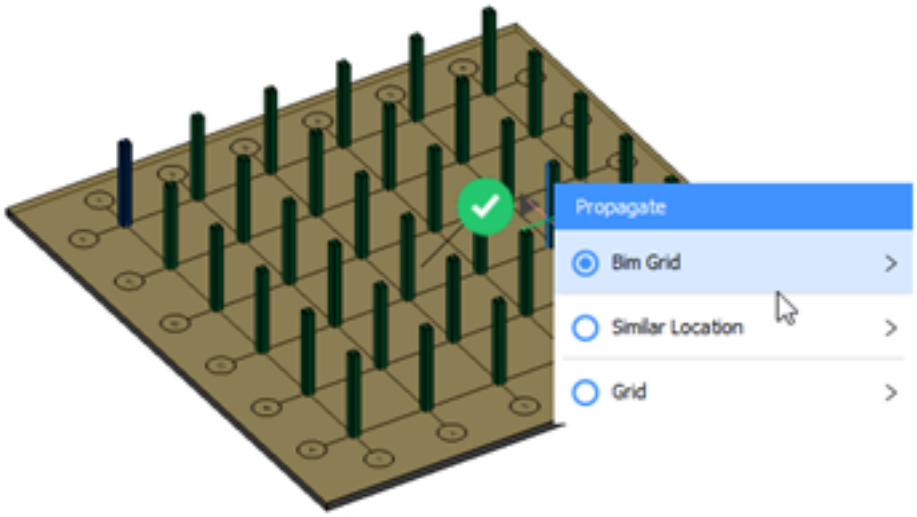


Figure 2.387

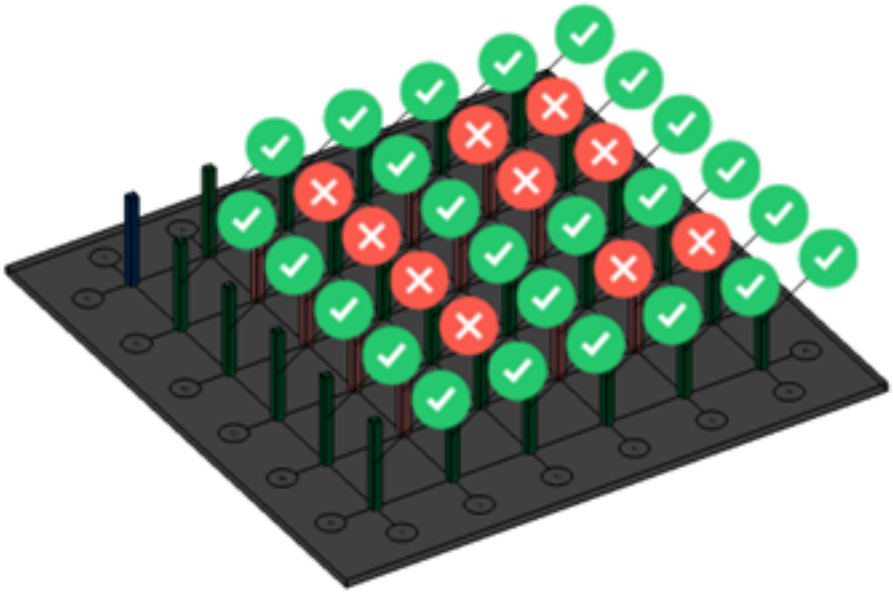
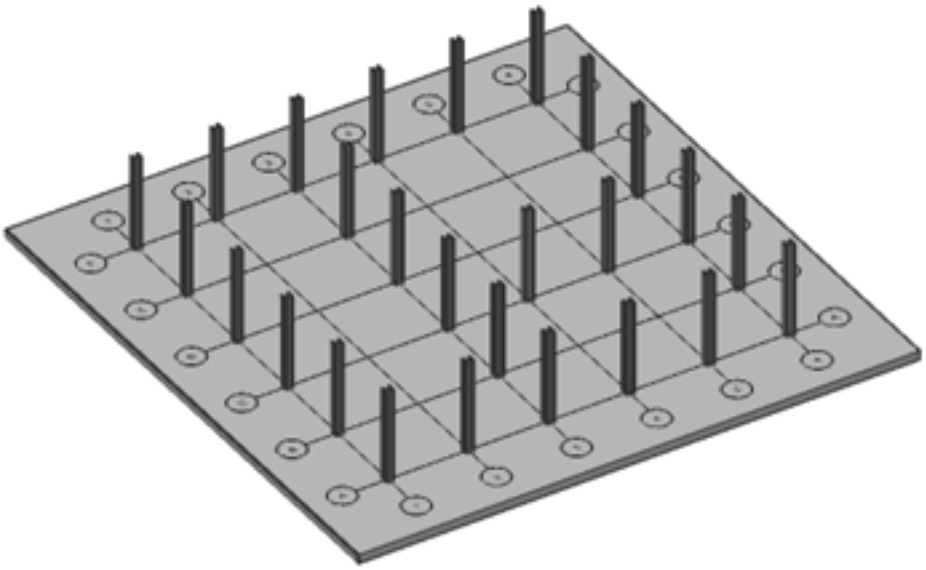


Figure 2.388



2.8.6 Propagate Planar

About

BIMPROPAGATEPLANAR is one of the five Propagate flavours. It can be used to copy detailed connections between two or more planar entities. Common examples are wall-slab connections or wall-wall connections. For a more in-depth explanation of the general principles of Propagate, read thisarticle on Propagate (2.8.2).

Tutorial: How to use Propagate Planar Click here (2.8.6) to watch

Procedure: Propagating a wall-slab connection

Consider the following situation of four walls and one slab, created with QUICKDRAW. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagatePlanarFourWalls.png}) The four walls were given a composition, as well as the floor slab. Without any further editing or modelling, creating a vertical section through the building will show the connection one of the walls (1) and the slab (2) to be something similar to the following image. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/Propaga
While this section is active, we can Push/Pull the different ply faces to obtain the desired connection between the wall and the slab: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagateP
We can detail this connection even further, for example by locally adding a skirting board (3). This was done by simply drawing a box and classifying this solid as a Building Element. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/
Now that we have this detailed connection, we want it to be applied to every location where a wall and slab of this type come together. Let's deactivate the section and launch BIMPROPAGATEPLANAR. **You are prompted:Select planar solids to connect.** Select the wall and slab, and press Enter. **You are prompted:Select detail objects (solids or faces) to be extruded along the connection (optional).** Select the skirting board solid, and press Enter. **You are prompted: The detail section varies too much along the extrusion direction.** In this particular case, BIMPROPAGATEPLANAR cannot find a solution: because the skirting board is not modelled along the entire length of the wall, BIMPROPAGATEPLANAR finds contradicting detail volumes. This problem can be solved easily by activating a section where the skirting board is present (e.g. the section we used earlier to create this detail), and trying again. If we now follow the same steps as before, BIMPROPAGATE finds a detail volume as shown in the image below. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagatePlanarDetailVolume.png
You are prompted: Propagate detail as an extrusion of this 2D section? [Yes/No/Inflate first] <Yes>: Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate (2.8.2). Choosing 'No' will exit the command. Press Enter to accept. BIMPROPAGATEPLANAR now scans the entire drawing for similar locations. When this is done, all sections are deactivated and the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignAssistance/PropagatePlanarWidgets.png}) **You are prompted: Choose next action [Apply all/Cancel] <Apply all>:** A green check mark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information. In this case it says **"Location of original detail. Click to replace original detail by adapted extrusion".** Because we modelled the skirting board only locally, we should replace the original detail by an adapted extrusion so that it covers the entire length of the wall. Clicking the widget will turn it into a green check mark. Press Enter to complete the command. Now all four walls should have this particular connection with the floor slab, including the skirting board.

Figure 2.389

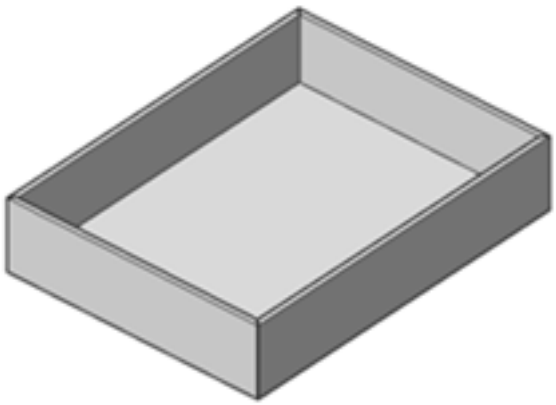


Figure 2.390

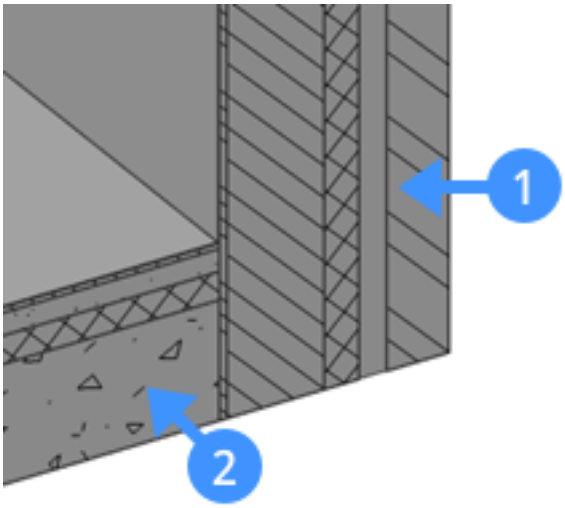


Figure 2.391

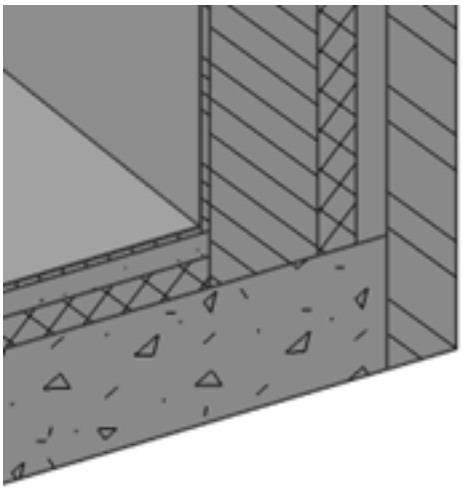


Figure 2.392

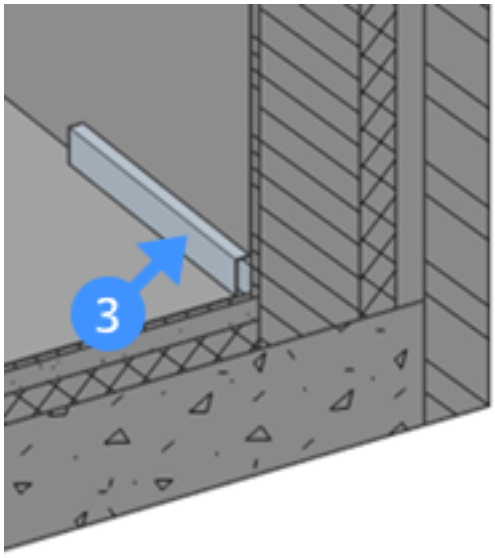


Figure 2.393

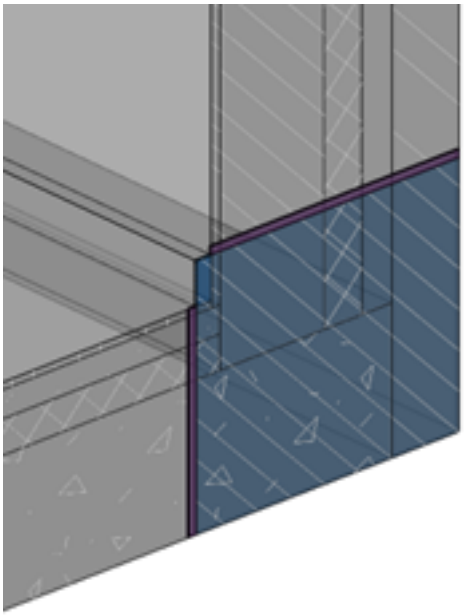
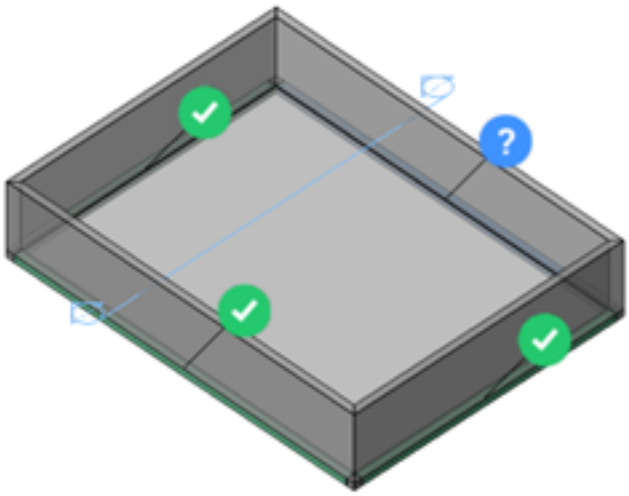


Figure 2.394



2.9 Project Collaboration

2.9.1 How to do clash detection in BricsCAD BIM

https://www.youtube-nocookie.com/embed/XmCQ8VOLcnQ This tutorial shows how the Interfere command can be used for clash detection between solids. Use the Structure browser and Layer panel for easy navigation between the collisions. Back to Tutorial Overview

2.9.2 How to import .RFA files

https://www.youtube-nocookie.com/embed/4Kff1Rzf5zA This video shows how to import .RFA files, either individually or by adding the folder location to your 'components path' so it shows up in your Components Panel. Back to Tutorial Overview

2.9.3 How to import & export IFC files in BricsCAD BIM

https://www.youtube-nocookie.com/embed/6Q1ABXp1w2Q This tutorial shows how you can easily Import from and Export to IFC files to be able to communicate with other BIM software packages. Back to Tutorial Overview

2.10 Project Collaboration

2.10.1 Connect your BIM project with Bricsys 24/7

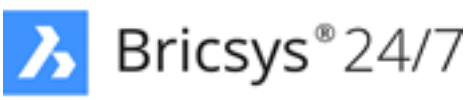
Command: CloudProject (6.9.30), CloudDownload (6.9.27), CloudAccount (6.9.26) **Overview:** In this lesson, you learn how to connect your BIM project with the cloud using Bricsys 24/7. **Lesson Objectives** After completing this lesson, you will be able to:

- Use the direct connection to Bricsys 24/7 inside BricsCAD BIM.
- Launch your Bricsys 24/7 project in a browser.
- Upload the current BIM models to Bricsys 24/7.
- Download and open a BIM model from Bricsys 24/7.

About Bricsys 24/7

The cloud-based service helps the team members to find and share their drawings faster than the conventional methods. BricsCAD BIM performs the project collaboration via Bricsys 24/7, which is a cloud collaboration tool. 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible. A big advantage of this cloud-based service is that BricsCAD is directly linked to it by multiple commands. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/Bricsys247.pr

Figure 2.395



Procedure:Using Bricsys 24/7 in BricsCAD BIM

1. Check the current status of your Bricsys 24/7 account using the command bar, simply type CLOUDACCOUNT.

Reports in the command bar display. If you are logged into your 24/7 account, you will see output similar to what is shown below: Server: http://my.bricsys247.com User name: your.name@email.com If you are logged out of your 24/7 account, you will see output similar to what is shown below: Bricsys 24/7: not logged in.

2. To log on to Bricsys 24/7, use the command of CLOUDLOGON.

The Bricsys 24/7 Logon dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Proj
Enter your E-mail/ Username and Password in the corresponding fields, then press the **OK** button. Optionally check the **Remember** me box to save your login information for your next visit.

3. To log off to Bricsys 24/7, use the command of CLOUDLOGOFF.

The command prompts to display: Logged off.

4. To open your Bricsys 24/7 account in a browser, type CLOUDPROJECT in the command line.

The Log on page of Bricsys 24/7 in your default web browser displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/

5. To download a BIM model from your Bricsys 24/7 project to a local folder, type CLOUDDOWNLOAD in the command line.

Note: If you are not yet logged on to the Cloud, the Cloud Logon window will display first. If you are already logged in the Download from Bricsys 24/7 dialog box will display. The following illustration shows the Download from Bricsys 24/7 dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/Clouddow

View on Bricsys 24/7 (1)	Shows the selected document in the Cloud Viewer.
Refresh projects (2)	Reloads the project list.
Download (3)	Downloads the selected document to a local folder.
Cancel (4)	Cancels files downloading from 24/7.

6. To upload your current model to Bricsys 24/7, type CLOUDUPLOAD in the command line.

Note: If you are not yet logged on to the Cloud, the Cloud Logon window will display first. If you are already logged in the Upload to Bricsys 24/7 dialog box will display. The following illustration shows the Upload to Bricsys 24/7 dialog box. The table below presents each component of the dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

New folder (5)	Creates a new folder as a subfolder of the currently selected folder.
Refresh projects (6)	Reloads the project list.
Upload (7)	Uploads the drawing to the selected folder. If a drawing of the same name already exists in the selected folder, the drawing is uploaded as a revision of the original drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollal If the drawing is dependent from other files, such as external references or images, the**CloudUploadDependencies**user preference defines what to do: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollal 0: Upload the drawing only and ignore dependencies. 1: Launch theeTransmitcommand first to compose a zip-file if the drawing contains external references. 2: Always launcheTransmitfirst.
Cancel (8)	Cancels files uploading on 24/7.

7. To download and open a drawing from Bricsys 24/7, type CLOUDOPEN in the command line.

Figure 2.396

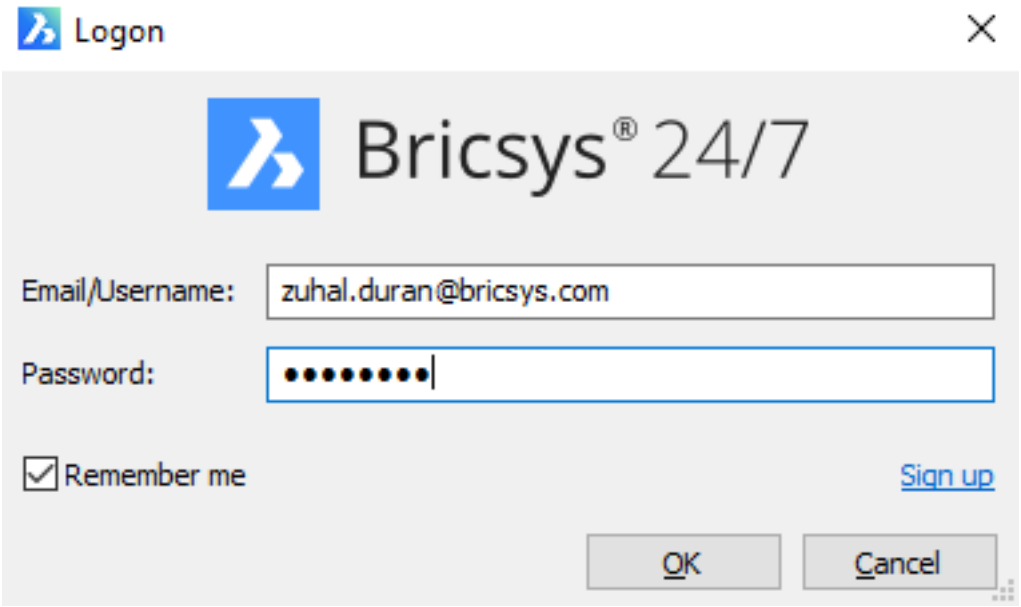


Figure 2.397

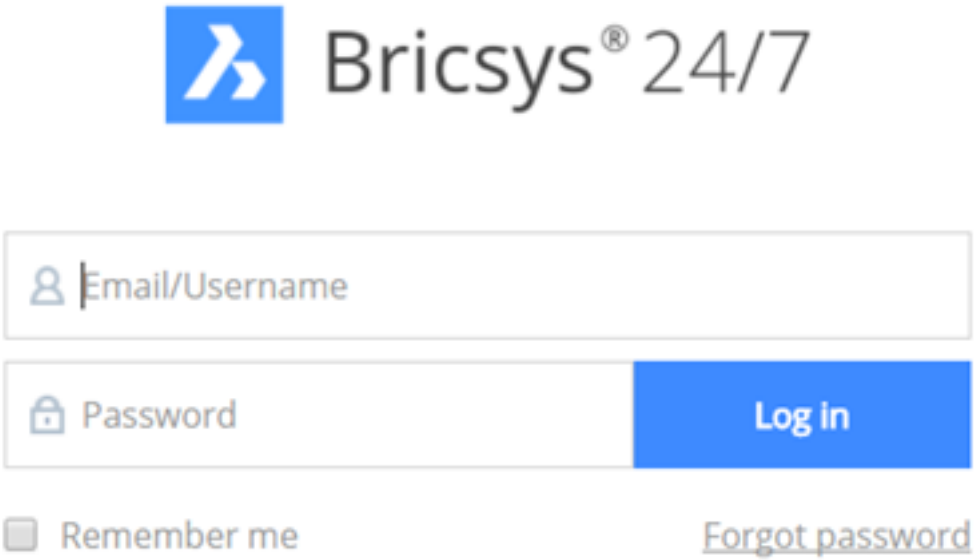


Figure 2.398

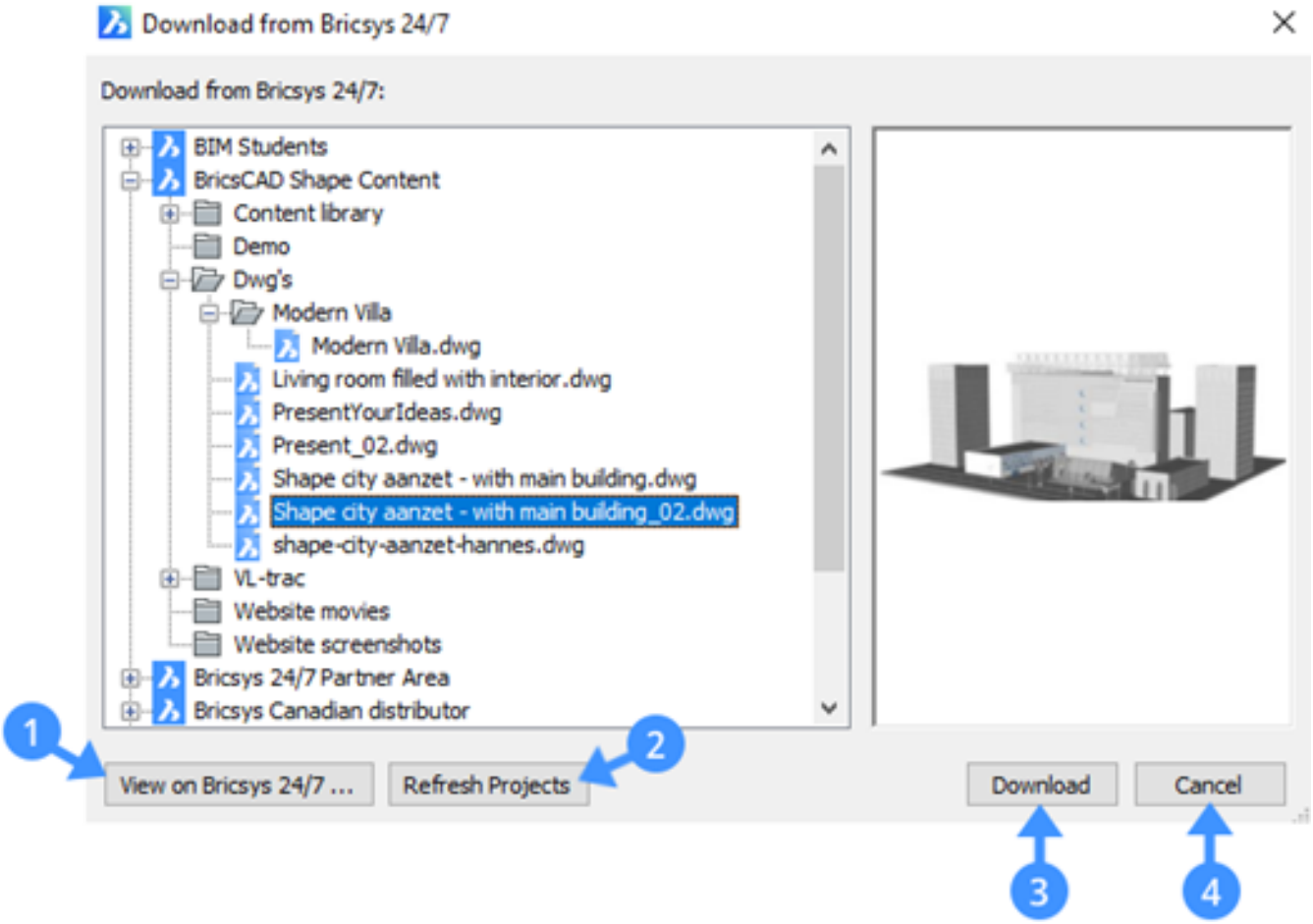


Figure 2.399

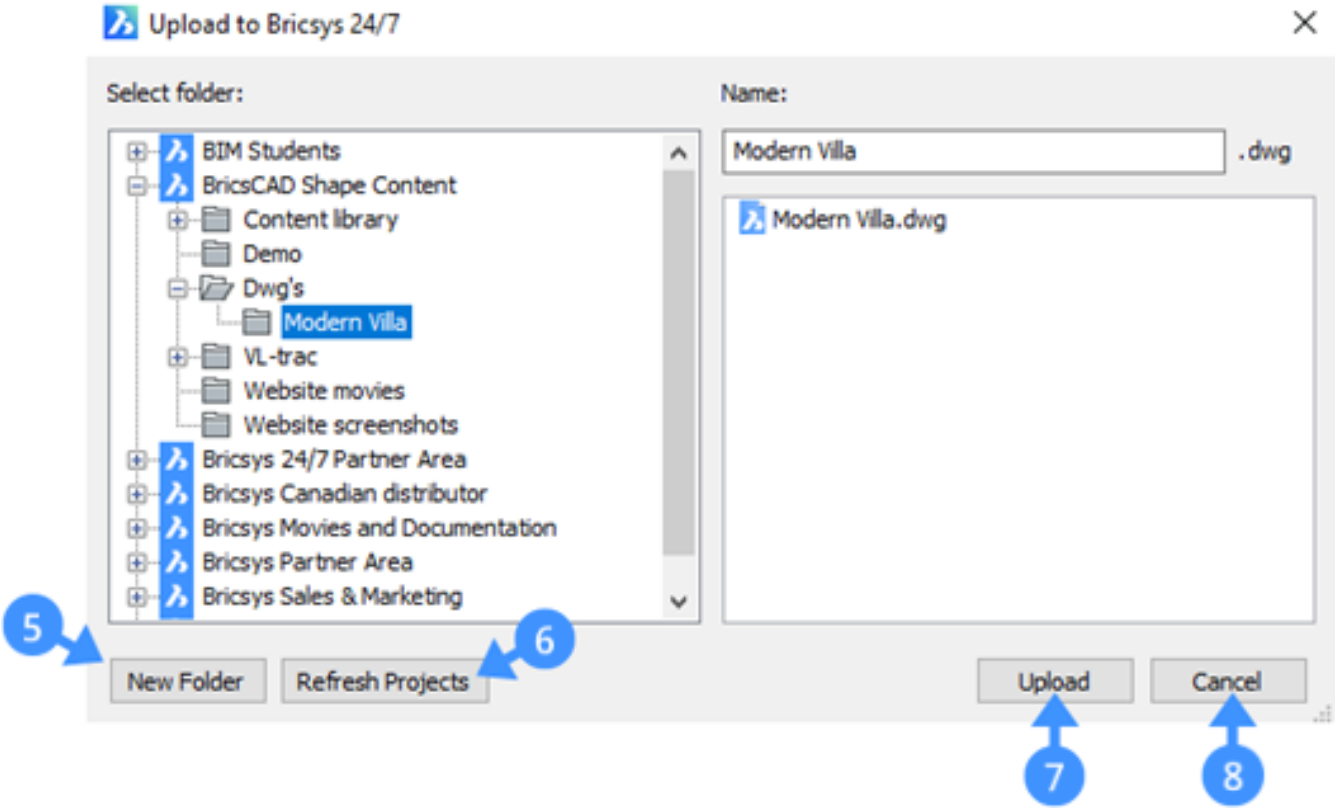
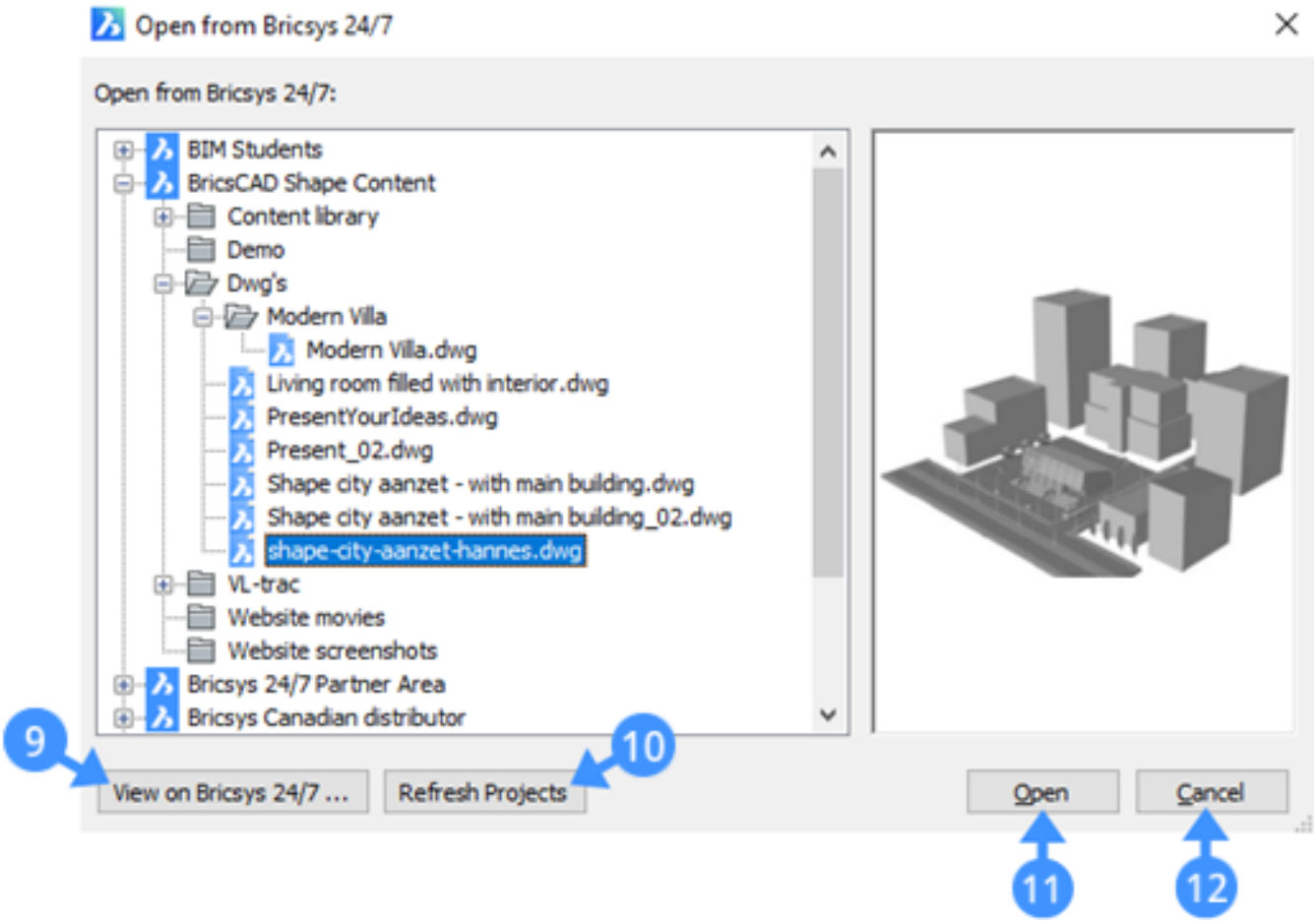


Figure 2.400



Note: If you are not yet logged on to the Cloud, the Cloud Logon window will display first. If you are already logged in the Open from Bricsys 24/7 dialog box will display. The following illustration shows the Open from Bricsys 24/7 dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/CloudopenDialogB

View on Bricsys 24/7 (9)	Shows the selected document in the Cloud Viewer.
Refresh projects (10)	Reloads the project list.
Open (11)	Downloads and opens the selected drawing. A copy of the drawing is saved in the subfolder of the folder which is defined intheCloudTempFolderuser preference (e.g. C:\Users\<UserName>\AppData\Local\Temp\Bricsys_24_7\). When saving the drawing, a dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollal **Upload**: Uploads the drawing to Bricsys 24/7 as a revision of the original drawing. **SaveLocally**: Saves a copy of the drawing locally. If the selected drawing is linked to one or more other drawings, a dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollal Do one of the following: Click theYesbutton and select the linked documents you want to download. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollal Click the **No** button to conclude the command. The drawing is not downloaded.
Cancel (12)	Cancels files uploading on 24/7.

Figure 2.401

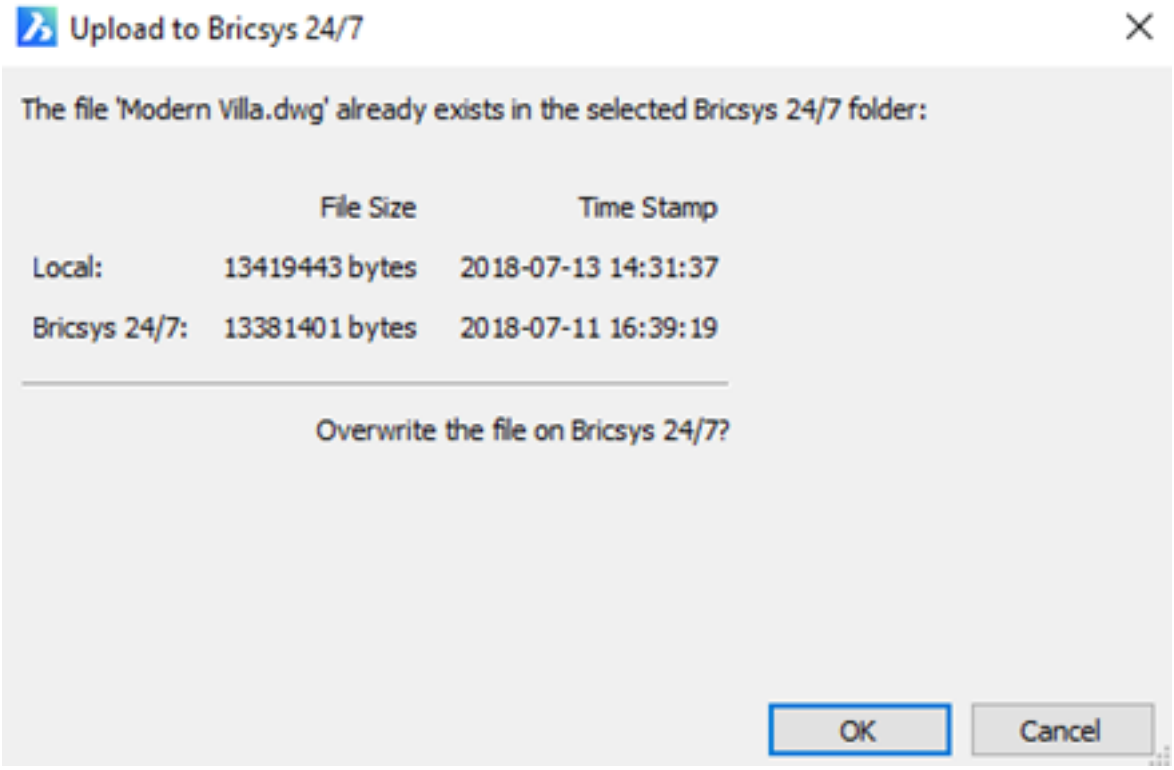


Figure 2.402

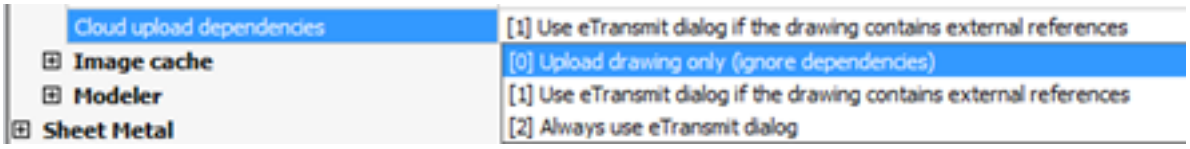


Figure 2.403

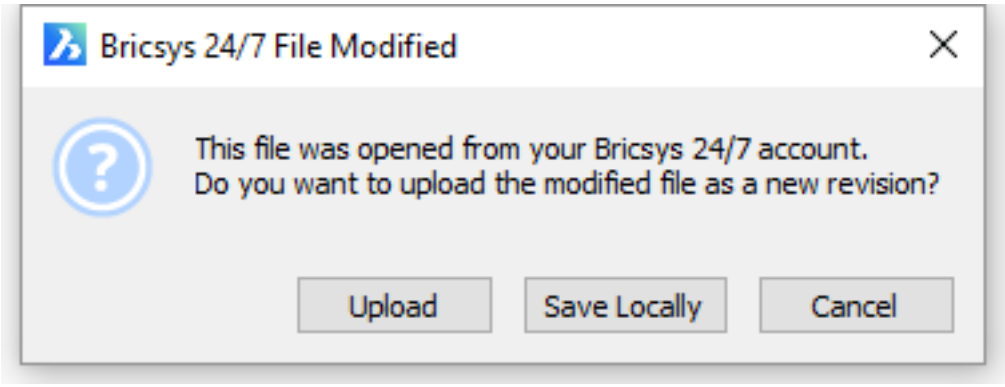


Figure 2.404

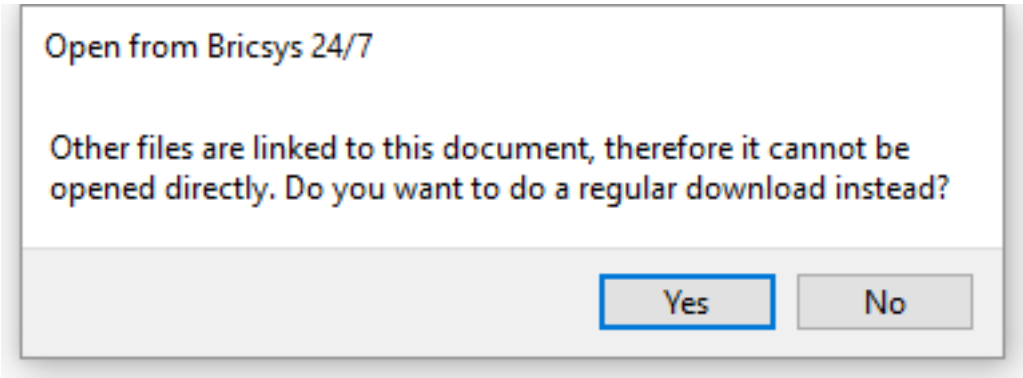
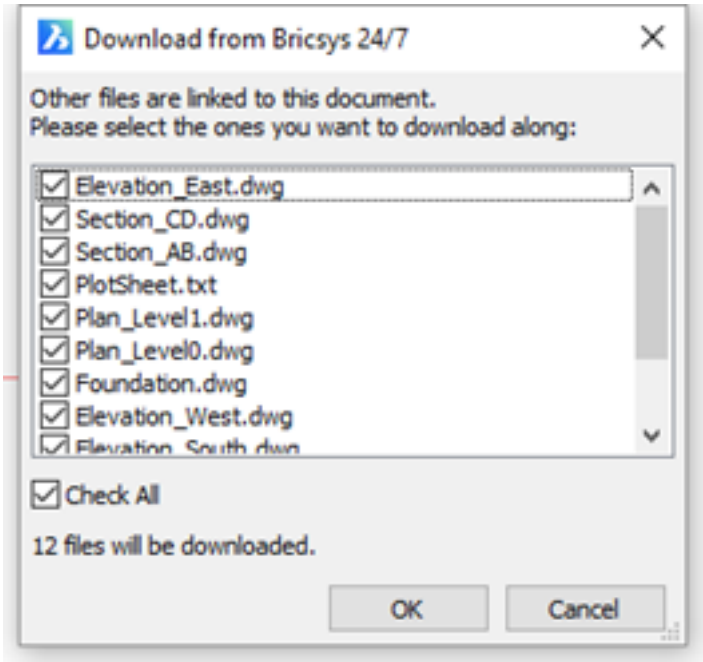


Figure 2.405


Download from Bricsys 24/7
Other files are linked to this document.
Please select the ones you want to download along:
Elevation_East.dwg
Section_CD.dwg
Section_AB.dwg
PlotSheet.txt
Plan_Level1.dwg
Plan_Level0.dwg
Foundation.dwg
Elevation_West.dwg
Check All
12 files will be downloaded.
OK
Cancel

2.10.2 IFC Import and Export in BricsCAD

Commands: IMPORT (6.17.10),IFCEXPORT (6.17.2),EXPORT (6.13.15) **Overview:** In this lesson, you learn how to import and export IFC files in BricsCAD BIM. **Lesson Objectives** After completing this lesson, you will be able to:

- Import an IFC model to BricsCAD.
- Export an IFC model from BricsCAD.

Tutorial: How to import & export IFC files in BricsCAD BIM Click here (2.9.3) to watch

About IFC

TheIndustry Foundation Classes (IFC)data model is designed to describe building and construction industry information. IFC is a platform neutral data exchange format. The goal of IFC is to provide interoperability between different BIM software platforms. BIM authoring tools like BricsCAD BIM can export a building model to an IFC file, allowing interoperability between IFC-compliant software applications. BricsCAD BIM is certified for IFC 2X3 coordination view 2.0. The IFC standard includes both parametric and non-parametric shapes, material definitions, metadata in the BIM file, and property sets (PSets) that can be assigned to the diverse BIM types available in IFC (e.g. PSet_WallCommon). When you export an IFC file from BricsCAD the BricsCAD properties will be exported as IfcPropertySet. These property sets contain common IFC properties. If you have any user-defined property sets in your BIM model, these properties can also be exported. Mechanical system layouts, structural models, and architectural models that are exported from different IFC compliant software applications can be imported into BricsCAD BIM. ===ERROR WITH DOWNLOADED IMAGE=== **Keyboard:** IMPORT, EXPORT, IFCEXPORT **Menubar:** File > Import, Export

Procedure:Importing an IFC file in BricsCAD

1. To import an IFC file in your project, click the **File** menu, and then select **Import**

The import file dialog box displays.

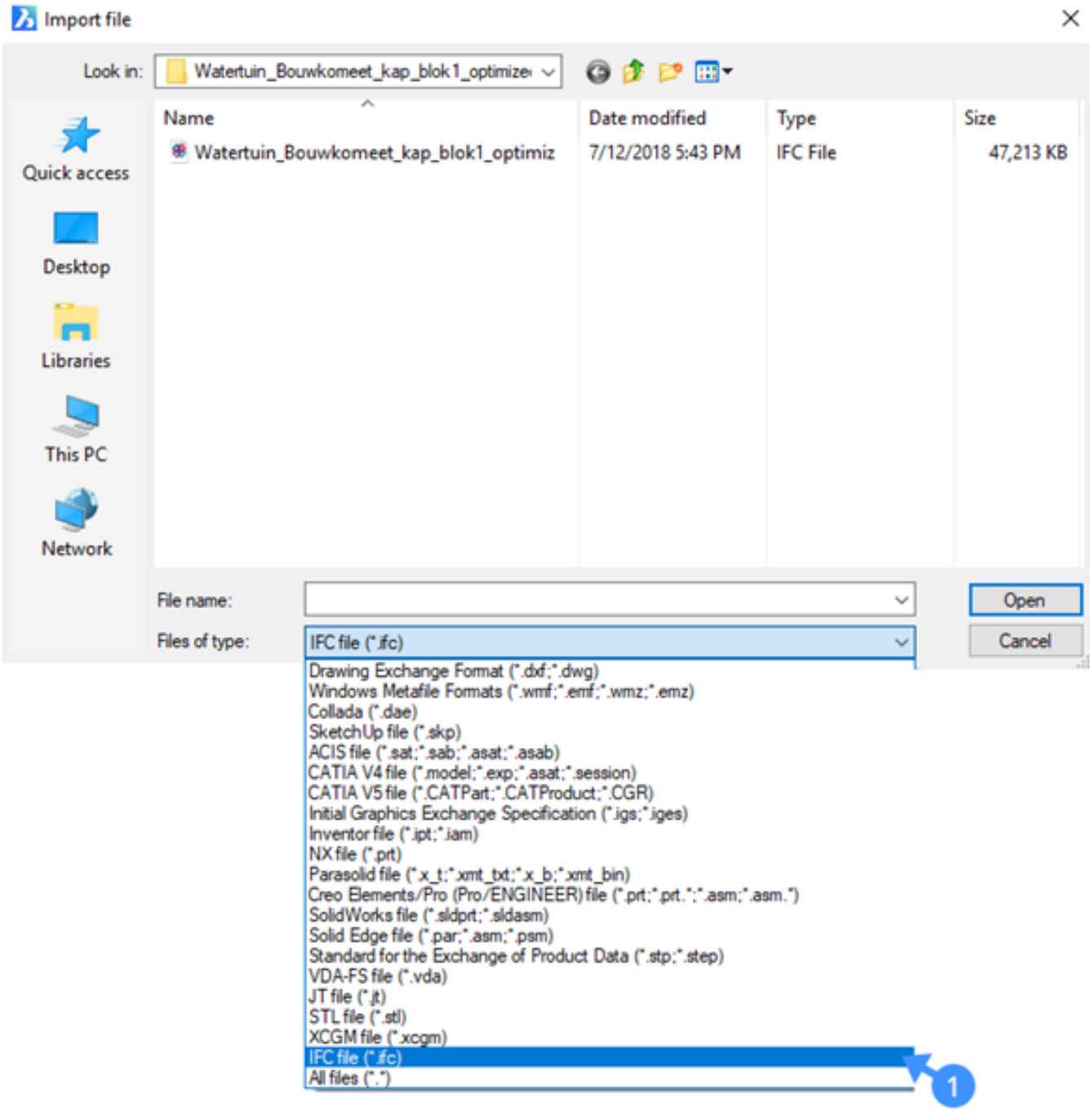
2. Select the .ifc file format (1) in the **Files Of Type** list and choose the IFC model you want to import.

The following illustration shows the import file dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V

3. Click **Open** to import the IFC model.

When the IFC file is imported the data from the imported file integrates with the BricsCAD data structure which means you can work with them right away. If the imported data contain custom properties these property sets will be available for the related entities. **Note:** You have the ability to create custom property sets and properties directly in BricsCAD BIM and associate them with imported entities.

Figure 2.406


Import file
Look in:
Watertuin_Bouwkomeet_kap_blok1_optimize
Quick access
Desktop
Libraries
This PC
Network
Name
Date modified
Type
Size
Watertuin_Bouwkomeet_kap_blok1_optimiz
7/12/2018 5:43 PM
IFC File
47,213 KB
File name:
Files of type:
IFC file (*.ifc)
Open
Cancel
Drawing Exchange Format (*.dxf;*.dwg)
Windows Metafile Formats (*.wmf;*.emf;*.wmz;*.emz)
Collada (*.dae)
SketchUp file (*.skp)
ACIS file (*.sat;*.sab;*.asat;*.asab)
CATIA V4 file (*.model;*.exp;*.asat;*.session)
CATIA V5 file (*.CATPart;*.CATProduct;*.CGR)
Initial Graphics Exchange Specification (*.igs;*.iges)
Inventor file (*.ipt;*.iam)
NX file (*.prt)
Parasolid file (*.x_t;*.xmt_txt;*.x_b;*.xmt_bin)
Creo Elements/Pro (Pro/ENGINEER) file (*.prt;*.prt.*;*.asm;*.asm.*)
SolidWorks file (*.sldprt;*.sldasm)
Solid Edge file (*.par;*.asm;*.psm)
Standard for the Exchange of Product Data (*.stp;*.step)
VDA-FS file (*.vda)
JT file (*.jt)
STL file (*.stl)
XCGM file (*.xcgm)
IFC file (*.ifc)
All files (*.*)
1

Procedure: Exporting an IFC file in BricsCAD

1. To export an IFC file in your project, click the **File** menu, and then select **Export**

The **Export Drawing As** dialog box displays.

2. Select the .ifc file format (2) in the **Save as type** list and save the IFC model in a folder where you want to store.

The following illustration shows the Export Drawing As dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help

3. Click **Save** to export the IFC model.

The exported BIM model is saved as an IFC file and it is ready to use in any IFC certified applications.

4. (Optional) Change the IFC import and export settings using the BIM section in the Settings dialog.

The following illustration shows the IFC import and export settings in the settings dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/IFCImportAndExportSettings.png}) **IFC setting preferences:**

Import Spaces	Enable/disable importing IfcSpace entities as BIM Rooms. If this setting is enabled, IfcSpace entities will be imported as BIM Room entities. By default, this setting is enabled.
Import BIM Data	Enable/disable importing BIM data from different BIM platforms to BricsCAD BIM. By default, this setting is enabled.
Import model position	Controls the insertion point of the imported model. The options are: • **0**: BricsCAD WCS matches the IFC Global Coordinate system. • **1**: BricsCAD WCS matched the IFC Site Location.
Import Parametric Components	Enable/disable importing windows and doors as parametric components.
Import IFC project structure as xrefs	Enable/disable importing IFC project structures as separate XRef drawings. If this setting is enabled, drawing files are created as XRefs. These files correspond to each building and each story of the IFC project structure.
Import BREP geometry as meshes	Enable/disable importing BREP geometry as meshes. If this setting is enabled, BREPs in the drawing is converted into meshes.
Export base quantities	Enable/disable exporting derived base quantities from BIM elements. If this setting is enabled , calculated quantities (e.g. length, thickness, height) for each BIM element can be exported to IFC.
Export elements on OFF and FROZEN layers	Enable/disable exporting BIM elements that are in OFF and Frozen layers. By default, this setting is enabled.
Explode external references in IFC spatial structure	Enable/disable exploding external references in IFC spatial structures. If this setting is enabled , each external reference will be divided into the elements (e.g.**IfcSite, IfcBuilding)** based on the spatial project structure.
Export multi-ply elements as aggregated elements	Enable/disable exporting building elements with multi-ply compositions as aggregated entities. If this setting is enabled , each ply material belonging to a building element (e.g. wall, column) is calculated and exported individually.

Level of tessellation	Controls the level of tesselation of exported geometry. If the 'Current' faceting option is chosen, the faceting as set by FacetRes or Modeler properties is used. Choosing the 'Low', 'Medium', 'High' options causes according regenerations of facets which makes the process longer.

Figure 2.407

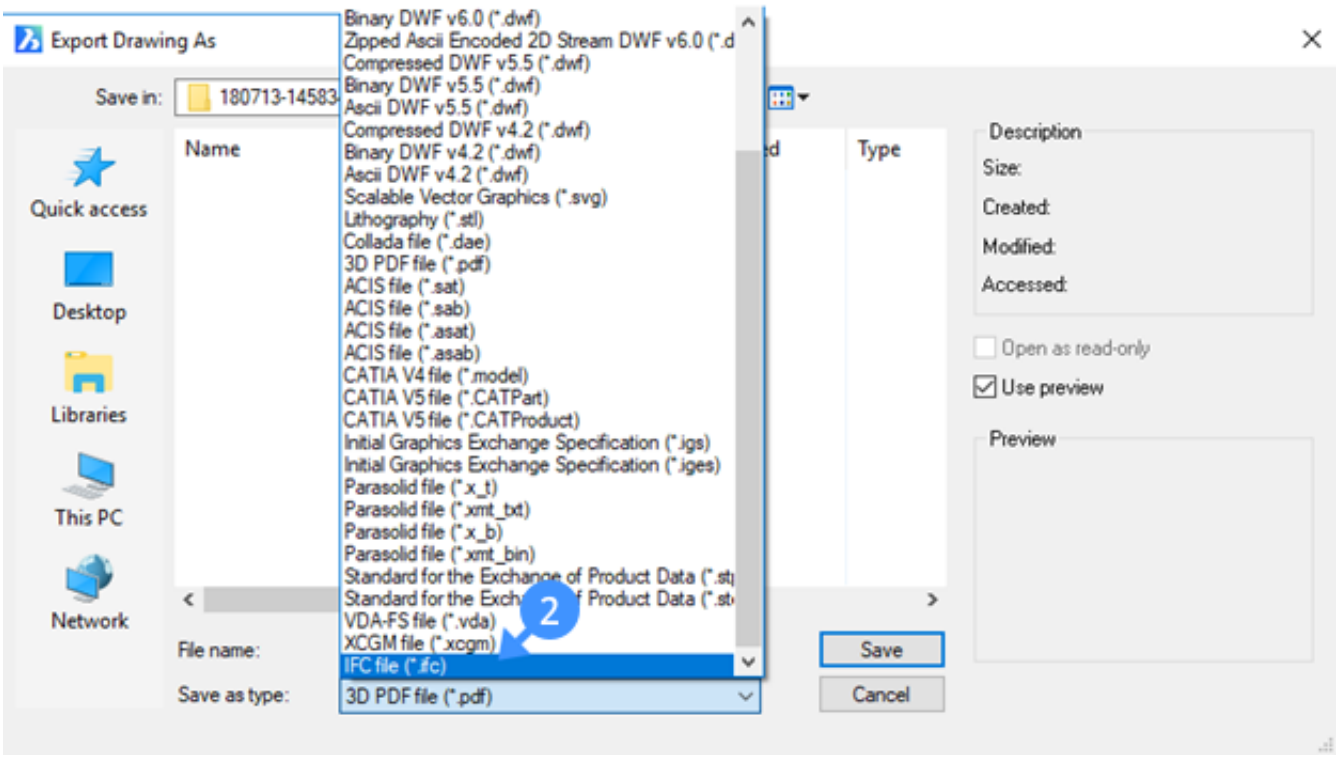
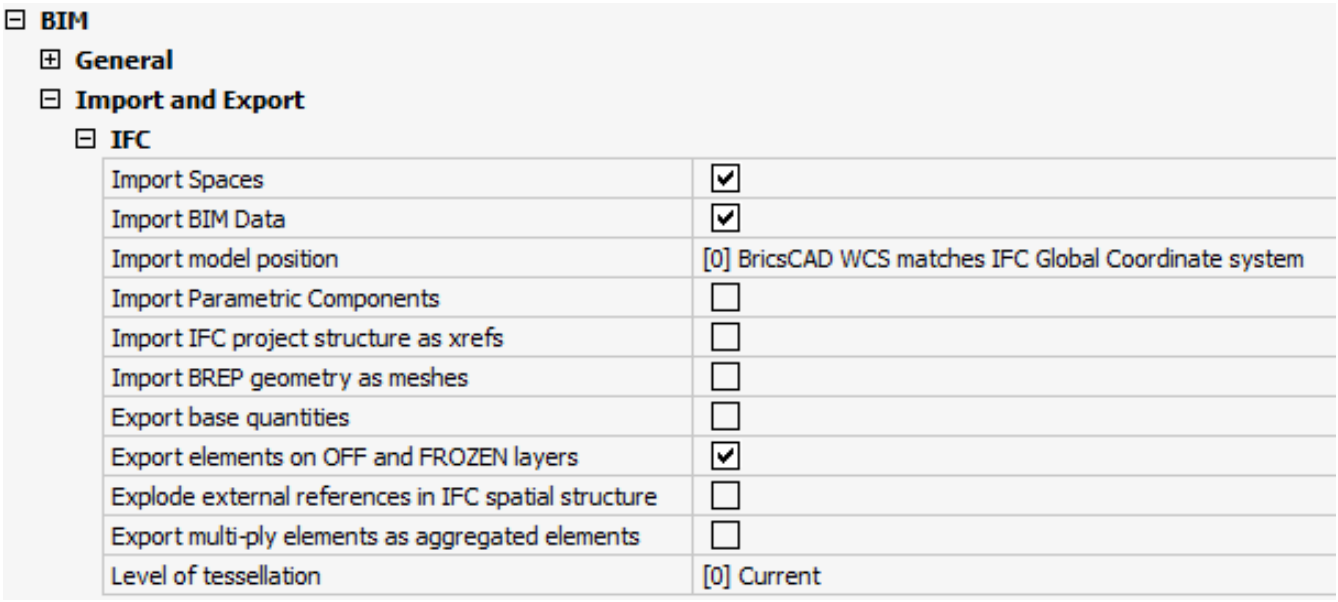


Figure 2.408



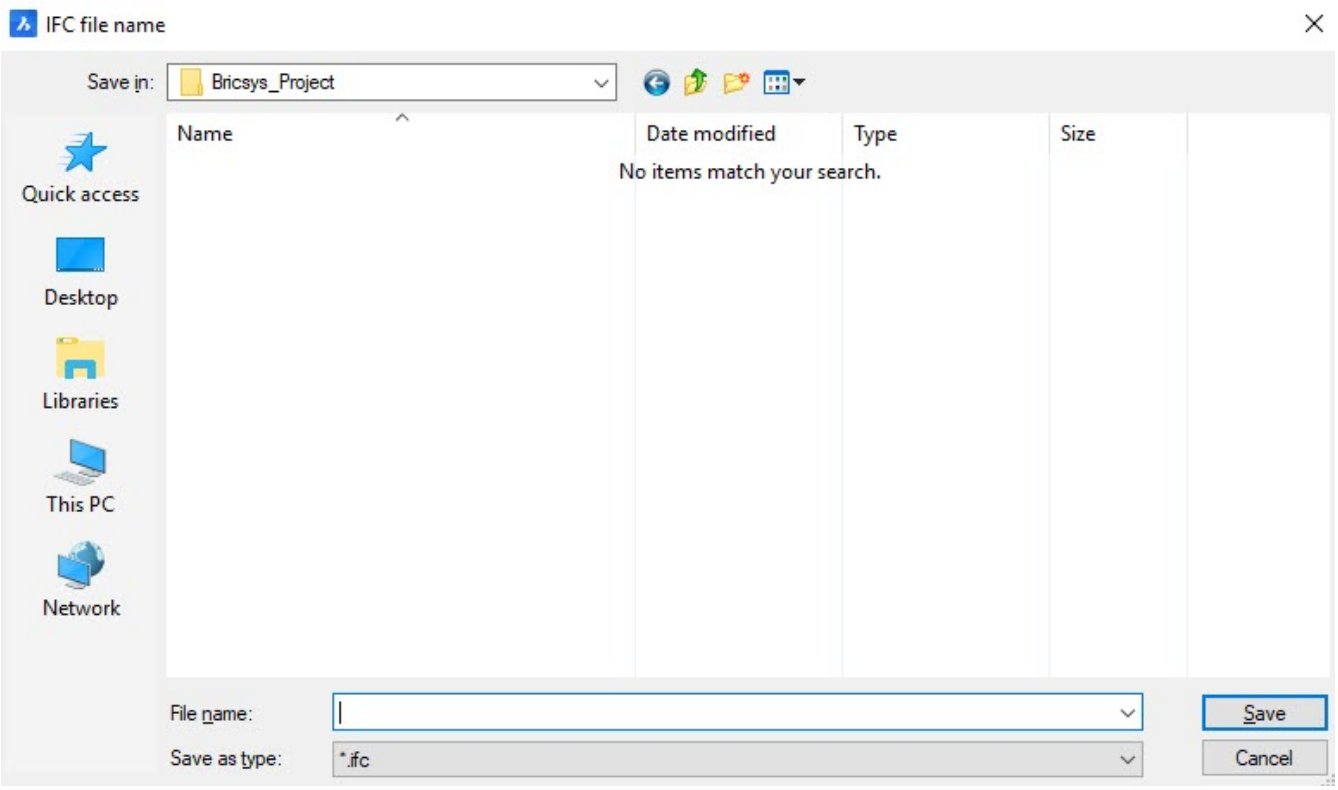
Procedure: Exporting an IFC file only for selected entities

1. To export IFC model partially in your project, type **IFCEXPORT** in the command line; You are prompted: **Select entities to export [Entire model] <Entire model>: (Select the entities in your model space to export as IFC)** After selecting the entities, press Enter. The **IFC file name**dialog box displays.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/IFCfilenameDialogBox.jpg}

2. Enter a name for the IFC file and click the **Save** button to save the entities as IFC. The selected entities exported to IFC file.

Figure 2.409


IFC file name
Save in:
Bricsys_Project
Quick access
Desktop
Libraries
This PC
Network
Name
Date modified
Type
Size
No items match your search.
File name:
Save as type:
*.ifc
Save
Cancel

2.10.3 Importing Revit RFA Families in BricsCAD

Command: IMPORT (6.17.10) **Overview:** In this lesson, you learn how to import Revit family files in BricsCAD BIM. **Lesson Objectives** After completing this lesson, you will be able to:

- Import a Revit RFA family file to BricsCAD.
- Use BimInsert (6.7.14) command to insert a RFA file as a BIM component.
- Insert RFA files from the components panel.

Tutorial: How to import .RFA files Click here (2.9.2) to watch

General information before getting started

- Revit family files created in versions before 2011 cannot be imported.
- Currently, families can only be imported with their geometry. The parameters and properties are attached to the families cannot be imported.
- Revit version 2019 family files are currently not compatible with BricsCAD.

Procedure: Importing RFA family files

1. To import an RFA family in your project, click the**File**menu, and then select**Import.**

The import file dialog box displays.

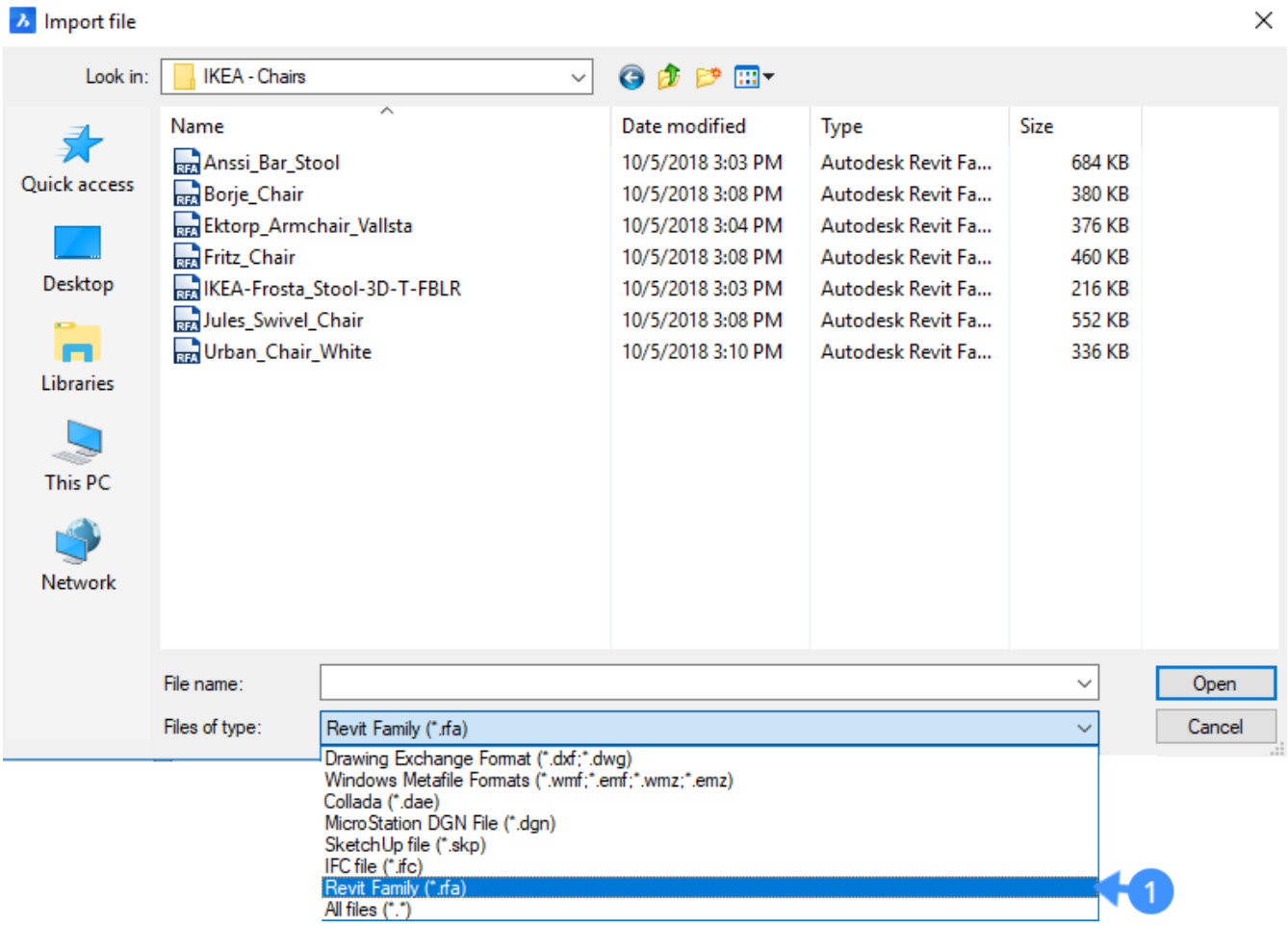
2. Select the .rfa file format (1) in the**Files Of Type**list and choose the .rfa file you want to import.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/RFAImportDialogBox.png}

3. Click**Open**to import the Revit Family.

When the RFA family file is imported the geometry from this file is included in BricsCAD as separate 3D solids.

Figure 2.410


Import file
Look in:
IKEA - Chairs
Quick access
Desktop
Libraries
This PC
Network
Name
Date modified
Type
Size
Anssi_Bar_Stool 10/5/2018 3:03 PM Autodesk Revit Fa... 684 KB
Borje_Chair 10/5/2018 3:08 PM Autodesk Revit Fa... 380 KB
Ektorp_Armchair_Vallsta 10/5/2018 3:04 PM Autodesk Revit Fa... 376 KB
Fritz_Chair 10/5/2018 3:08 PM Autodesk Revit Fa... 460 KB
IKEA-Frosta_Stool-3D-T-FBLR 10/5/2018 3:03 PM Autodesk Revit Fa... 216 KB
Jules_Swivel_Chair 10/5/2018 3:08 PM Autodesk Revit Fa... 552 KB
Urban_Chair_White 10/5/2018 3:10 PM Autodesk Revit Fa... 336 KB
File name:
Files of type:
Revit Family (*.rfa)
Open
Cancel
Drawing Exchange Format (*.dxf;*.dwg)
Windows Metafile Formats (*.wmf;*.emf;*.wmz;*.emz)
Collada (*.dae)
MicroStation DGN File (*.dgn)
SketchUp file (*.skp)
IFC file (*.ifc)
Revit Family (*.rfa)
All files (*.*)
1

Procedure: Inserting RFA family files with BIM Insert

Note: It is recommended to haveDynamic Dimensions (1.15.9)(DYN) active, which allows to type the distance in the dynamic entry field. If Dynamic UCS (DUCS (1.12.2))is ON, the section plane aligns to the face of a 3D solid under the cursor.

1. Select**BIM Insert**from the No Selection Quad.

The file explorer opens.By default, it is set to the .dwg file format and takes you to the library of predefined doors and windows.Browse to the folder where you place your .rfa family files.

2. Select the .rfa file format (2) in the**Files Of Type**list and choose the .rfa file you want to import.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/RFASelectfiletoinsertDialo
The selected file is attached to the cursor (3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Pro
Note: If this family object happens to the face of another solid, it will align with the face under the cursor. **Note:**The dynamic dimension shows the distances from the nearest parallel edge on the face of the 3D solid. The following illustration shows inserting a Revit family sofa when the DYN is on. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

3. Do the following to place the selected RFA family file using the dynamic dimension fields:

- Hit the TAB key to select one of the dynamic dimension fields, then type a value and hit the TAB key to lock the dimension. The locked dimension will turn red.

4. Press Enter to accept the insertion.

Figure 2.411

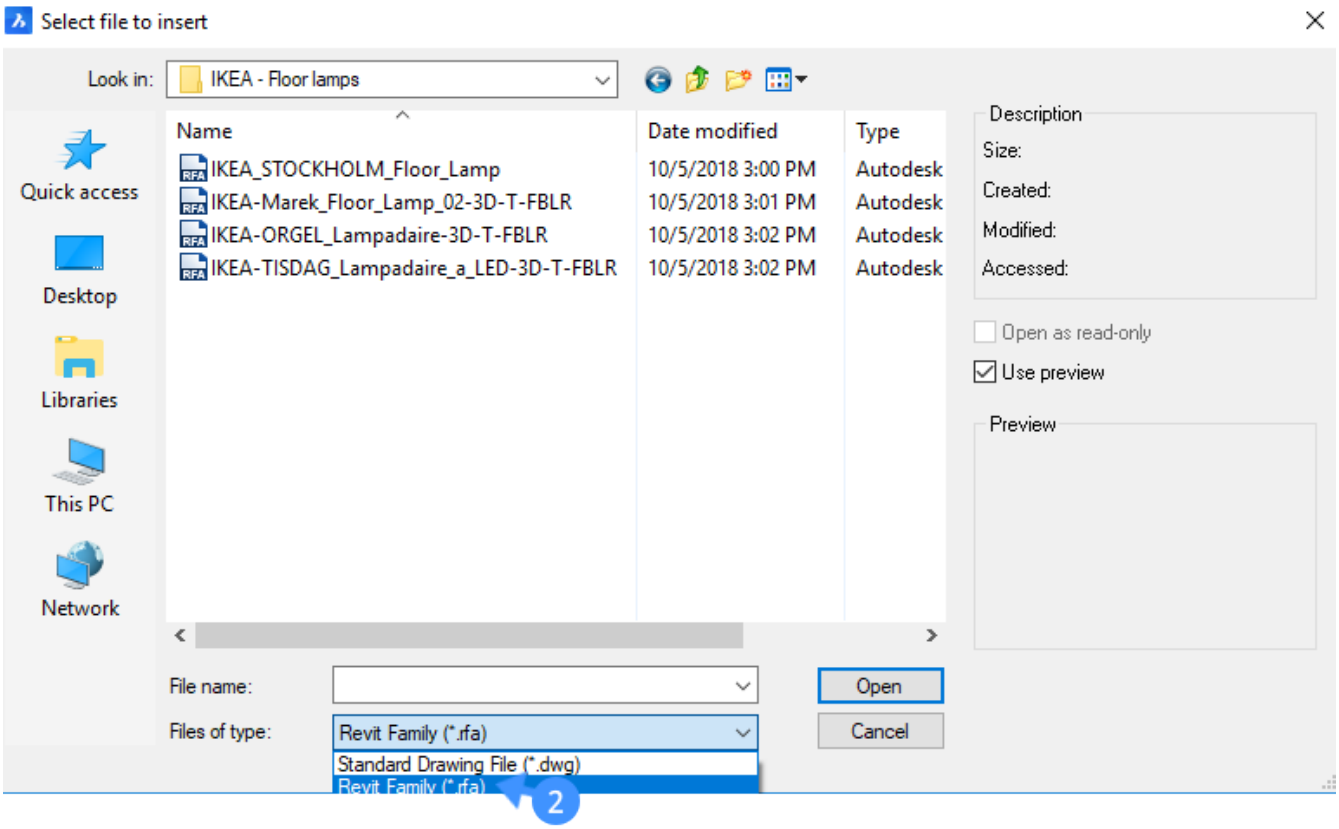


Figure 2.412

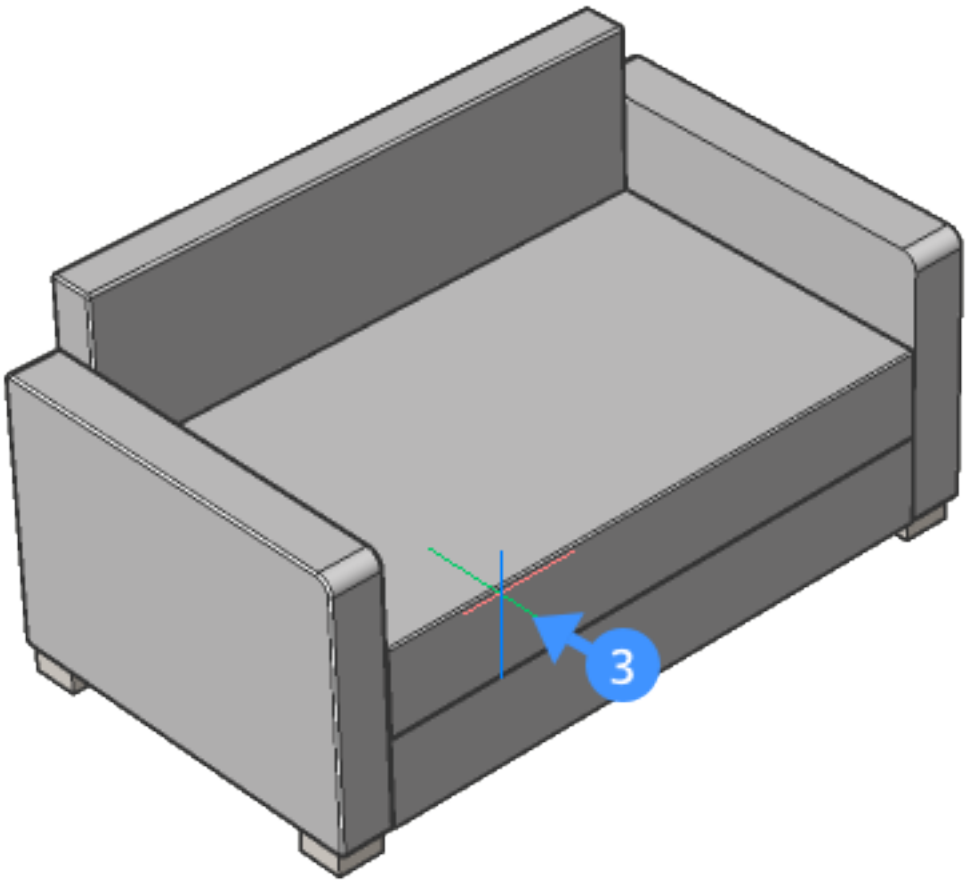
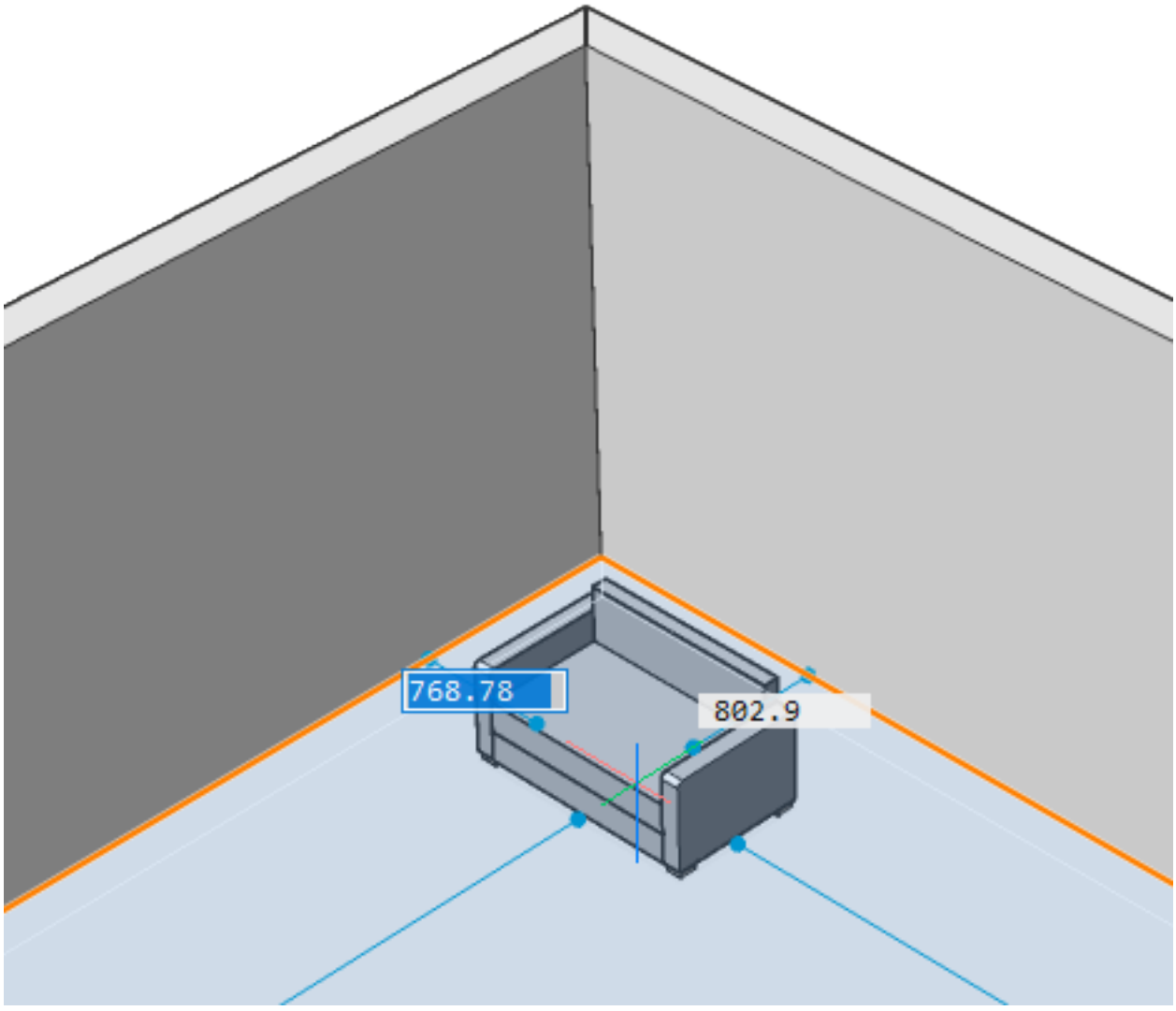


Figure 2.413



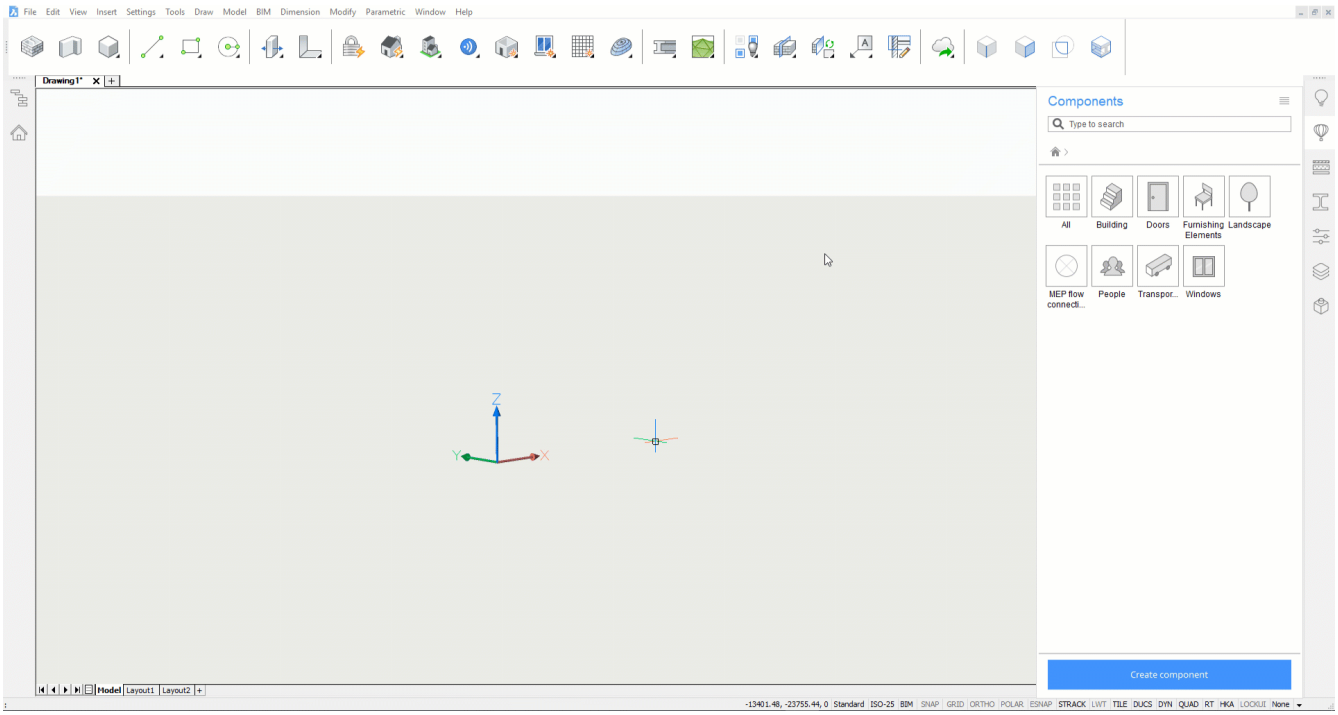
Procedure: Inserting RFA family files from the components panel

1. To insert an RFA family folder in the components panel, add the RFA root folder path to the components path in the settings. To do so, open the Settings (1.13.11) dialog box and place your RFA folder in COMPONENTSPATH. In this way, all folders inside the root folder will be categorized as components in the panel.

The following animated gif shows how to add the RFA root folder path to the components directory path. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/RFAInsertComponentsPanel.png
The RFA family files are ready to insert in the model space.

2. Drag and drop the RFA file from the components panel into your model space.

Figure 2.414


File Edit View Insert Settings Tools Draw Model BIM Dimension Modify Parametric Window Help
Drawing1*
Components
Type to search
All
Building
Doors
Furnishing Elements
Landscape
MEP flow connecti...
People
Transpor...
Windows
Create component
Model
Layout1
Layout2

2.10.4 Interference Checking

Command: Interfere (6.17.18) **Overview:** In this lesson, you learn how to do clash detection between solid models in BricsCAD BIM. **Lesson Objectives** After completing this lesson, you will be able to:

- Use the interference check tool on your existing model.
- Identify where 3D solids intersect each other.
- Remove undesired intersections among the two solid models.

Tutorial: How to do clash detection in BricsCAD BIM Click here (2.9.1) to watch

About Interference Checking

The Interference Checking tool is used during the modeling phase to find out conflicts in the model by comparing 3D solid models of building elements. Interference occurs when two 3D entities overlap in 3D space. In BricsCAD BIM, the Interfere tool helps you to find intersections between the entities that you have selected. You can also do the interference checking for all elements in your model at a time. Optionally, the tool creates interference solids after running the command. The newly created solids are placed on their own layer, as specified by the InterferenceLayer variable. Running an interference check on existing models allows you to prevent the conflicts in your entire model.

 **Keyboard:** Interfere **Menubar:** Model > 3D Solids > Interfere **Toolbar:** 3D Solids > Interfere **Quad:** Model > Interfere

Procedure:Performing an interference check

1. Select the desired 3D solids to be checked for interference, then launch the INTERFERE command.

Note: Multiple solids can be selected using the selection box or in the Structure Browser, depending on whether the selected entities appear when selected in the tree. The first set of entities is defined. The command line displays how many entities in total are set for interference checking e.g. Entities in set: 20. Note that **Nested Selection** option allows you to select 3D solid models that are inside blocks and external references (XRef) and once they are selected you can compare them against other entities in the selection set.

2. Select the 3D solids in the model to add them to the second set of entities.

The entities in the first set are compared with the entities in the second set. **Note**: If you Press **Enter** key without selecting any solids for defining the second set of entities, the entities in the first set will be compared to each other.

3. Press **Enter** to run the interference check.
4. The interference volume is created for each collision and it is shown in a solid model.

Note: The solids of the interference volume are displayed in red since they are placed on layer **Interference.** The default settings on the Interference Checking tool makes it easy to find the intersecting solids in the model. This setting can be changed using the prompt menu or the command line. To open it, click the Settings option in the prompt menu or type “S” while the interfere command is still active. The Interference Settings dialog box displays. The dialog box allows you to configure interference settings, which are stored in variables. The following illustration shows the Interference Settings dialog box. The table below presents each component of the dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/InterferenceSettingsDialogBox.p

Create Interference Solids box (1)	Determines whether a solid or region of the interference is created. Check to create a solid of the interference volume or region of the interference area.
Layer (2)	Shows the current layer for interferences.
Color (3)	Shows the current layer color.

The following illustration shows the detected collision (4) between the floor slab and column. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/DetectedCollision.png})

5. To remove the undesired intersections, turn off the Interference layer, and perform one of the 3D solid modifying tools e.g. **BIM Drag** or **Subtract** on the interfering solid. This process will let you make the necessary corrections in your model.

Figure 2.415

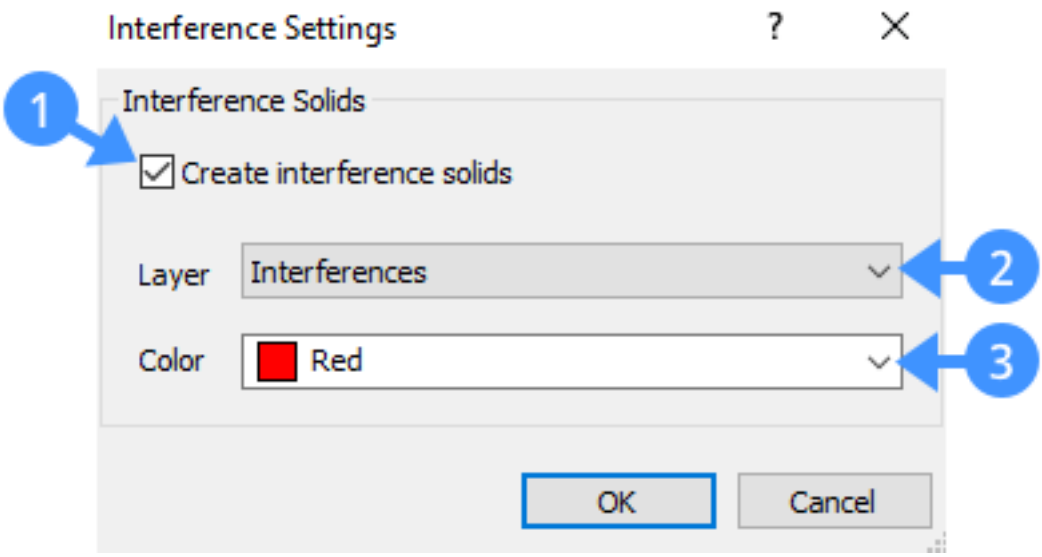


Figure 2.416

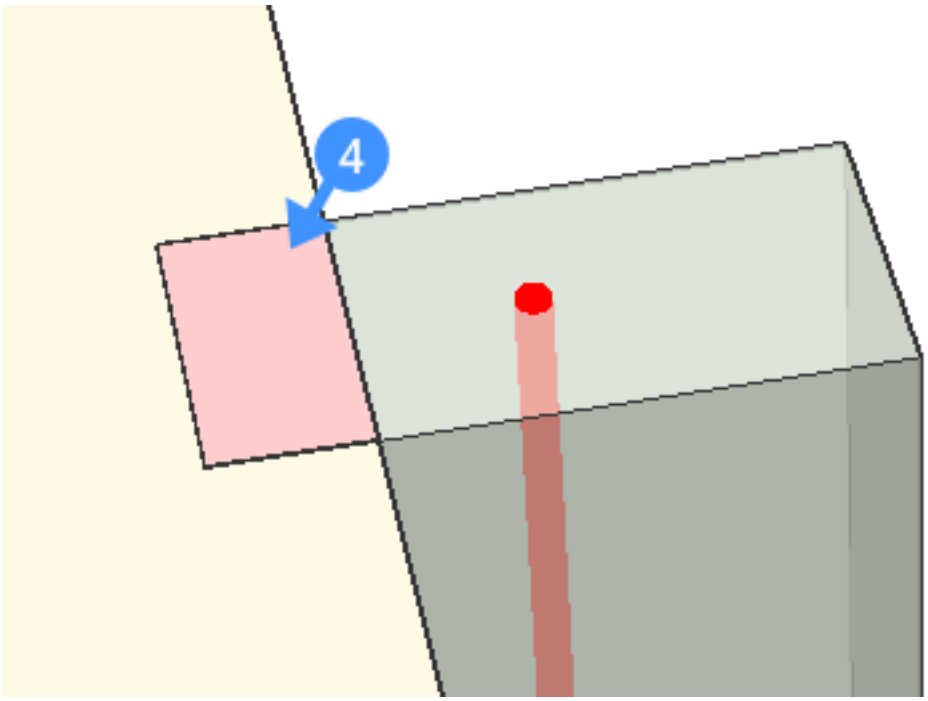
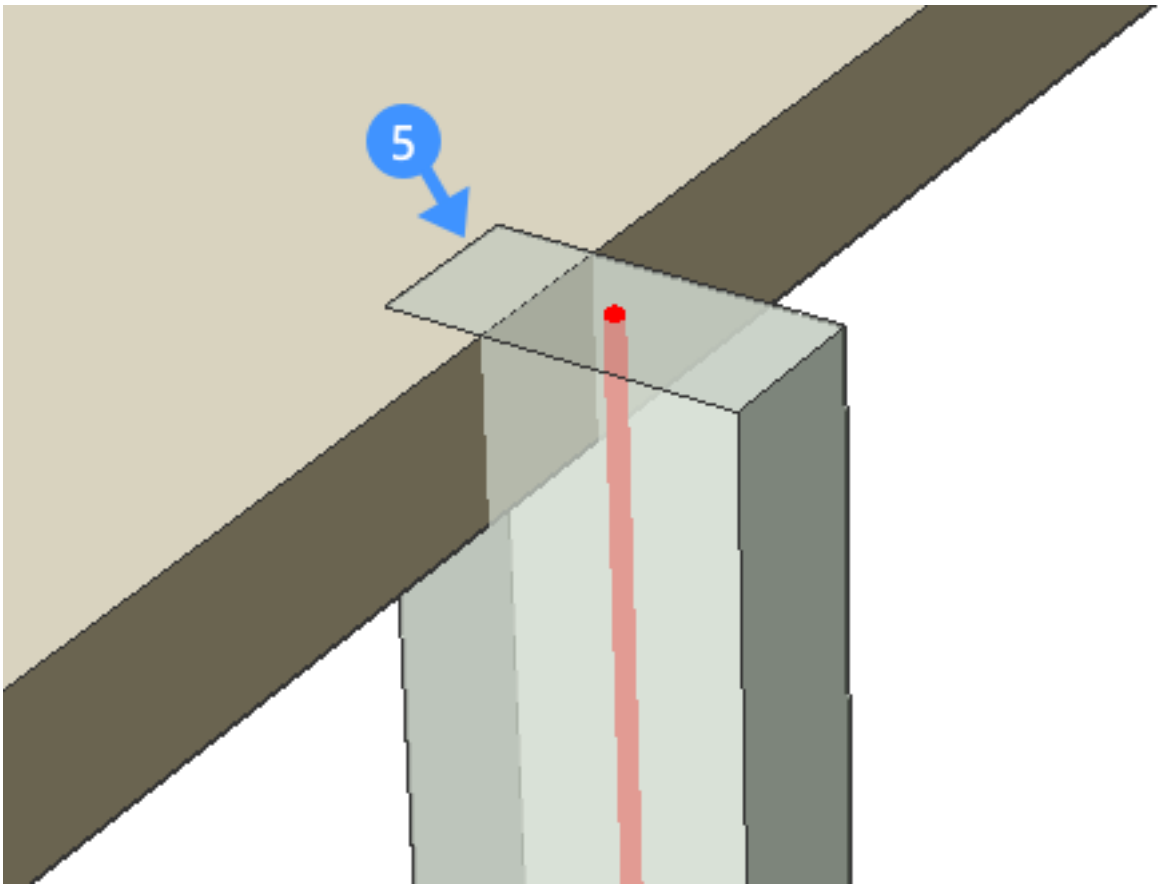


Figure 2.417

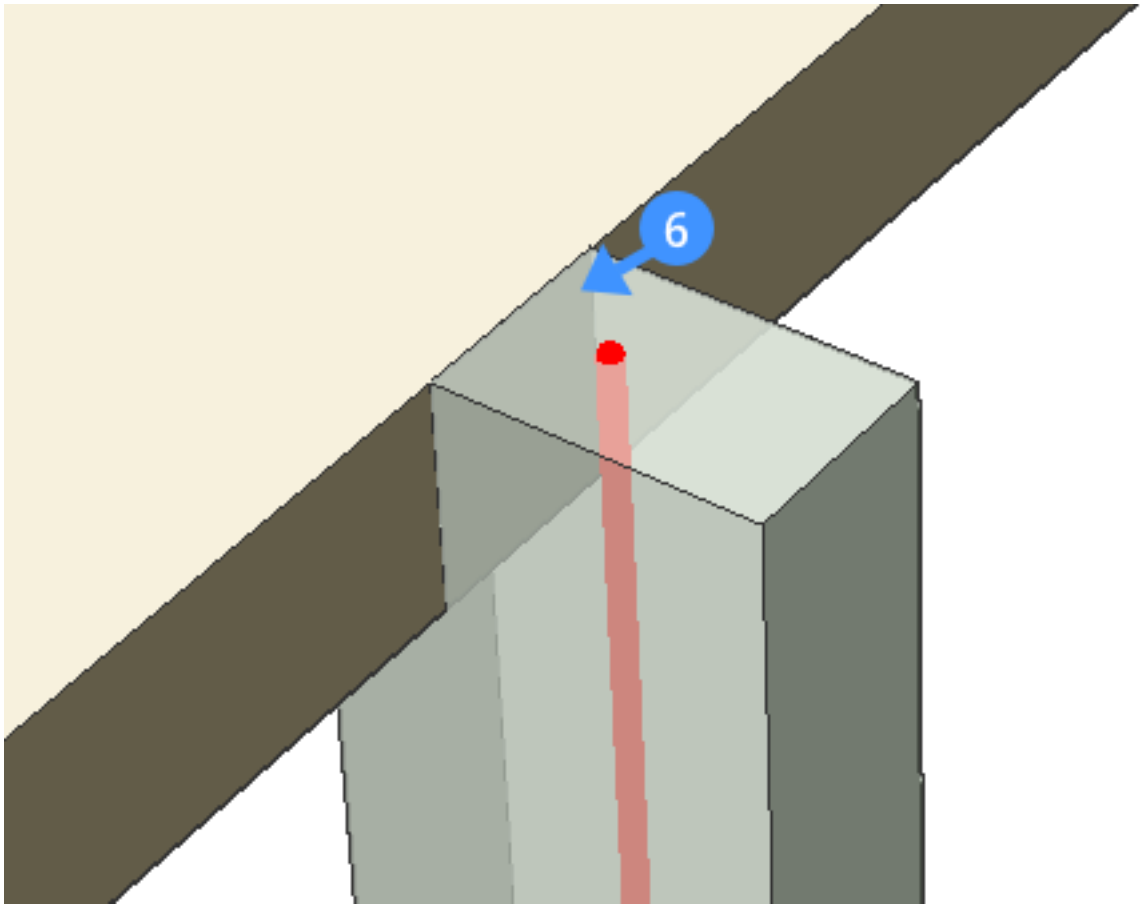


In the following illustration, the created solid of the interference volume is shown (5). In the second image, the intersection is fixed by subtracting the slab geometry from the column. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/InterferenceSolved.png})

6. Run the Interfere command a second time to check whether all interferences are fixed.

If there are no collisions left, the command line reports that “Solids do not interfere”.

Figure 2.418



2.10.5 Using Xrefs

Command: XRef (6.34.8), XAttach (6.34.1), XOpen (6.34.6), XClip (6.34.2) **Overview:** In this lesson, you learn how to attach and detach DWG files to the current model through the Drawing Explorer. **Lesson Objectives** After completing this lesson, you will be able to:

- Attach externally referenced models to a master model.
- Unattach, reload, and unload the XRefs in the master model.

About

Using external references or Xrefs allows you to insert any model in the current BIM model. Instead of copying the model, only a reference to the source model is saved in the master model. You can attach multiple models to a single master model without increasing the size of the current model significantly. Further advantages of using the Xrefs are that the source models can be edited simultaneously by different members of the design team, and the master model can always be updated to show the latest version of each of the source models. **Note:** You can distinguish between XRef entities and entities in the current drawing using the XDWGFADECTL system variable. This system variable allows to fade the display of XRefs. You can give the values between 0 (no fading) and 90 to change the default value of the fading. **Keyboard:** xref

Procedure:Managing XRefs in the current drawing

1. Open the Drawing Explorer dialog box and click the External References section. The XRef preview pane displays for attaching a model or managing the existing XRefs on the list. The following illustration shows the XRef preview pane. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/X

External references (1)	Shows the External References section.
Attach XRef (2)	Attaches a model as an external reference.
Detach XRef (3)	Removes the selected XRef completely from the current model.
Reload XRef (4)	Reloads the selected XRef. This is useful when the source models have been changed, and you want to see the updated versions in the master model.
Unload XRef (5)	Unloads the selected XRef. This hides the XRef from the view.
Bind XRef (6)	Merges the source models permanently to the current drawing. Merging the XRef to the current drawing makes the XRef part of the drawing. References to the XRef disappear from the Drawing Explorer.
Insert XRef (7)	Converts XRefs into blocks.
Open XRef (8)	Opens the referenced drawing.
Detail view (9)	Displays the details of XRefs.
Icon view (10)	Displays icons of the XRefs.
Tree view (11)	Displays the relationship between XRefs in a tree-like view.

2. To attach externally referenced models, click **Attach XRef** tool button on the Drawing Explorer dialog box. The **Select Reference File** dialog box displays. The following illustration shows the Select Reference File dialog box. From this dialog box, select the DWG file you wish to attach in your main model, and then click **Open** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/SelectReferenceFileDialog
Next, the **Attach External Reference** dialog box displays. The following illustration shows the Attach External Reference dialog box. The table below presents each option of the Attach External Reference dialog box. From this dialog box, select the options you want to apply, and then click **OK.** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectCollaboration/AttachExternalReferenceDialogBox.png})

Path field (12)	Displays the full path of the selected drawing.

Path type (13)	Includes full, relative, and no path options. **Full path:** When you move your drawing containing an XRef to another location,BricsCAD will still search for the XRef in the previous location and it will not find this XRef. **Relative path:** When you move your drawing containing an XRef to another location, the new file path will be created in the new location. The file folder will set up correctly using the same folder structure as the first one.
External reference type (14)	Includes Attachment and Overlay reference types. **Attachment:** Attaches all XRefs, including nested XRefs. For example, you have the first level of XRef "X" and this contains nested XRefs "Y" and "Z". When the attachment option is selected the first level of XRef and its nested XRefs Y and Z will be attached to the drawing. **Overlay:**Attaches only the first level of xref (X), nested xrefs (Y, Z) will be not loaded. This can also be used to prevent circular references.
Insertion point (15)	Allowsto specify the Insertion point on the screenor in the X-, Y-, Z- fields.
Scale (16)	Allows to specify the Scale on the screen or in the X-, Y-, Z- fields. Check the **Uniform Scale** option to specify the scale in the X- field.
Rotation (17)	Allows to specify the Rotation angle on the screen or in the Angle field.

Note that BIM XReference option from BIM Classify allows you to classify the model as an external reference. The identification of information will not explicitly be represented in the current model or in the project database. Such information appears after attaching the model to another project. To do so, follow the above step. The following illustration shows the BIM category in the properties panel after attaching the model that has been classified as an XRef. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/ProjectColl

3. To detach an existing XRef, select the XRef you wish to detach in your main model, and then click the **Detach XRef** tool button on the Drawing Explorer dialog box. The selected XRefs are detached. **Note:** Press and hold the CTRL key to select multiple XRefs or to unselect a selected XRef.

4. Do one of the following to unload and reload an existing XRef.
   - Click the Loaded (18) column of a loaded XRef. This XRef is unloaded in your main model. To reload the XRef check the load box of this XRef.
   - Select the XRef, then click the Unload XRef (19) tool button on the Details toolbar. Select the unloaded XRef, then click Reload XRef (20) tool button. The following illustration shows the Load column, Unload XRef and Reload XRef tool buttons on the Drawing Explorer. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/

5. To modify the attachment type, click the **Type** column of an existing XRef and switch from the Attach to Overlay or vice versa. The same process can be followed for modifying the **Saved path** of an XRef and **Found path** of an overlay.

6. Click **X** to close the dialog box. Notice that the changes that have been made appeared after you close the Drawing Explorer.

Figure 2.419

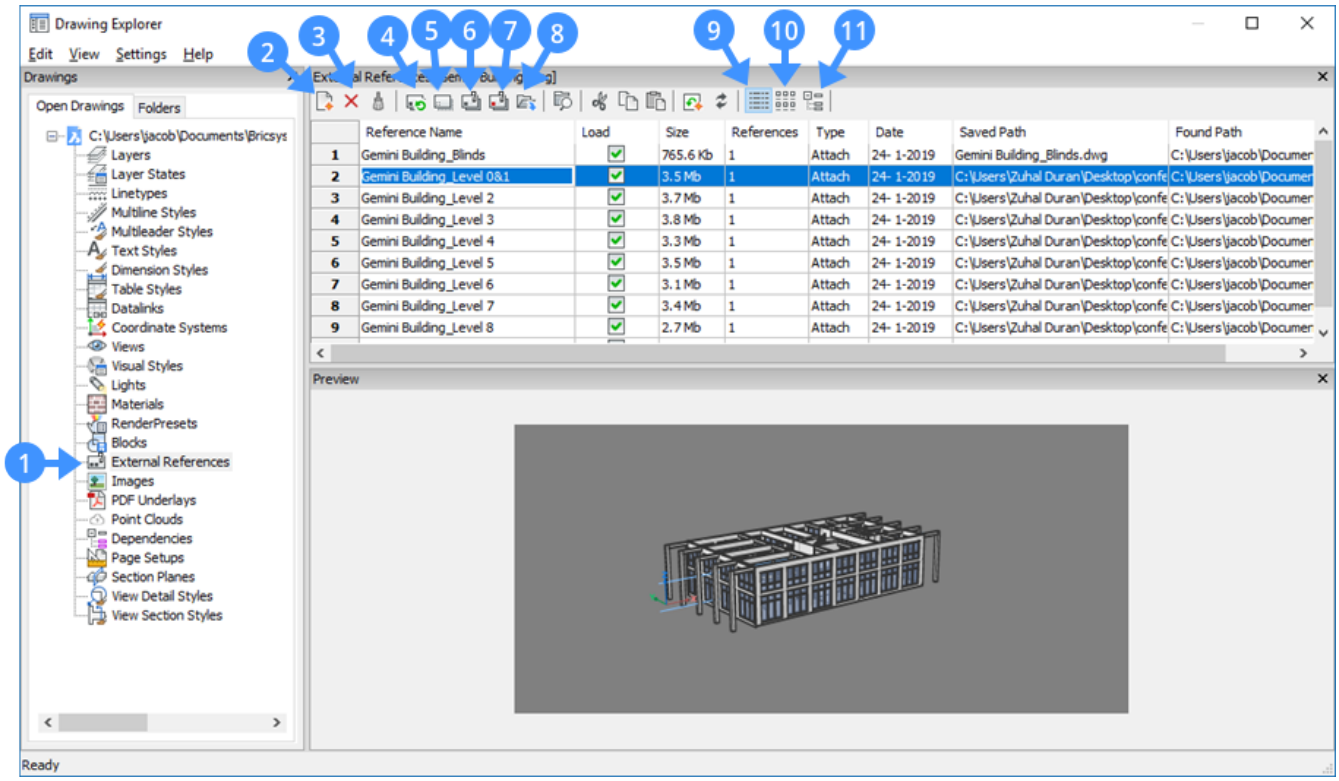


Figure 2.420

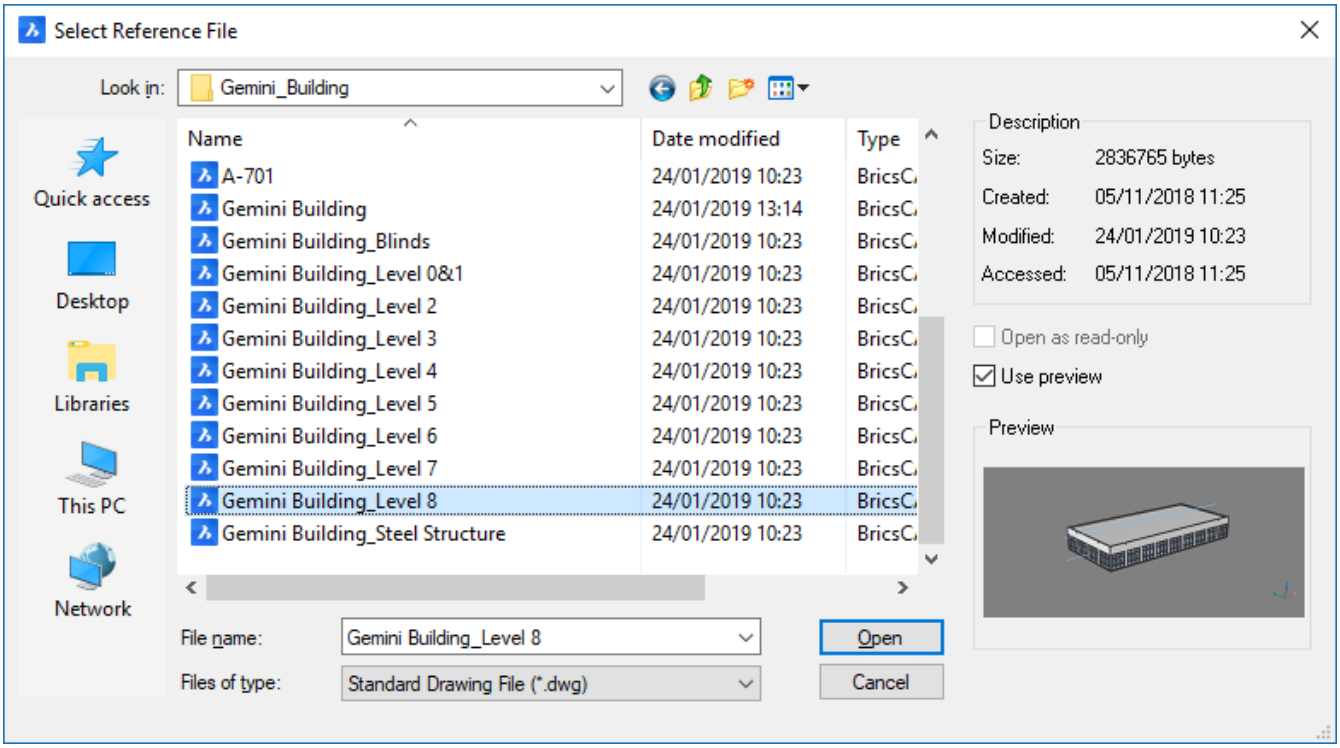


Figure 2.421

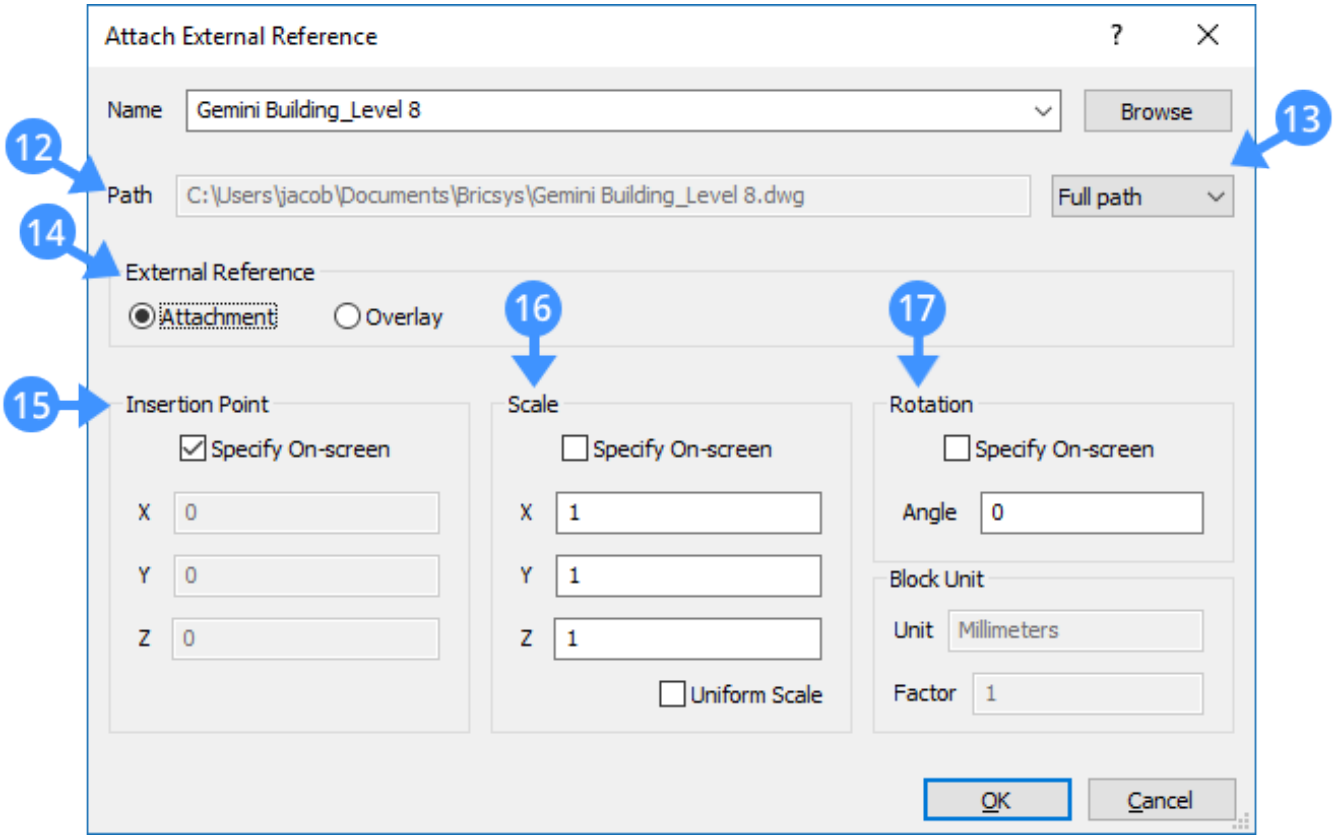
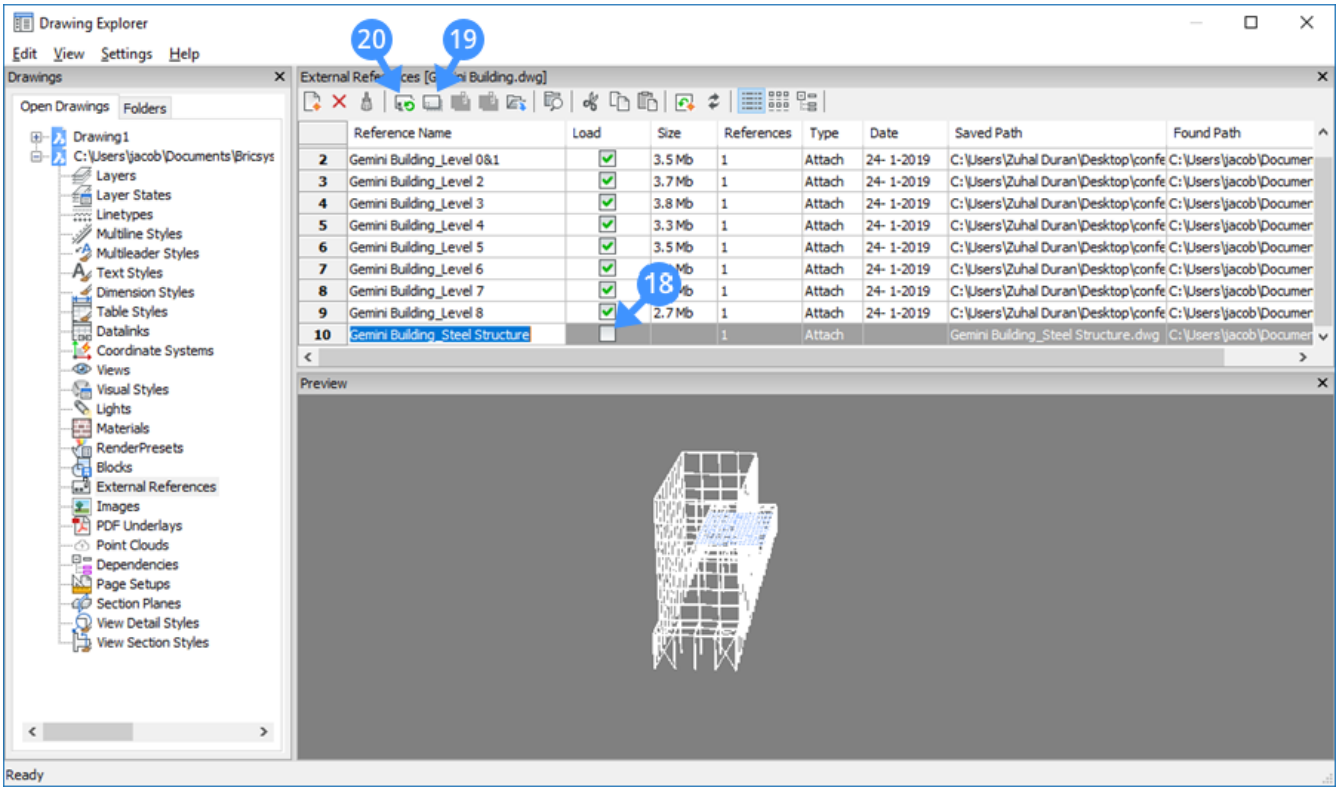


Figure 2.422



BIM	
Type	BIM XReference
Name	
Description	
Building	
Story	

Figure 2.423


Drawing Explorer
Edit View Settings Help
Drawings
Open Drawings Folders
Drawing1
C:\Users\jacob\Documents\Bricsys
Layers
Layer States
Linetypes
Multiline Styles
Multileader Styles
Text Styles
Dimension Styles
Table Styles
Datalinks
Coordinate Systems
Views
Visual Styles
Lights
Materials
RenderPresets
Blocks
External References
Images
PDF Underlays
Point Clouds
Dependencies
Page Setups
Section Planes
View Detail Styles
View Section Styles
20
19
18
Reference Name
Load
Size
References
Type
Date
Saved Path
Found Path
2 Gemini Building_Level 0&1 3.5 Mb 1 Attach 24- 1-2019
3 Gemini Building_Level 2 3.7 Mb 1 Attach 24- 1-2019
4 Gemini Building_Level 3 3.8 Mb 1 Attach 24- 1-2019
5 Gemini Building_Level 4 3.3 Mb 1 Attach 24- 1-2019
6 Gemini Building_Level 5 3.5 Mb 1 Attach 24- 1-2019
7 Gemini Building_Level 6 1 Attach 24- 1-2019
8 Gemini Building_Level 7 1 Attach 24- 1-2019
9 Gemini Building_Level 8 2.7 Mb 1 Attach 24- 1-2019
10 Gemini Building_Steel Structure 1 Attach Gemini Building_Steel Structure.dwg
Preview
Ready

2.11 Design Documentation

2.11.1 How to add tags

https://www.youtube-nocookie.com/embed/lA2Bn0zfgvs

This video shows how to add labels to entities on a generated 2D plan or section, either automatically or manually.
Back to Tutorial Overview

2.11.2 How to create Interior Elevations

https://www.youtube-nocookie.com/embed/oPxPF9o6d68 This tutorial shows how Interior Elevations can be generated from a BIM Room, by first creating the section planes and then generating them together or individually.
Back to Tutorial Overview

2.11.3 How to create Schedules

https://www.youtube-nocookie.com/embed/OPkqYVjrNDA This video shows how the Project Browser can be used to create schedules. Either use existing .dxd files or create your own. Back to Tutorial Overview

2.11.4 How to extract data in BricsCAD BIM

https://www.youtube-nocookie.com/embed/yiRaYGwd_-8 This tutorial explains how to use the DATAEXTRACTION command and its dialog windows to extract the necessary information from your drawing. Back to Tutorial Overview

2.11.5 How to generate sections

https://www.youtube-nocookie.com/embed/gQQ8PXSZXNg This video shows how the BIM Section planes can be used to generate 2D plans and sections for your construction drawings and sheet sets. Back to Tutorial Overview

2.12 Design Documentation

2.12.1 BIM Sections

Command: BimSection (6.7.27) **Overview:** In this lesson, you learn how to create and manage plan, elevation and section planes. Creating design documentation out of these section planes will be covered in a future lesson. **Lesson Objectives** After completing this lesson, you will be able to:

- Cut through a BIM model horizontally to show the details on a floor plan.
- Cut through a BIM model vertically to show the interior detail.
- Display a BIM model with a vertical section outside the model.
- Toggle the Clip Display property of a BIM section entity.
- Control the visibility of your BIM section cuts.

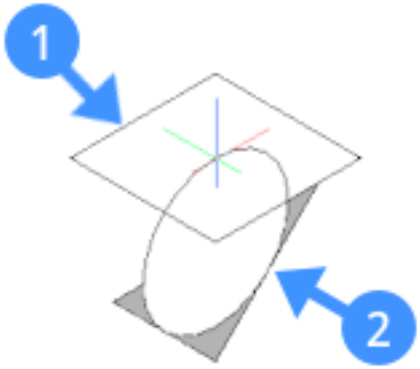
Tutorial: How to create and edit sections Click here (2.3.1) to watch

Define Section

In BricsCAD BIM, the **Define Section** tool allows you to create BIM section entities. The BIM section allows you to section a BIM model by a plane and thus view the interior details.  Menubar: **BIM > Define Section** Toolbar: **BIM > Define Section** Quad: **Model tab > Define Section** Keyboard: **BIMSECTION** When defining a BIM section, the section tracker is attached to the cursor. The following illustration shows the BIM Section Tracker elements such as Section plane (1) and View direction (2). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignD The section plane lies in the XY-plane of the current coordinate system (WCS or UCS). The view direction (2) is in the negative direction of the current coordinate system. **Note:**

- If **Dynamic UCS** (DUCS) is ON, the section plane aligns to the face of the 3D solid under the cursor. Hit the TAB-key to select an obscured face.
- You can press the Shift-key to lock highlighted plane, allowing you to specify the definition point for the BIM section entity outside the face of the solid.
- The section line is parallel to the X-axis of the coordinate system (WCS, UCS or DUCS).

Figure 2.424



Procedure: General steps to create a BIM Section

1. Select **Define Section** from the No Selection Quad. You are prompted: **Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:**
2. Do one of the following:

- To create a plan section, click a point anywhere outside the model.
- Hover the cursor over the face of a 3D solid which is parallel to the section plane you want to create. The section tracker aligns to the face under the cursor, on the condition that DUCS is active. Left-click.
- Optionally, hit the Shift key to lock the highlighted plane, allowing you to start from a point outside the selected 3D solid face. Next, left-click.

The initial section plane displays dynamically and the 3D model is clipped accordingly.

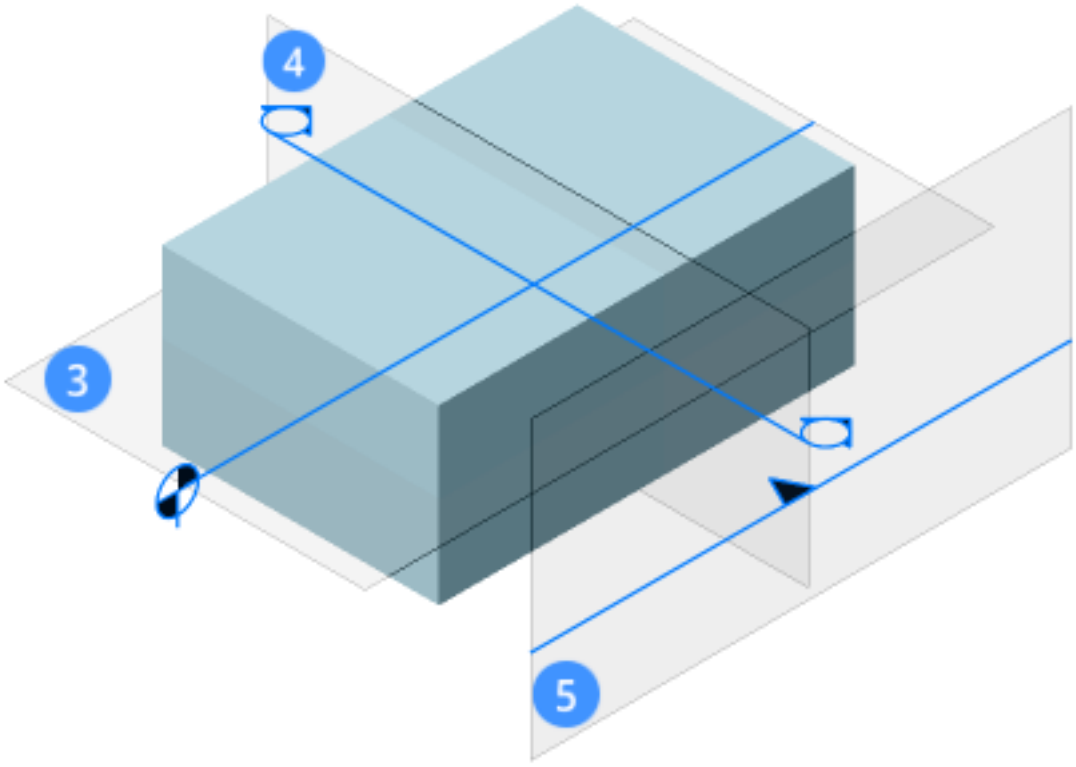
3. You are prompted: **Specify distance:** Do one of the following:

- Type a value in the dynamic dimension field to offset the section from the initial position.
- Left-click when the section plane is at the location you want it to be.

The BIM Section entity is defined. The Section Type will depend on the section plane directions and its place in the model. The following illustration shows the BIM section types on the 3D entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/BIMSections.png})

Plan (3)	Shows horizontal section plane
Section(4)	Shows vertical section that cuts through the model
Elevation (5)	Shows the exterior elevation view

Figure 2.425



Procedure: Defining a plan section

Note: A plan section is best defined in the WCS (World Coordinate System).

1. Select the Define Section tool from the Quad. You are prompted: **Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:**

2. Place the **Section Tracker** near the 3D model and click. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
The initial section plane lies in the XY-plane (Z = 0) of the current coordinate system (e.g. the WCS). The section plane displays dynamically and the 3D model is clipped accordingly. (\vref {fig:C:/Dev/Task-2448/images/staticbrics

3. You are prompted: **Specify distance:** IfDynamic DimensionisON, the current distance displays in a dynamic dimension field. Do one of the following:

- Move the cursor and click when the section plane is at the desired height.
- Use an Entity Snap to specify a point.
- Type a value to define the offset from the initial position of the section plane.

The plan section is defined.

Figure 2.426

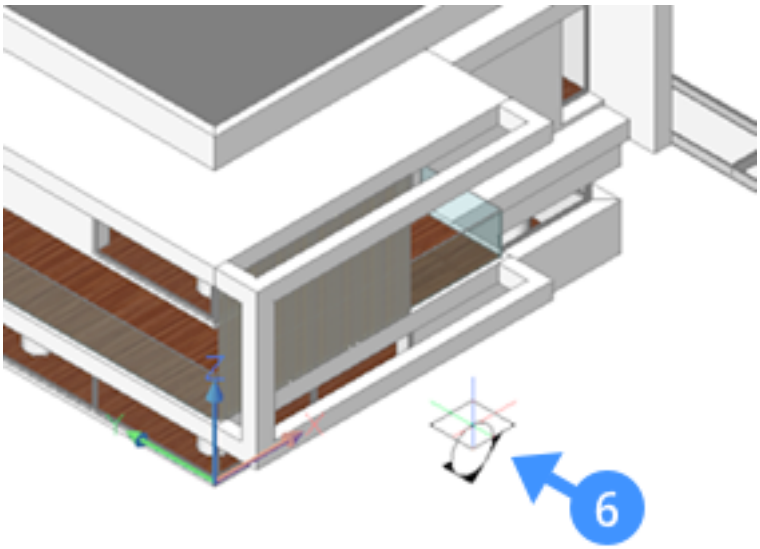
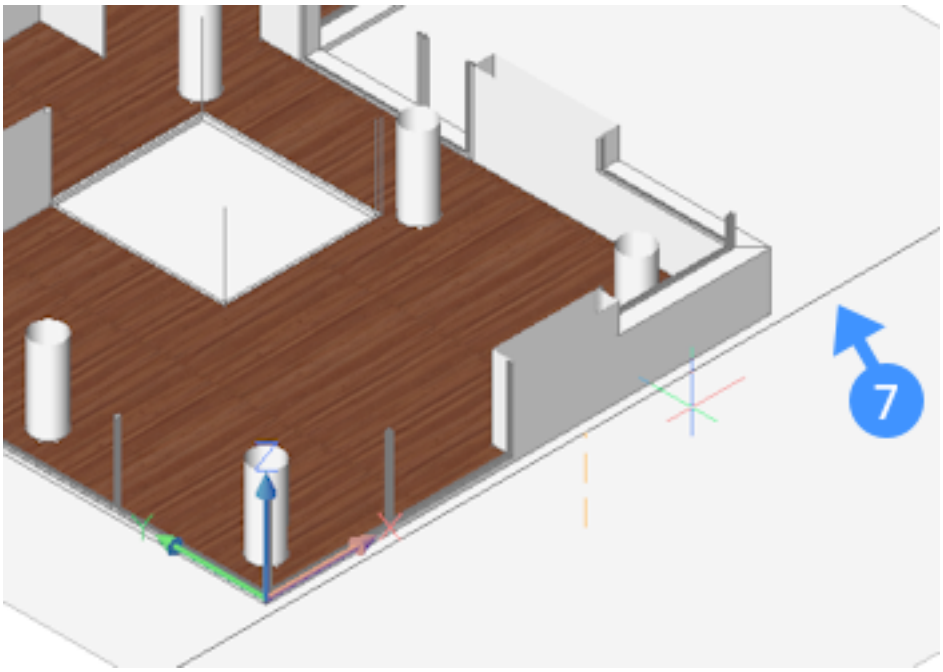


Figure 2.427



Procedure: Defining a cross section

Note: Make sure Dynamic UCS (DUCS) is enabled when defining a cross-section.

1. Select the Define Section tool from the Quad. You are prompted: **Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:**
2. Place the section tracker over a 3D solid face (8) which is parallel to the cross-section to be defined. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/DefineCrossSection1.png Click when the face highlights and move the section tracker inside the 3D model.
3. The section plane displays dynamically and the 3D model is clipped accordingly. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/DefineCrossSection2.png}) You are prompted: **Specify distance:** If Dynamic DimensionisON, the current distance displays in a dynamic dimension field.
4. Click when the section plane is at the desired location or specify a distance. The cross-section is defined.
5. (Optional) Drag the Start grip to move the section plane.

- Drag the End grip to rotate the section.
- Click the Arrow grip to flip the view direction.

Figure 2.428

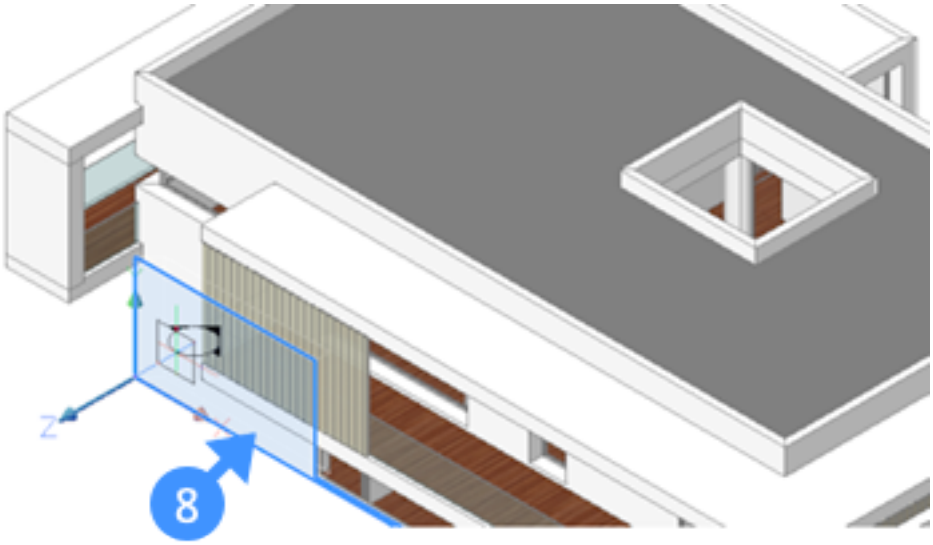
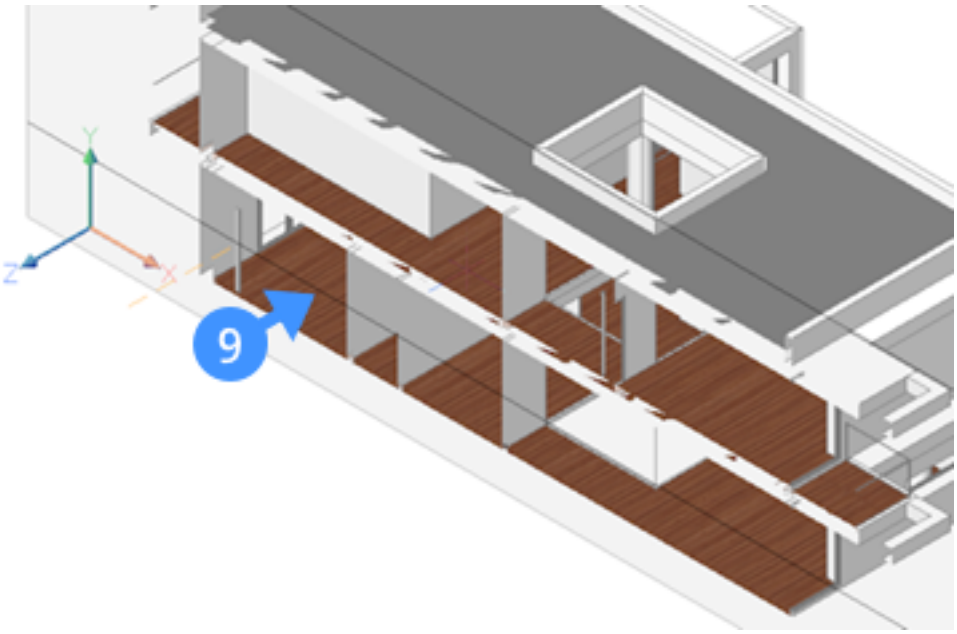


Figure 2.429



Procedure: Defining an elevation

Note: Make sure Dynamic UCS (DUCS) is enabled when defining an elevation.

1. Select the Define Section tool from the Quad. You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:
2. Place the Section Tracker over a 3D solid face (10) which is parallel to the elevation to be defined and click when face highlights. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentatic
   You are prompted: Specify distance:
3. Move the section tracker outside the 3D model and click to define the elevation. The elevation is defined (11).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/DefineElevation2.png})
Vertical sections outside the model (no geometry is sectioned) get the**Elevation s**ection type automatically. **Note** that elevations are used to create design documentation. This is a topic that will be covered in future lessons.

Figure 2.430

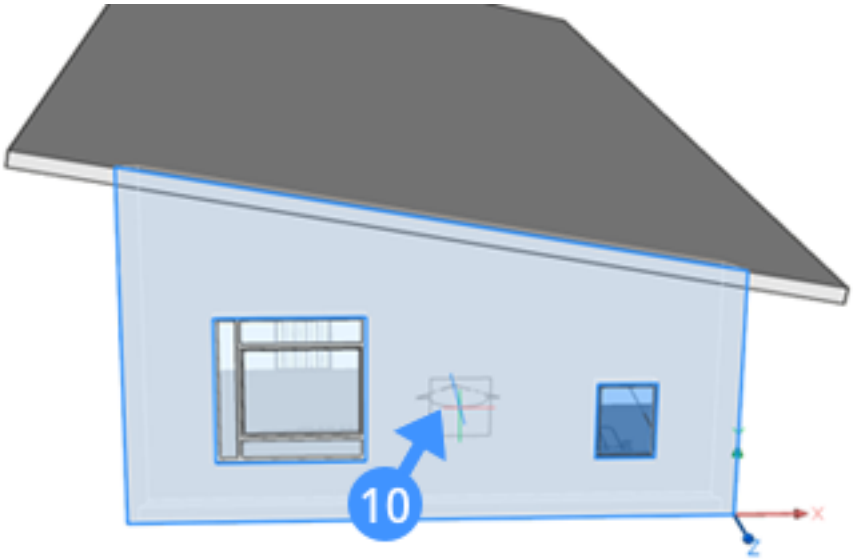
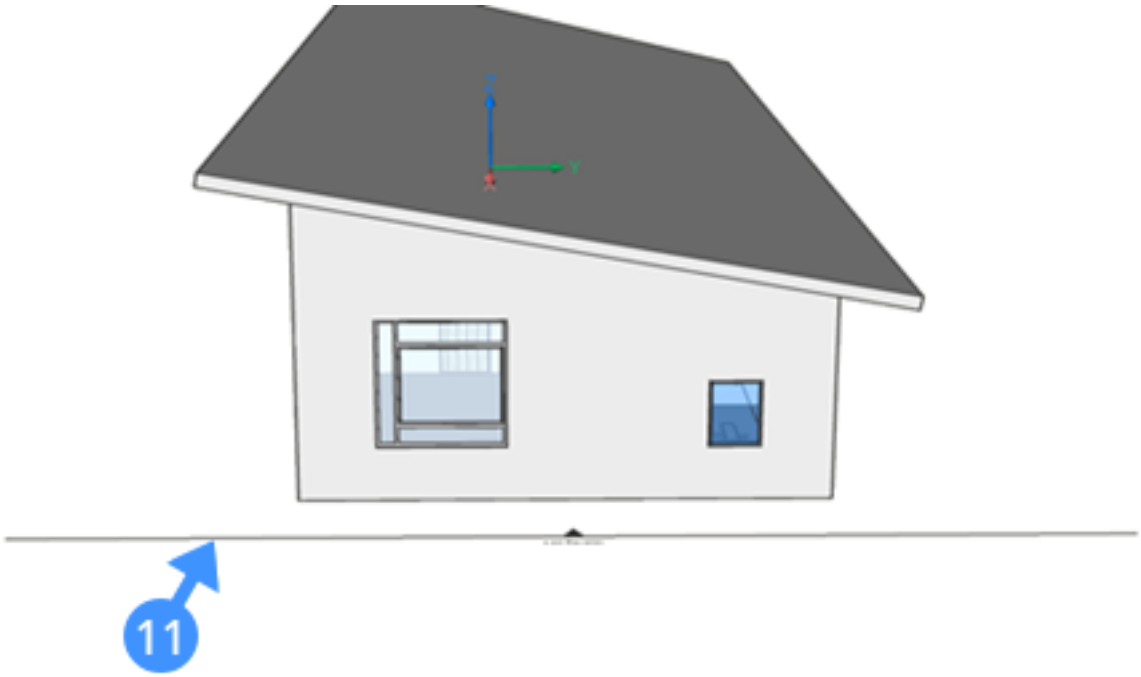


Figure 2.431



Procedure: Changing the default properties of an existing BIM Section

The default properties of a BIM section entity are based on the .dwg file of 'Section Settings' drawing in the BIM/Sections subfolder of the Support folder.

1. Select an existing BIM section in the drawing. The current properties display in the Properties Bar.
2. Define the properties of the BIM section entity. The following illustration shows the Properties bar with the current properties of a BIM section. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDo

Section Entity:

Name	Type a name.
State	Click the field and select a state. The selected state determines how the section result will be clipped.
Live section	Sets the live section property of the section plane. Use this mode for visual checking only, not for 3D modeling. It is always recommended to use the "Clip Display" property instead of the "Live Section" property.
Clip display	Clips the 3D model. As opposed to the **Live** status, the Clip Display status can be set for multiple section entities simultaneously.
Use lids	Closes the top faces of the clipped geometry.
Plane transparency	Sets the transparency of the section plane in rendered Visual Style.
Plane color	Sets the fill color of the section plane in rendered Visual Styles.

Geometry:

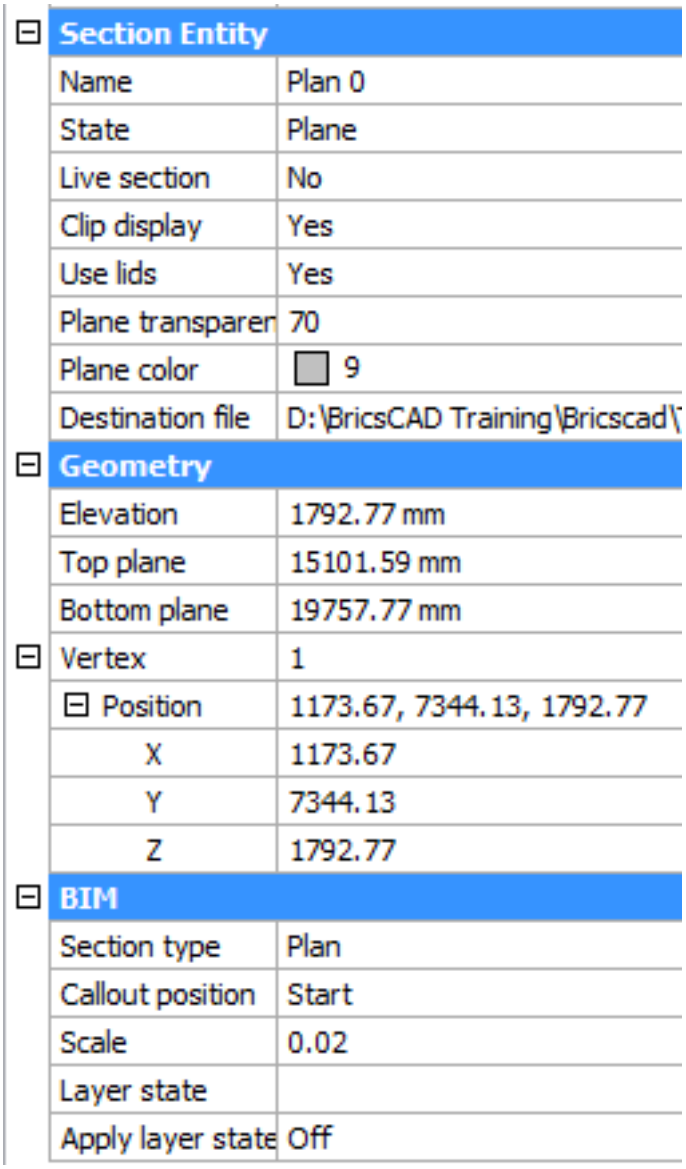
Elevation	Defines the Z-value of the start point of the section plane.
Top plane	Defines the top plane of the section volume.
Bottom plane	Defines the bottom plane of the section volume.
Vertex	Defines to coordinates of the start point of the section plane.

BIM:

Section type	Sets the BIM section type.
Callout position	Defines the position of the callout(s).
Scale	Sets the scale of the viewport in the generated drawing.
Layer state	Indicates the layer state to be used to calculate the section.
Apply layer state	Determines whether the layer state will be used to calculate the section or not.

The following illustration shows the BIM section callout positions e.g. start (12), mid (13), end (14), and start-end (15). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/BIMSectionCallout

Figure 2.432



Section Entity	
Name	Plan 0
State	Plane
Live section	No
Clip display	Yes
Use lids	Yes
Plane transparer	70
Plane color	9
Destination file	D:\BricsCAD Training\Bricscad\
Geometry	
Elevation	1792.77 mm
Top plane	15101.59 mm
Bottom plane	19757.77 mm
Vertex	1
Position	1173.67, 7344.13, 1792.77
X	1173.67
Y	7344.13
Z	1792.77
BIM	
Section type	Plan
Callout position	Start
Scale	0.02
Layer state	
Apply layer state	Off

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/BIMSectionCalloutPo
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/BIMSectionCalloutPositi
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/BIMSectionCalloutPositi
Click the green arrow ( ) to flip the view direction of the section.

Figure 2.433



Figure 2.434

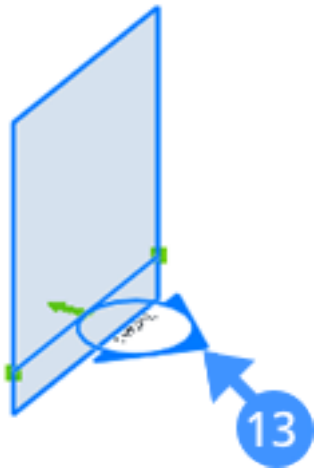


Figure 2.435

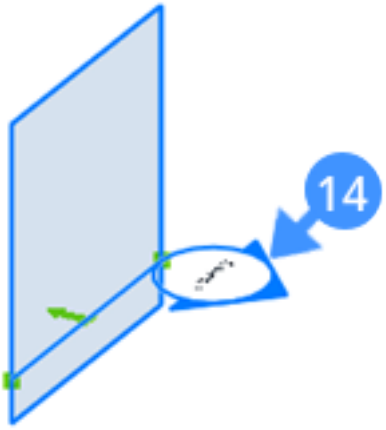
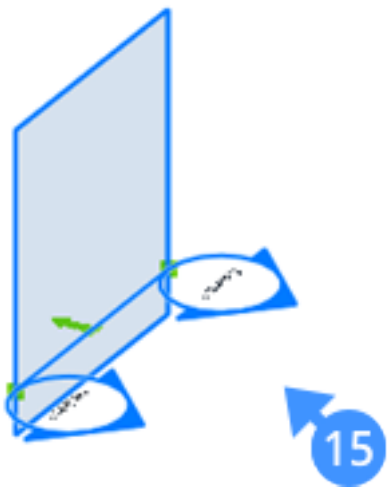


Figure 2.436



Procedure: Toggle the clipped display property of a BIM section entity

The **Clip Display** property of the BIM section allows to temporarily clip 3D models. The Clip Display status allows to progress using all modeling and editing operations on all drawing content.

1. Create a section view in your drawing. The following illustration shows the 3D model of a villa with a plan section (16) for the ground floor plan.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/ClipDisplayOff.png})

2. To clip the 3D model, do one of the following:

- Double click the section entity.
- Select Clip Display from the Quad.
- Select the section plane and set the Clip Display property to Yes.

Note: The **Clip Display** can be on for multiple section entities simultaneously. In the following illustration, the Clip Display is turned **ON** for two BIM section entities (17,18). These section entities are in opposite view directions: section plane 18 is a plan section, section plane 17 is a reflected ceiling section. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/ClipDisplayOn.png}) Optionally, the State property for the current section can be changed from **Plane** to **Volume** in the properties panel. In the following illustration, the clipped section is shown in the **Volume** state (19). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU

Figure 2.437



Figure 2.438

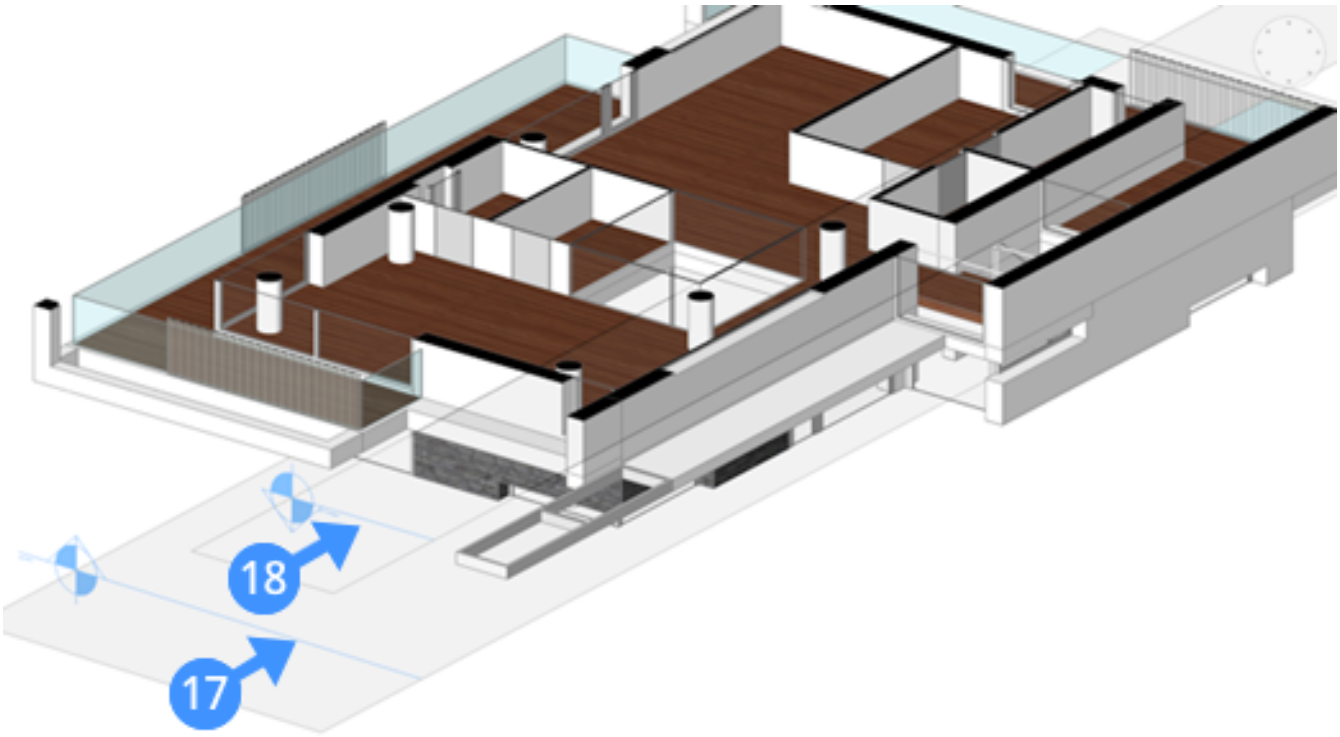
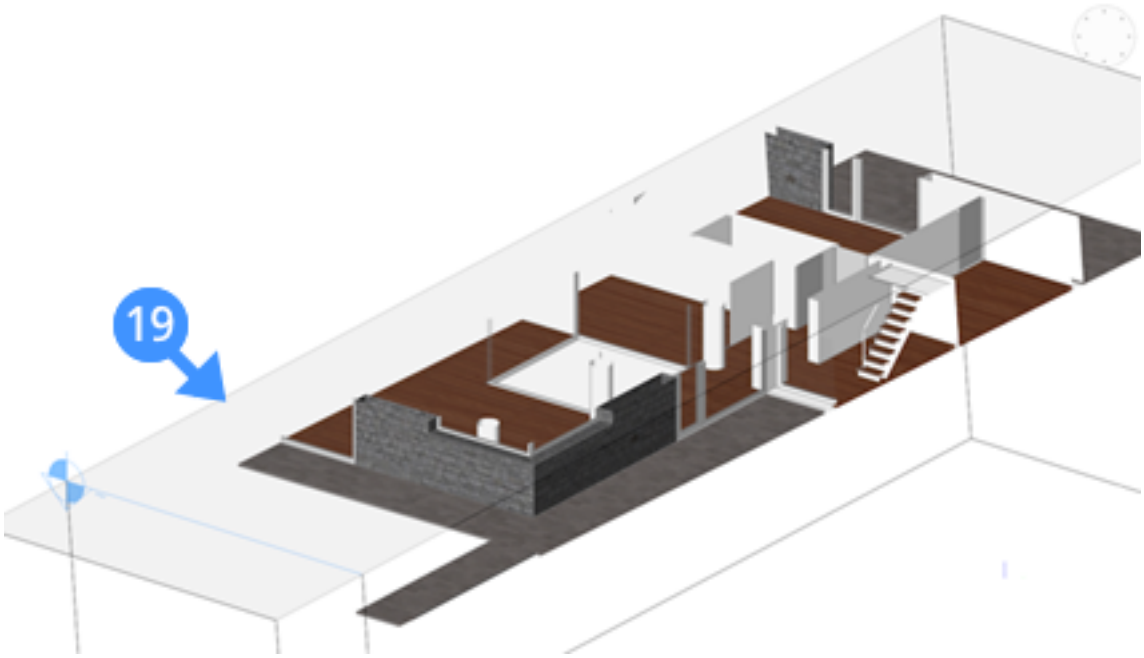


Figure 2.439



2.12.2 Editing Data Extraction Definition (.dxd) files.

DXD (Data Extraction Definition) files are plain text files.You can edit DXD files created by the DataExtraction (6.10.1) command using a text editor such as MS Notepad and use them in the Creating Schedules (2.6.9) procedure in the Project Browser (2.6.9). Lines starting with semicolon (;) are comments and ignored on parsing. Empty lines are ignored as well. The file can contain 4 sections, each section name is written between square brackets;, section names are English case-insensitive (ECI) strings. The sections are:

- Settings
- Entity Types
- Properties
- Filter

The [Settings] section

The section contains values of the following parameters which control the behavior:

- **ExtractFromBlocks** - whether to take entities from blocks into account, considering all nesting levels.
- **ExtractFromXrefs** - whether to take entities from external reference blocksinto account.
- **CountXrefs** - whether xrefs are taken into account at all.
- **CombineEqualRows** - whether to group equal rows in the output table; if yes, then the 'Count' column is added, which indicates how many equal data rows are included.
- **IncludeNameColumn** - whether the output includes the 'Name' column. The 'Name' column indicates the source entity type: 3D Solid, 3D Solid Ply, ...

Parameter names are ECI strings.The format is: ParameterName=value, The value is either 1 or 0. No spaces should be written around the equal (=) character.

The [Entity Types] section

This section contains list of entity types which will be processed. The format of each line can be one of the following types:

EntityType|DisplayEntityType

- **EntityType** is the ECI entity type name, such as Circle or AttributeDefinition . This is the mandatory part of the line. .
- **DisplayEntityType** is optional and can be any character string including spaces in any language. If present, this string appears in the DATAEXTRACTION dialog and in the output table.

Block|BlockName|DisplayBlockName

- **Block** is the predefined ECI word, and **BlockName** is case-insensitive name of the block to be processed, including xrefs and minserts.
- **DisplayBlockName** is optional and has the same meaning and format as DisplayEntityType.

Block|* This is a variant of the previous format meaning that all blocks will be taken into account; if present, then all other lines regarding blocks in this section are ignored. You can use the **Filter** section to filter the content to be processed.

***** All entity types and blocks will be taken into account, all other lines in this section are ignored then. You can use the **Filter** section to filter the content to be processed.

The [Properties] section

This section contains a list of all properties to be included in the data extraction output.The format of each line can be one of the following types:

EntityPropertyName|DisplayEntityPropertyName|ValueOutputFormat

- **EntityPropertyName** is the ECI name of entity property, e.g. Lineweight, Color, Center Y and so on. Note, all names do not contain spaces except those which represent coordinates of 2D or 3D points or vectors. In the last case, coordinate letters X, Y and Z should be separated by single space from the general property name. EntityPropertyName is the mandatory part of the line, the remaining part is optional.
- **DisplayEntityPropertyName** is user-defined and can be in any string including spaces in any language. By default it is equal toEntityPropertyName.
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting (1.21.23).

Attribute|AttributeTagName|DisplayAttributeTagName|ValueOutputFormat

- **Attribute** is the predefined ECI word.
- **AttributeTagName**is the ECI attribute tag name as it is stored in the drawing.
- **DisplayAttributeTagName** and has the same meaning as for **EntityPropertyName** (see above).
- **ValueOutputFormat**is a string which is used to format the output. The syntax of the string is the same as in fields formatting (1.21.23).

Drawing|DrawingPropertyName|DisplayPropertyName|ValueOutputFormat

- **Drawing** is a predefined ECI word.
- **DrawingPropertyName** is an ECI name, which can be one of the following:
  - Comments
  - DrawingRevisionNumber
  - FileAccessed
  - FileCreated
  - FileLastSavedBy
  - FileLocation
  - FileModified
  - FileName
  - FileSize
  - HyperlinkBase
  - Keywords
  - Subject
  - Title
  - TotalEditingTime
- **DisplayPropertyName** has the same meaning for **EntityPropertyName**(see above).
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting (1.21.23).

Formula|EntityPropertyName|DisplayEntityPropertyName=FormulaExpression|ValueOutputFormat,

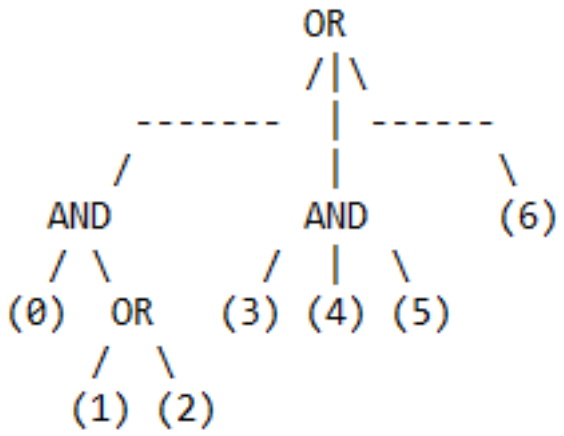
- **Formula** is predefined ECI word.
- **EntityPropertyName** is an unique name for this formula item.
- **DisplayEntityPropertyName** is an optional output name of the property, if it is omitted, then **EntityPropertyName** is used as the table column title.
- **=** is a mandatory sign, which must exist even if **DisplayEntityPropertyName** is omitted.
- **FormulaExpression** is the expression in format as in **\AcExpr** formula fields, which can containconstants such as PI or functions such asSIN (see Using Expressions (1.15.6)). An operand in an expression can be a DisplayPropertyName enclosed in arrow brackets (<>). For instance: =2 * PI * <Radius> calculates the circumference of a circle.
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting (1.21.23).

*** If present all entity properties are taken into account; all other lines in this section, except Drawing and Formula lines, are ignored.**

The [Filter] section

Syntax description:

Figure 2.440



Operator	Meaning
==	equal to
!=	not equal to
<	less than
>	greater than
<=	less than or equal to
>=	greater than or equal to

Relational operators:

- Relational expressions are enclosed in '()' brackets
- Logical operators AND and OR are case-insensitive andenclosed in '<>' brackets.
- Logical expressions use a prefix notation.
- No spaces are allowed between '<' and a logical operator ('<OR', '<AND')
- Tokens can be separated by zero or any number of spaces, except logical expression begin, example: <AND (Layer == "MyLayer")(Thickness<=3) >
- Expressions can be split into any number of lines; tokens are not broken.
- Lines can be commented with ';' .
- String values are enclosed in double quotation marks ""

<OR (Type == Window)(Type == Door)> The result of the filter in the above example is that only BIM entities classified as windows or doors are included in the data extraction. See the BimClassify (6.7.5) command to learn more about BIM entity types.

The filter expression is either a single relational one or several ones grouped by logical operations.

Example: <OR <AND (0) <OR (1) (2)>> <AND (3) (4) (5)> (6)> where (0),...,(6) are relational expressions such as (Radius >= 3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/DesignDocumentation/DataExtractio

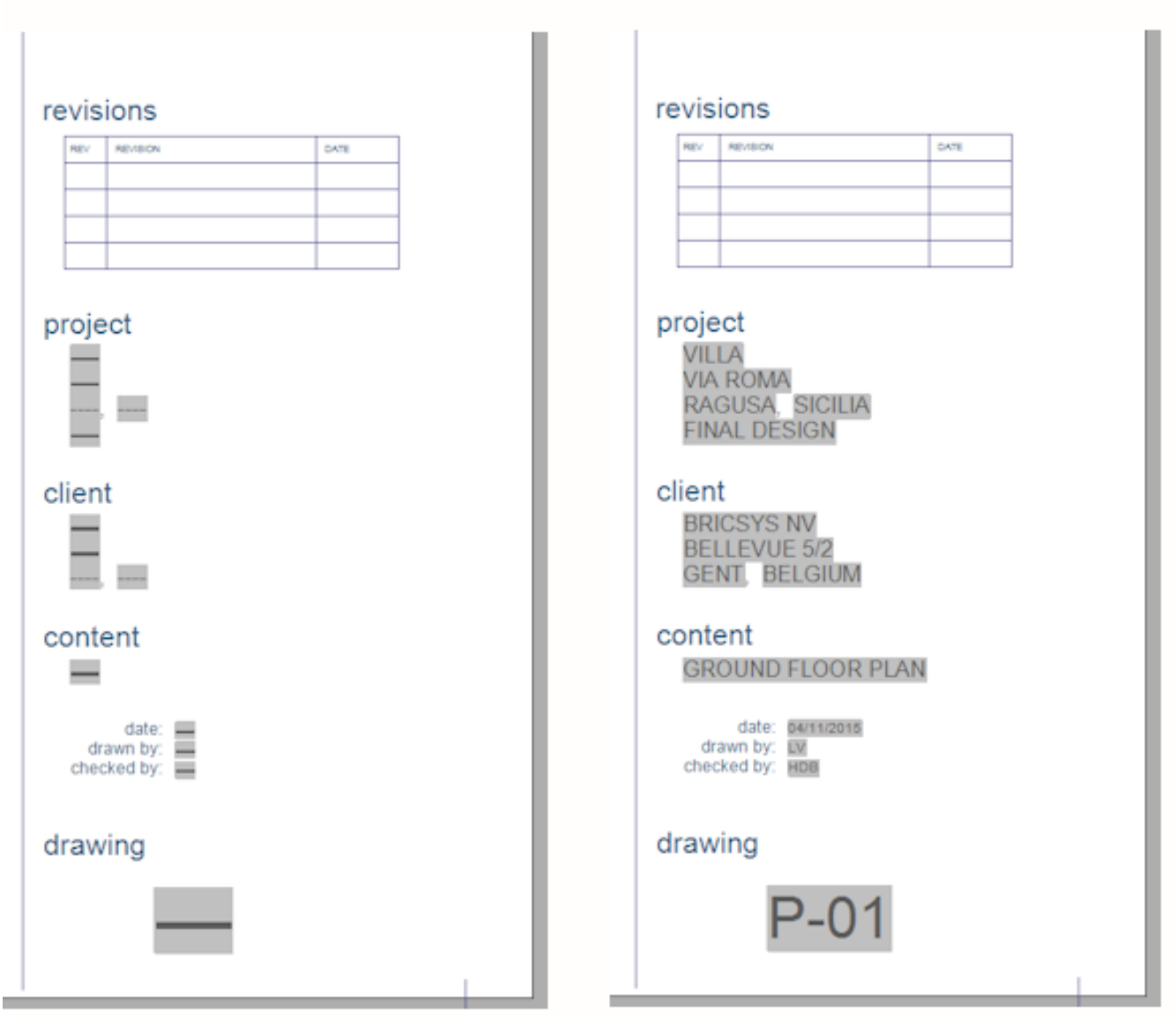
2.12.3 Editing Generated Drawings

When a BIM Section is generated, a block containing the drawing generation result is added to the model space of the target drawing, as well as a paper space viewport looking onto this block. The block and the viewport initially are placed so they do not overlap with existing geometry contained in the drawing. You can move this block or viewport to a new location, which will be preserved on subsequent updates. In the default template drawing which is used to create the generated drawings a **Viewlabel** and **NumberBubble** blocks exist. You can place them associated to the selected sheet view. The **Title** block in this drawing contains attributes that will be filled in automatically when the associated Sheet Set properties are defined. A sheet list table can be placed on any sheet of the sheet set. In the current implementation, the table content is not associative yet.

Inserting the Title block

1. Open the Drawing Explorer - Blocks (1.22.8) dialog.
2. Double click the **Title** block. The Drawing Explorer dialog closes and the block is attached to the cursor. In a metric units project the size of the **Title** block fits the A1 paper size (594 x 841 mm) In an imperial units project the size of the **Title** block fits the Arch D paper size (24 x 36 inches)
3. Click at the lower left corner of the paper sheet in the layout.
4. Define the Project and custom properties in the Sheet Sets panel.
5. Select the **Title** block, then click the **Update Fields** tool button (===ERROR WITH DOWNLOADED IMAGE===) in the **Redraw/Regen** toolbar. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtech

Figure 2.441


revisions
REV
REVISION
DATE
project
client
content
date:
drawn by:
checked by:
drawing
revisions
REV
REVISION
DATE
project
VILLA
VIA ROMA
RAGUSA, SICILIA
FINAL DESIGN
client
BRICSYS NV
BELLEVUE 5/2
GENT, BELGIUM
content
GROUND FLOOR PLAN
date: 04/11/2015
drawn by: LV
checked by: HDB
drawing
P-01

Inserting a view label for a sheet view

1. Select the sheet view In the Sheet Sets panel.
2. Right-click the sheet view you want the view label block to insert for. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco
3. Choose **Place View Label Block** in the context menu. You are prompted: : VIEWLABEL Units: Millimeters Conversion: 1 Insertion point for block:
4. Specify the insertion point. You are prompted: Scale factor for block <1.0>:
5. Specify the scale factor or right-click to accept the current scale. You are prompted: Rotation angle for block <0>:
6. Specify the rotation angle or right-click to accept the current angle. The **View Label** block is inserted. The Sheet Number (1), Sheet View Number (2), Sheet View Title (3) and Viewport Scale (4) of the selected sheet view are filled in automatically. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingte

Figure 2.442

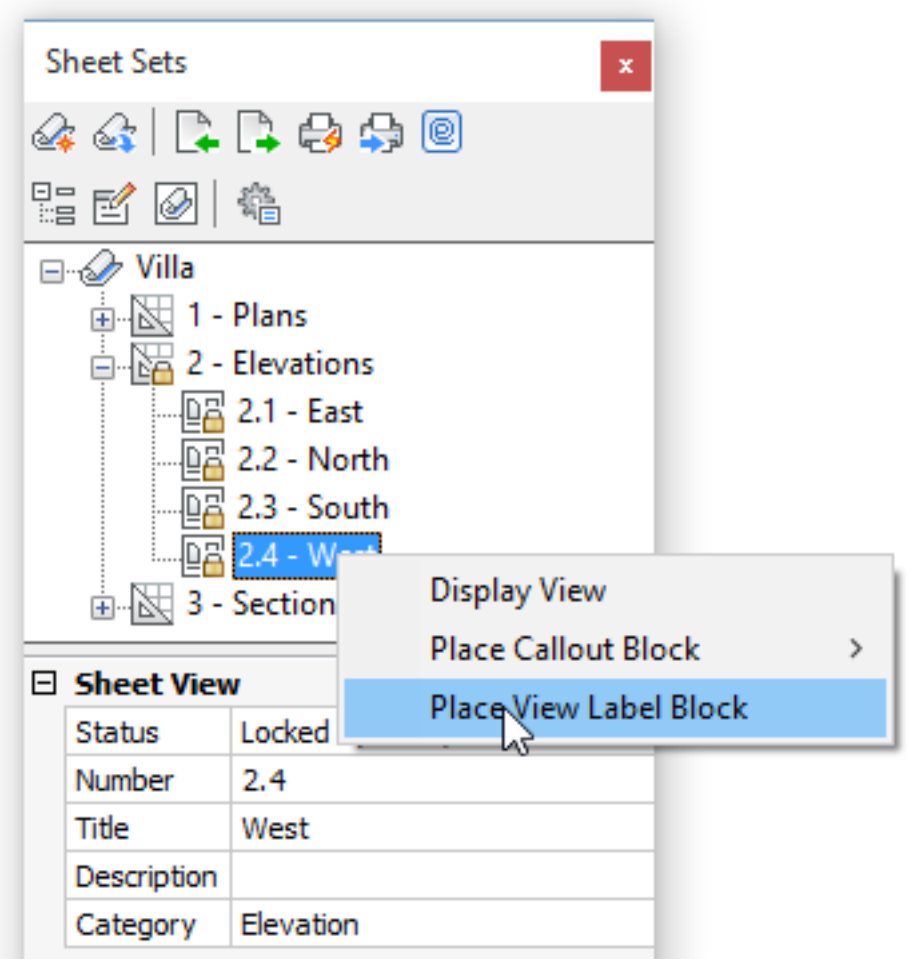
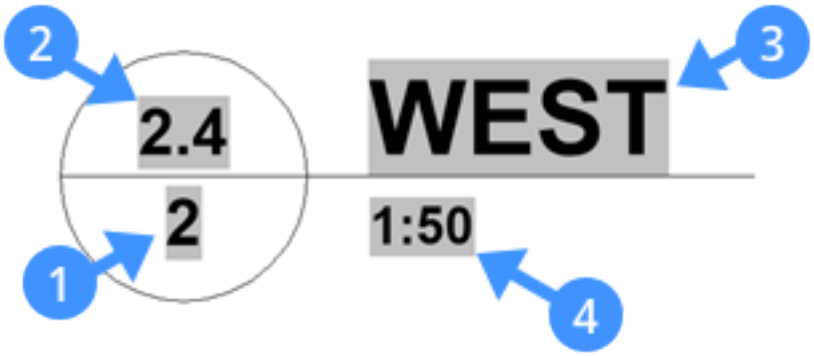


Figure 2.443



Inserting a number bubble block for a sheet view

1. Select the sheet view In the Sheet Sets panel.
2. Right-click the sheet view you want the number bubble block to insert for. (\vref {fig:C:/Dev/Task-2448/images/staticbrics
3. Choose **Place Callout Block > Number Bubble** in the context menu. You are prompted: : ._CALLOUT Units: Millimeters Conversion: 1 Insertion point for block:
4. Scale factor for block <1.0>:
5. Rotation angle for block <0>:
6. Specify the insertion point. You are prompted: Scale factor for block <1.0>:
7. Specify the scale factor or right-click to accept the current scale. You are prompted: Rotation angle for block <0>:
8. Specify the rotation angle or right-click to accept the current angle. The **Number Bubble** block is inserted. The Sheet Number (5) and Sheet View Number (6) of the selected sheet view are filled in automatically. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/NumbersBubble.png})

Figure 2.444

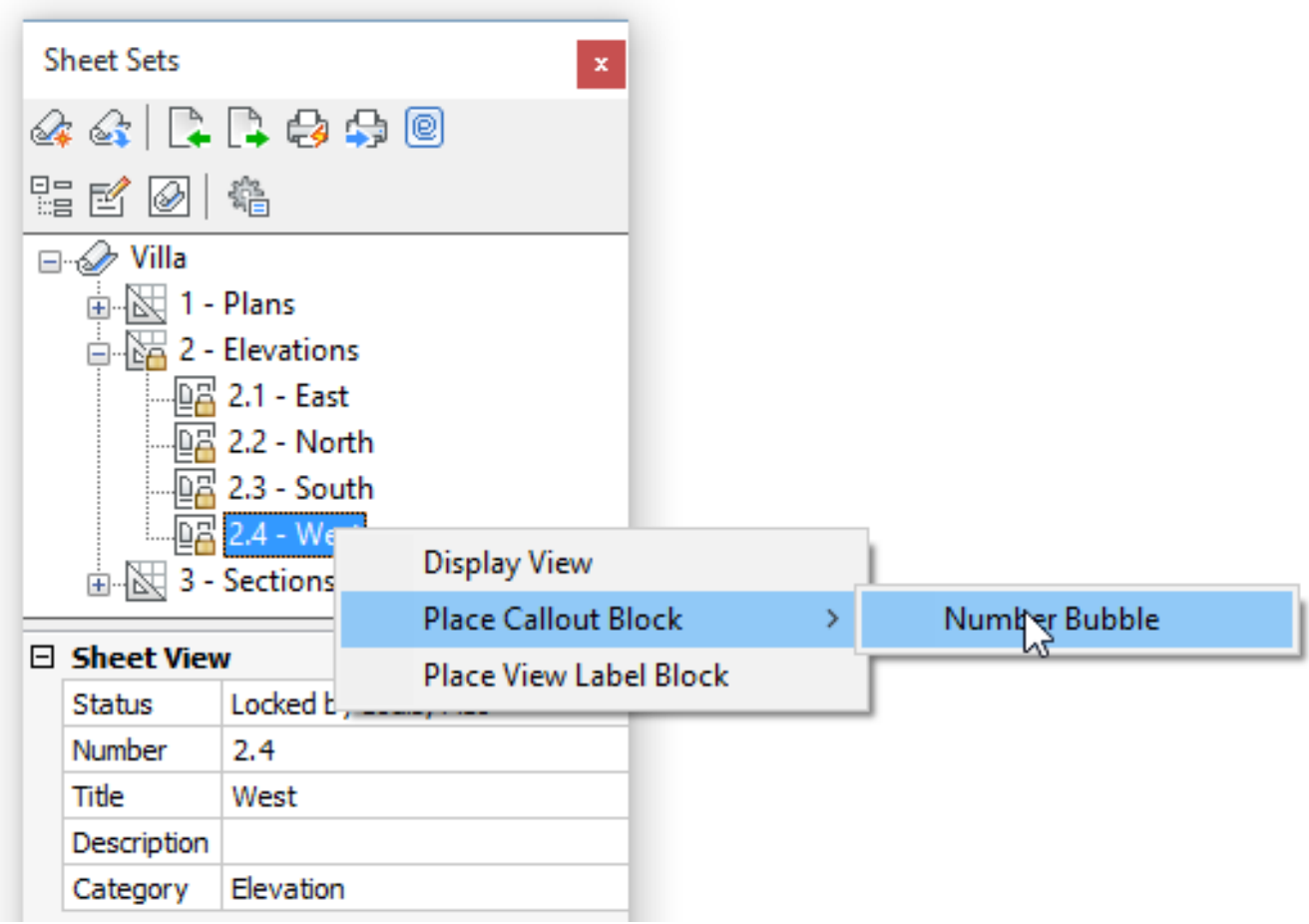
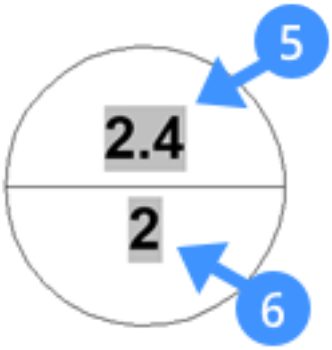


Figure 2.445



Inserting a sheet list table

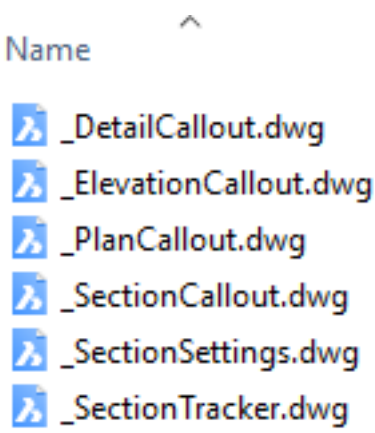
A **Sheet List** table can be inserted on any sheet of the sheetset. In the current implementation, the table content is not yet associative and therefore will not be updated automatically when sheets are added, removed or renamed.

1. Right-click the sheetset name in the **Sheet Sets** panel and choose **Insert Sheet List Table** in the context menu. You are prompted: Select insertion point.
2. Specify a point in the drawing.

Customizing callout blocks

Source files of the various callout blocks and the section tracker exist in the **..\Support\Bim\Sections** subfolder of the **Roamable Root** folder. By default this is: **C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\Vxx\en_US\Supp**
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMCalloutFolder.png})
You can edit these files to customize the layout of the callouts and the section tracker blocks. When a drawing is generated, the callout blocks needed are created from the callout blocks source file(s). When a BIM section is updated, the existing callout blocks in the target drawing are used.

Figure 2.446



2.12.4 Generating Drawings

Commands : BIMSECTIONUPDATE (6.7.29), BIMSECTIONOPEN (6.7.28) **BimSectionUpdate** generates section drawings from BIM Section entities (**??**). The command can be used either by selecting a BIM Section entity in the 3D model, or by selecting an existing section result in a generated drawing. The result is placed in a new drawing, saved in the same folder as the model drawing, unless a different folder is specified in the Destination File (**??**) property of the BIM Section entity. **BimSectionOpen** allows to navigate between the model drawing containing a BIM Section entity and the corresponding generated drawing. The command works bi-directional: either from the section result back to the model or from the model to the section result, depending on what was selected: a BIM Section entity in the 3D model or a section block or a viewport in a generated drawing.

Tutorial: How to generate sections Click here (2.11.5) to watch

To generate or update BIM sections

1. Do one of the following:

- Place the cursor over a BIM Section callout or section line, then choose **Generate Section**( ) in the **Model** command group in the Quad.
- Select one or more BIM Section entities, then choose **Update Section** ( ) in the **Model** command group in the Quad.
- Click the **Update Section** tool button in the **BIM** toolbar, then select a BIM section entity or entities.
- Choose **Update Section** in the BIM menu, then select a BIM section entity or entities.

BIM Section entities can be selected by its Callout(s) and section line only.

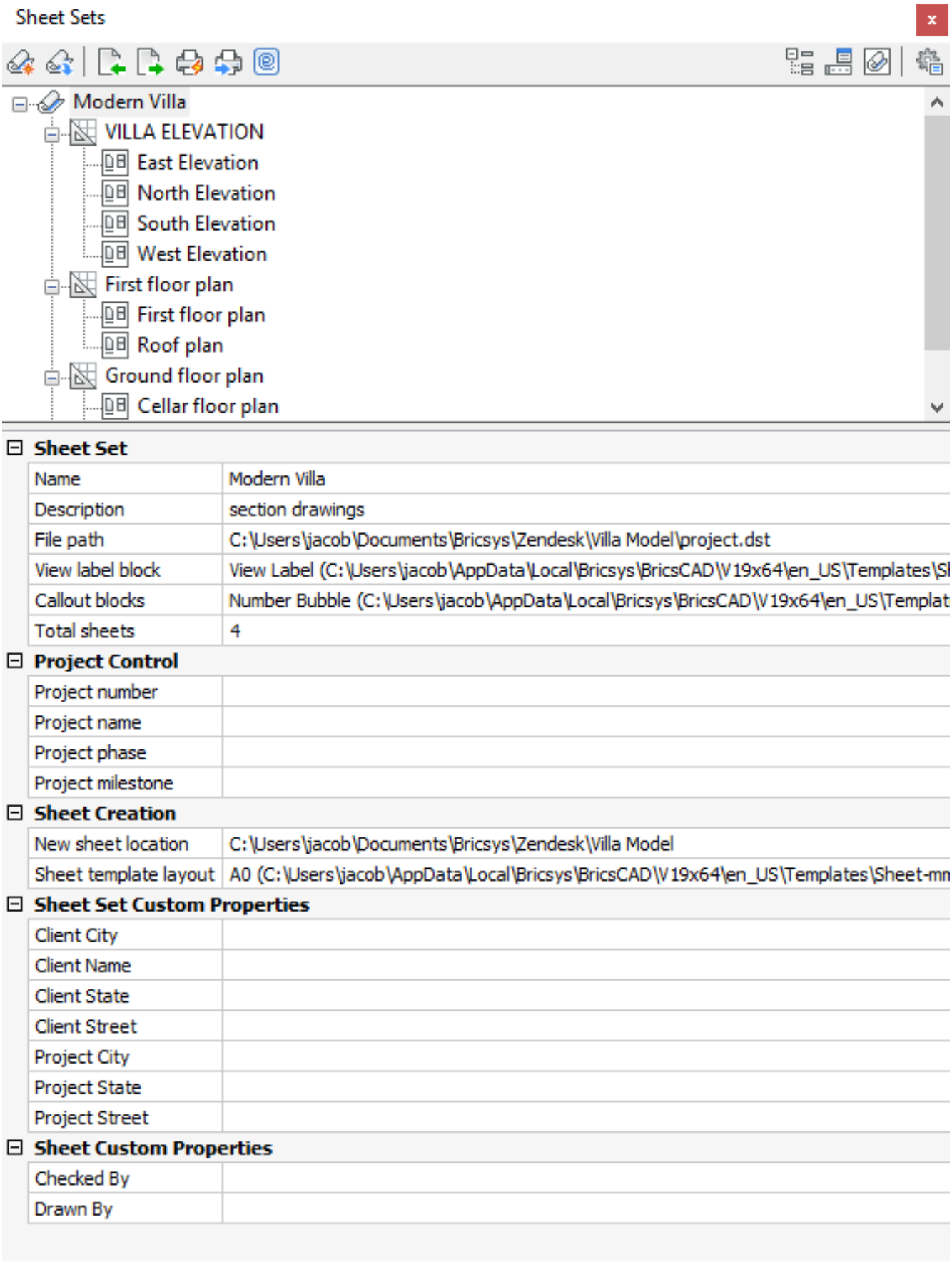
2. BricsCAD reports the number of generated sections and their name(s) in the Command Window (1.12.2), e.g.:

8 section(s) were generated D:\Bricsys Reports\V16\Villa\Elevations.dwg D:\Bricsys Reports\V16\Villa\Plans.dwg D:\Bricsys Reports\V16\Villa\Sections.dwg In the example above 2 plans, 4 elevations and 2 cross sections were generated and exported to 3 drawings. Upon the calculation of the first BIM Section in a project, a new project sheetset (6.27.31) is created, using the sheetset template pointed to by the **SectionSheetSetTemplateMetric** (default = BIM-section-metric.dst) or **SectionSheetSetTemplateImperial** (default = BIM-section-imperial.dst) user preference. The sheetset is stored in the same folder as the drawing. If the PROJECTNAME (**??**) system variable in the model drawing is not empty, it is used as the name for the generated SheetSet, else the name of the model drawing is used. The template drawing for the drawing generation process is defined by the **Sheet Template Layout** setting of the project sheetset.. Paper space view definitions that correspond with the location of the viewport are created and used to set up sheet views. The generated sheetset offers view categories per section type: **Plan** , **Section** , **Elevation** or **Detail** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/SheetSetsPa

NOTES

- BIM_SECTIONS_PREVIOUS layer: the result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer, which is frozen by default. Thawing the layer allows for a pragmatic and fast visual check of modifications
- BIM_VIEWPORTS layer: generated viewports are placed on the dedicated BIM_VIEWPORTS layer. If desired, this layer e.g. can be set as 'non-printing' in the template drawing.
- Using the same folder for the sheetset definition, models and generated drawings enables painless migration from one machine to the next: the SheetSet manager will automatically and silently adjust the file paths after the project folder has been relocated.

Figure 2.447


Sheet Sets
Modern Villa
VILLA ELEVATION
East Elevation
North Elevation
South Elevation
West Elevation
First floor plan
First floor plan
Roof plan
Ground floor plan
Cellar floor plan
Sheet Set
Name
Modern Villa
Description
section drawings
File path
C:\Users\jacob\Documents\Bricsys\Zendesk\Villa Model\project.dst
View label block
View Label (C:\Users\jacob\AppData\Local\Bricsys\BricsCAD\V19x64\en_US\Templates\Sl
Callout blocks
Number Bubble (C:\Users\jacob\AppData\Local\Bricsys\BricsCAD\V19x64\en_US\Templat
Total sheets
4
Project Control
Project number
Project name
Project phase
Project milestone
Sheet Creation
New sheet location
C:\Users\jacob\Documents\Bricsys\Zendesk\Villa Model
Sheet template layout
A0 (C:\Users\jacob\AppData\Local\Bricsys\BricsCAD\V19x64\en_US\Templates\Sheet-mn
Sheet Set Custom Properties
Client City
Client Name
Client State
Client Street
Project City
Project State
Project Street
Sheet Custom Properties
Checked By
Drawn By

To open a generated drawing

Do one of the following

- Place the cursor over a BIM Section callout or section line, then choose **Open Section Model** ( ) in the **Model** command group in the Quad.
- Double click the drawing in the sheet list in the SheetSet panel.

To Update a BIM Section

A red frame around the section result indicates that the BIM Section generated block is no longer in sync with the 3D model. Such section results can be updated without opening the 3D model. This can be done both in Model Space and in a Layout.

Update a BIM section in Model Space

1. Click the **Model** tab.
2. Place the cursor over the section block that needs to be updated.
3. Choose **Update Section** ( ) in the **Model** command group in the Quad.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMSectionUpdateModel.p

Update a BIM Section in a layout

1. Click the **Layout** tab.
2. Place the cursor over the viewport of the BIM Section that needs to be updated.
3. Choose **Update Section** ( ) in the **Model** command group in the Quad.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMSectionUpdateLayout.
The result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer. Thaw the layer to check the modifications. The content of the BIM_SECTIONS_PREVIOUS layer displays in magenta. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/BIMSectionUpdatePrevious.png}

Figure 2.448

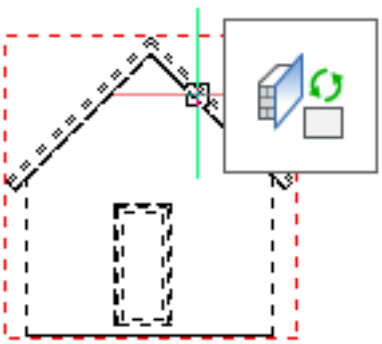


Figure 2.449

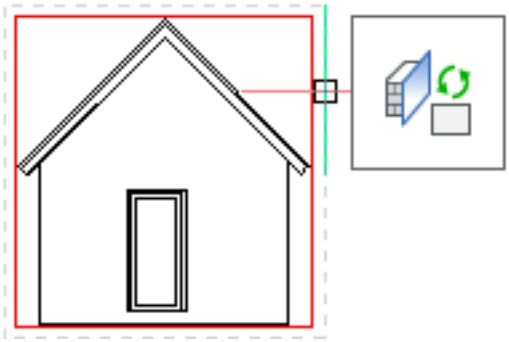


Figure 2.450



Chapter 3

BricsCAD Mechanical

3.1 Part Design - Workflows

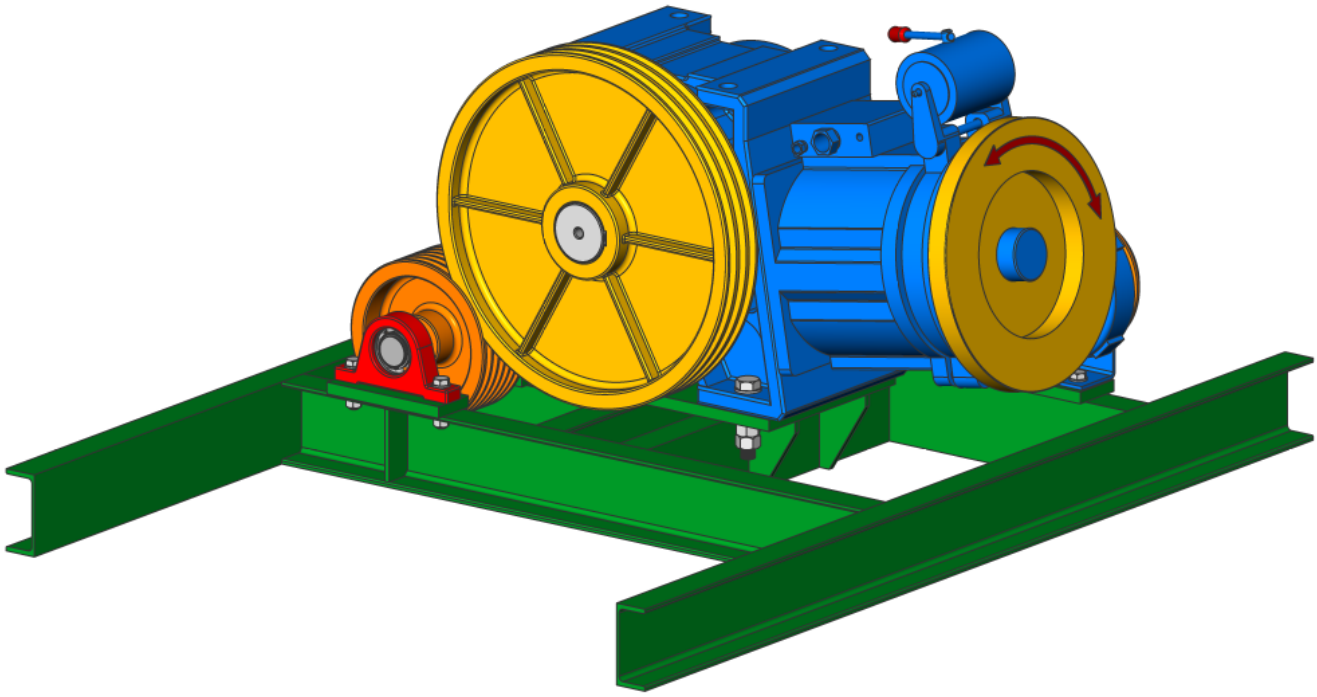
3.1.1 Deflector sheave editing

1. You need a **BricsCAD Platinum**license for this tutorial.
2. Open the **Geared_Traction_Machine_2_PMI_assembly.dwg**file from Geared_Traction_Machine_2_PMI_assembly.zip. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038879233/mceclip7.png})
3. Select **Geared_Traction_Machine_2_PMI** in theMechanical Browser (3.2.2).
4. Click **Open (6.23.8)** ( ) on the right-click menu.
5. Click the **Form Component (6.8.9)** ( ) tool buttonon the **Assembly** | **Modify**panel of theRibbon (1.13.6).
6. Select **Deflector Sheave**.
7. Select the **center point** of the front face of the **Deflector Sheave** as the base point of the component.
8. Save the file as **Deflector_Sheave_2.dwg**.
9. Select **Deflector_Sheave_2** in theMechanical Browser (3.2.2).
10. Click**Open (6.23.8)** ( )on the right-click menu. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachr
11. In the **Layers** drop-down menu on the **Home** | **Layers**panel of the Ribbon (1.13.6), select layer **"0"** and click **Color**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038879933/mceclip10.png})
12. Change the color to **40** and click the **OK** button. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachme



13. Switch to **Front** view in the LookFrom (6.20.39) widget by clicking the bottom triangle.
14. Make sure that the **Select Faces** option in the Settings (6.27.23) dialog is switched on (the**Select Faces** ( ) button on the **Home** | **Setting**panel of the Ribbon (1.13.6)).
15. Using a Green Crossing Window Selection (1.23.6), select the groves on the new deflector sheave. Use the CTRL key to change the selection method. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/
16. Delete the groves on the new deflector sheave.
17. Using the **Select same or less radius fillets (6.12.26)**( ) button on the **Solid** | **Select**panel of the Ribbon (1.13.6), select all fillets on the deflector sheave.
18. Delete all fillets.
19. Open the Deflector_Sheave.dwg file.
20. Click the **Copy with base point (6.9.51)**( ) tool button on the **Home** | **Edit**panel of theRibbon (1.13.6).
21. Enter **0,0,0** as the base point.
22. Select **Deflector Sheave** and hit **Enter**.

Figure 3.1



23. Switch to the **Deflector_Sheave_2.dwg**file.
24. Click the **Paste (6.24.10)** ( ) tool button on the **Home** | **Edit**panel in theRibbon (1.13.6).
25. Enter **-300,0,0** as the insertion point.
26. Switch to **Front** view in theLookFrom (6.20.39)widget by clicking the bottom triangle.
27. Using a Blue Window-Inside Selection Box (1.23.6) select the lateral faces of the deflector sheave. Use the CTRL key to change the selection type. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360
28. Select **Move (6.12.17)** ( ) from the **Model** section of theQuad (2.2.3)**.**
29. Move the faces left, enter a distance of **12** and hit **Enter**.
30. Select the left face of the internal part of the wheel.
31. Select **Solid Extrude (6.12.13)** ( ) from the **Model** section of theQuad (2.2.3)**.**
32. Move the face right, enter a distance of **10** and hit **Enter**.
33. Select the left face of the external part of the wheel.
34. Click the **Same area faces (6.12.26)** ( ) on the **Solid** | **Select**section of the Ribbon (1.13.6).
35. Select **Push/Pull (2.4.12)** ( ) from the **Model** section of theQuad (2.2.3)**.**
36. Move the mouse cursor left.
37. Enter a distance of **17** and hit **Enter**.
38. Using the**Same area faces (6.12.26)** ( )button select the 2 internal cylindrical faces of the deflector sheave. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038999753/mceclip2.png})
39. Switch to **Left** view in theLookFrom (6.20.39)widget by clicking the left triangle.
40. Select**Push/Pull (2.4.12)** ( )from the **Model** section of theQuad (2.2.3)**.**
41. Align with the internal cylindrical face of the original deflector sheave (or enter a distance of **234**) and hit **Enter**.
42. Select the external face of the new deflector sheave.
43. Select**Push/Pull (2.4.12)** ( )from the **Model** section of theQuad (2.2.3)**.**
44. Align with the external face of the original deflector sheave (or enter a distance of **262.5**) and hit **Enter**.
45. Switch to **Front** view in theLookFrom (6.20.39)widget by clicking the bottom triangle.

Figure 3.2

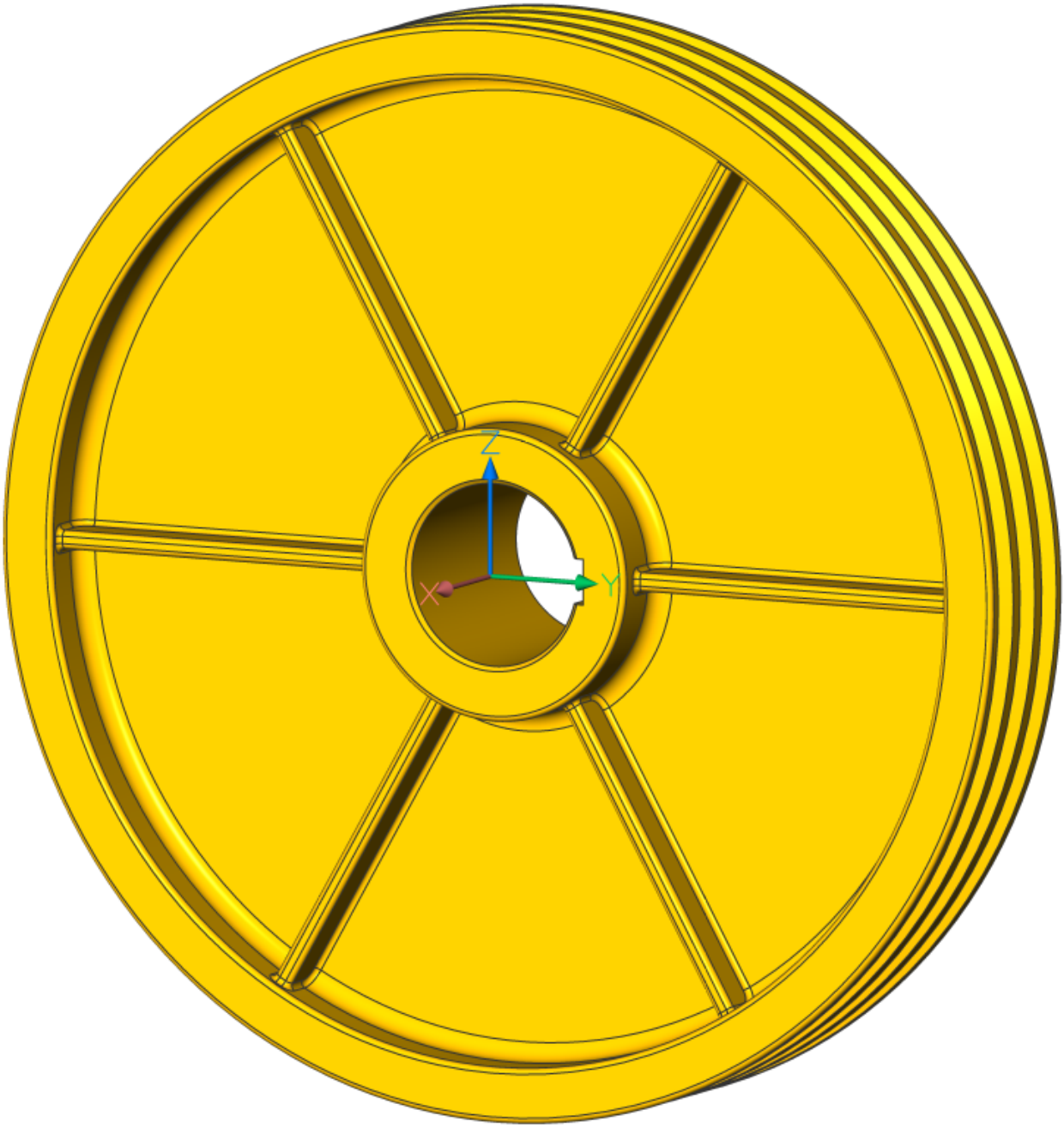


Figure 3.3

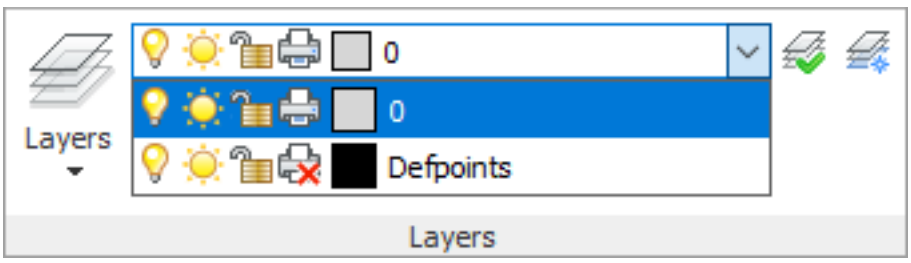


Figure 3.4

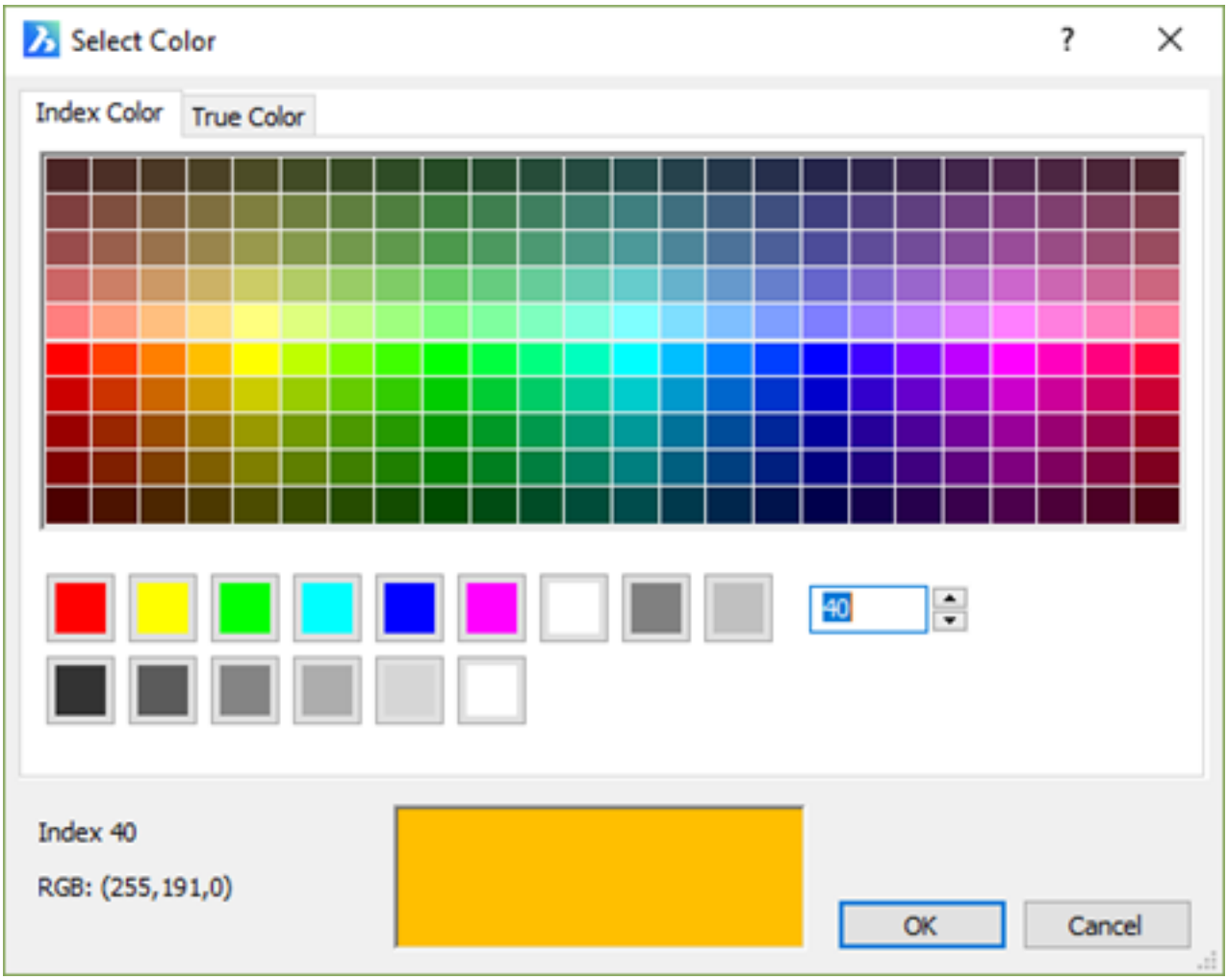
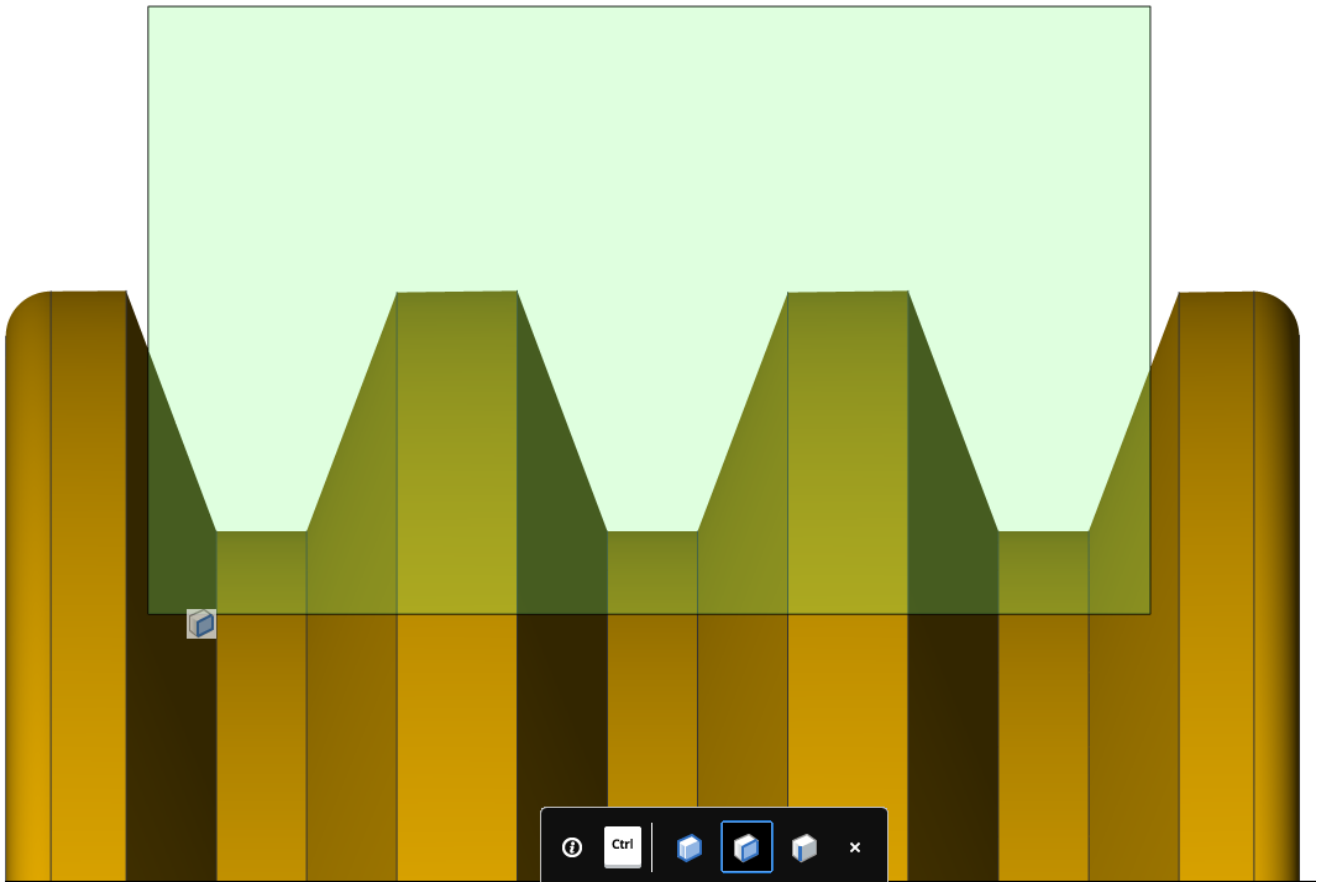

Select Color
Index Color
True Color
40
Index 40
RGB: (255,191,0)
OK
Cancel

Figure 3.5



46. Make sure that the **Snap to center** option in the Settings (6.27.23)dialog is switched on (Snap to center (6.9.3) ( ) button on the **Home** | **Entity Snaps** panel of the Ribbon (1.13.6)).
47. Repeat step 15 for select faces of groves on the original deflector sheave. (\vref {fig:C:/Dev/Task-2448/images/helpbrics
48. Click the **Copy Faces (6.12.8)** ( ) from the **Model** section of theQuad (2.2.3)**.**
49. Select the **center point** on the left face of the original deflector sheave as the base point (hit SHIFTto fix the dynamic UCS).
50. Select the center point on the left face of the new deflector sheave as the insertion point (hitSHIFTto fix the dynamic UCS). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039000213/mceclip5.pn
51. Delete the original deflector sheave. This is no longer needed.
52. Make sure the **Select edges** option in Settings (6.27.23)dialog is switch on (**Select Edges** ( ) button tools on the **Home** | **Setting**panel of the Ribbon (1.13.6)).
53. Select the edges as shown in the picture below and select **Fillet (6.12.14)** ( ) from the **Model** section of the-Quad (2.2.3)**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038160154/mceclip8.**

Figure 3.6

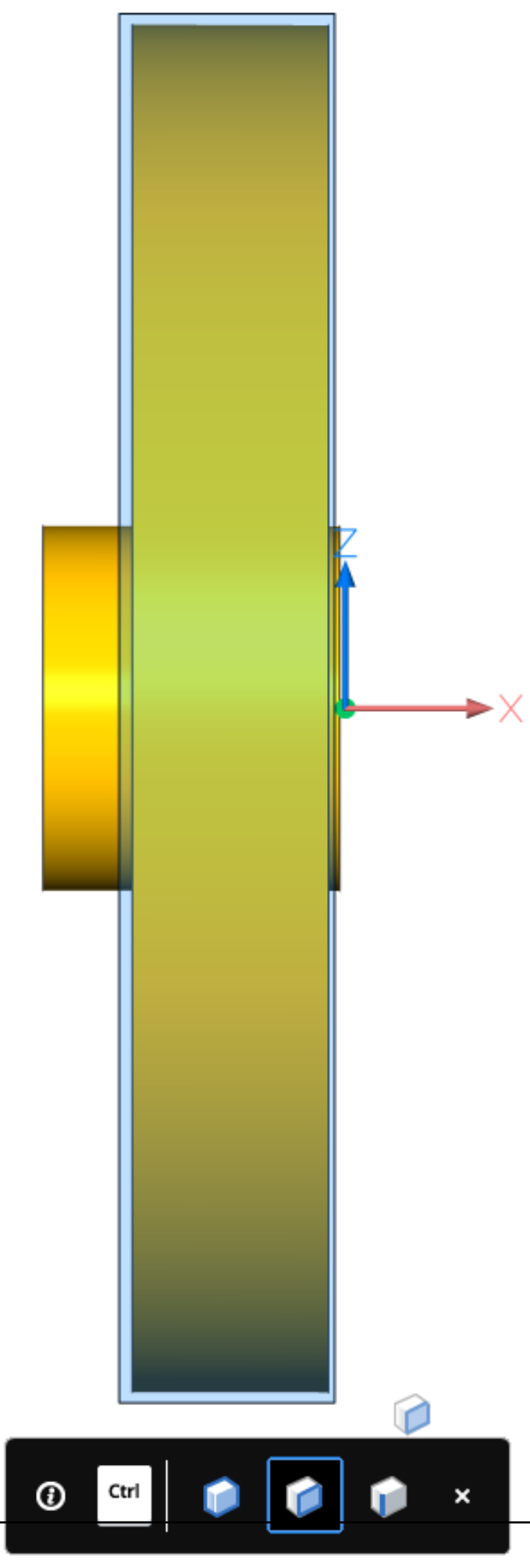


Figure 3.7

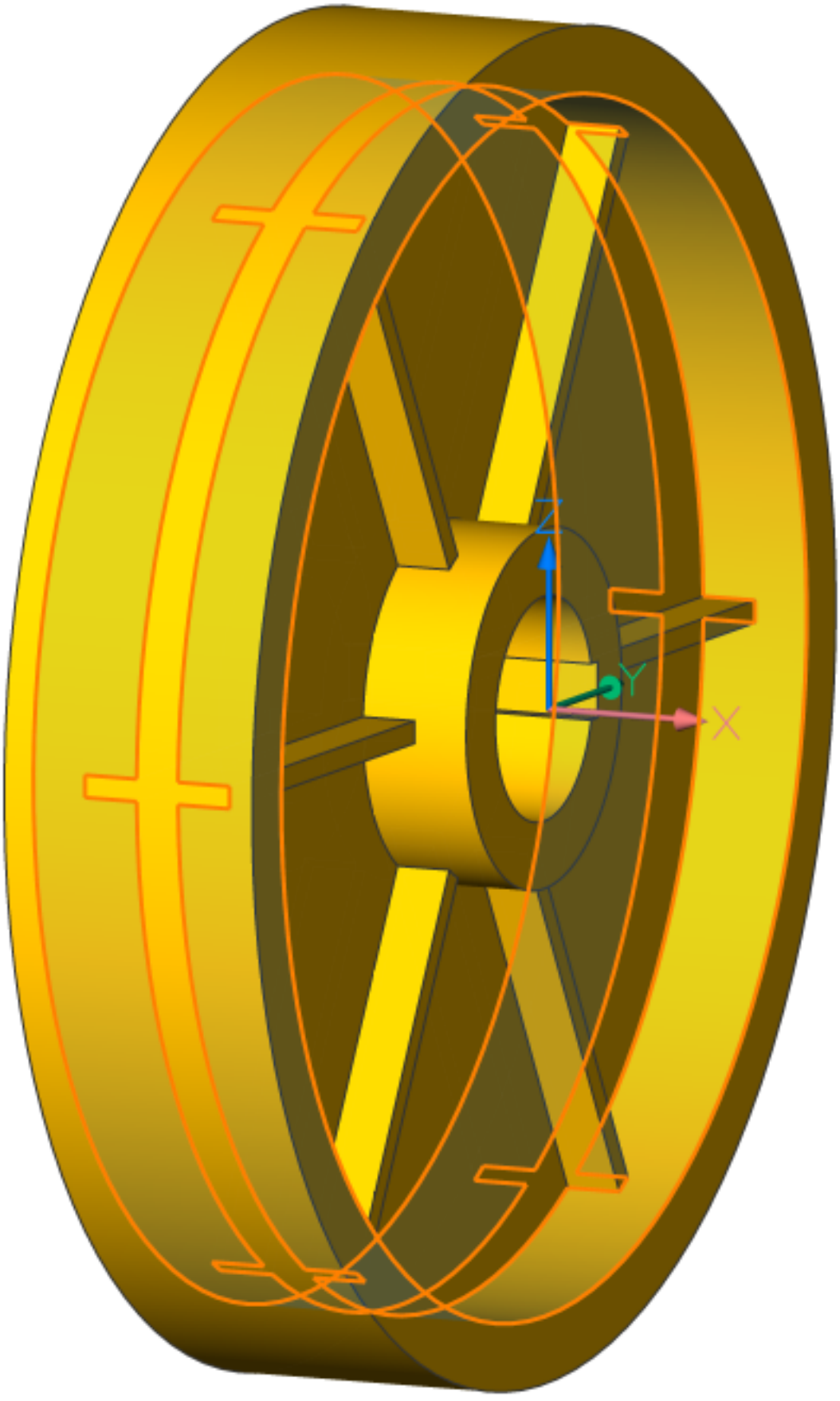


Figure 3.8

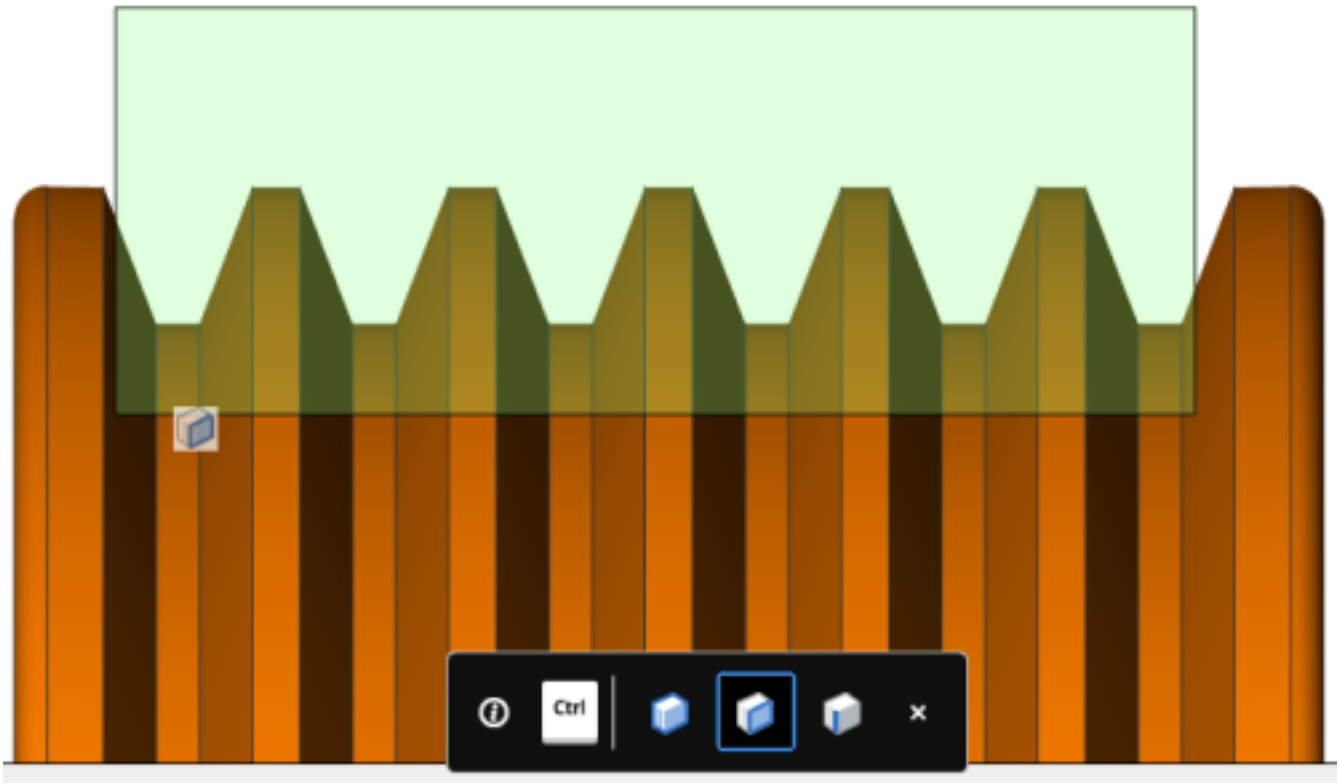
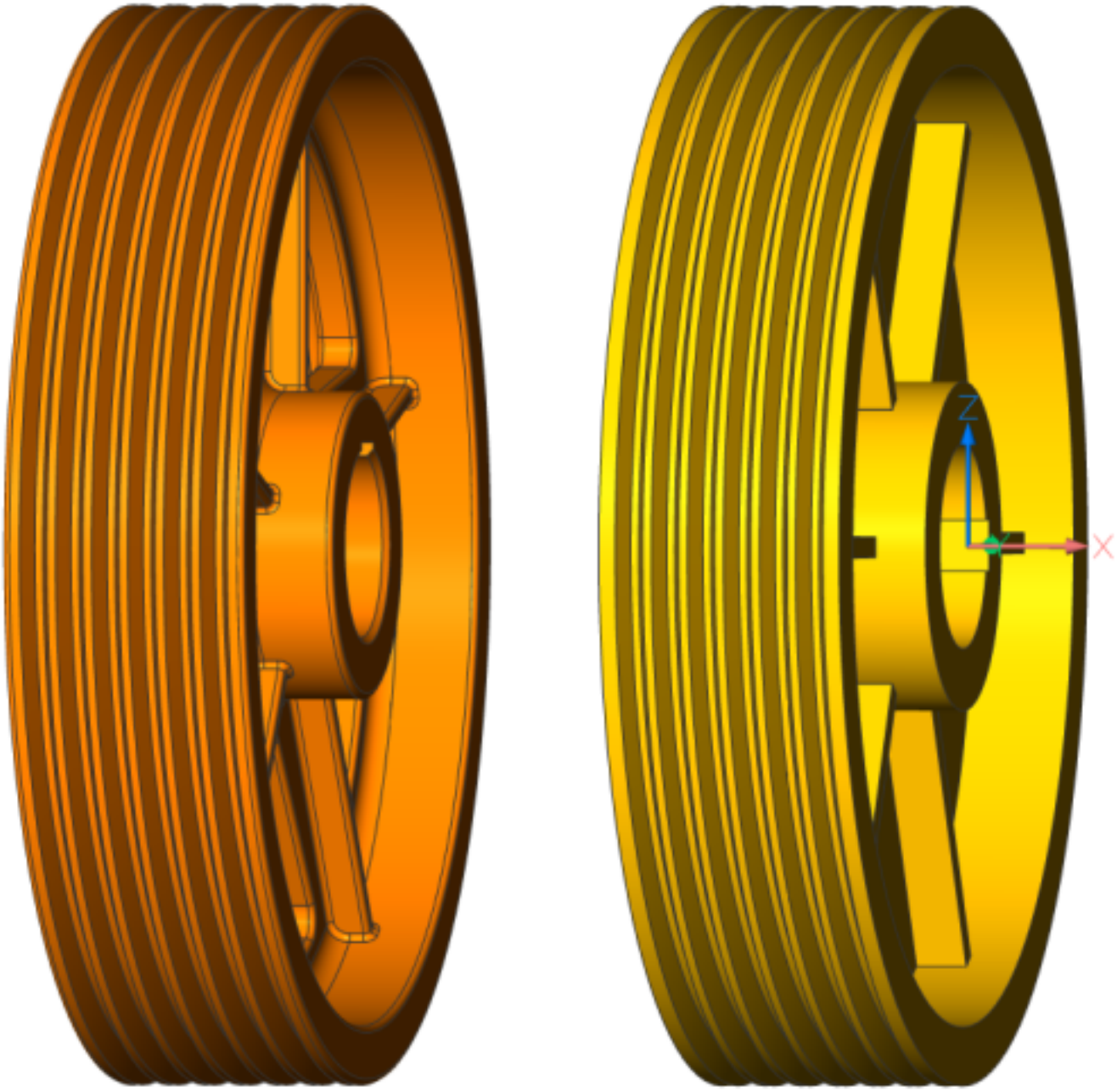


Figure 3.9



54. Enter a radius of **10** and hit **Enter**.
55. Select the edges as shown in the picture below and select**Fillet (6.12.14)** ( )from the **Model** section of the-Quad (2.2.3). **(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038160114/mceclip7.**
56. Enter a radius of **3** and hit **Enter**.
57. Make sure that the **Snap to geometric center** option in the Settings (6.27.23) dialog is switched on (**Snap to geometric center (6.15.4)** ( ) button on the **Home** | **Entity Snaps**panel of the Ribbon (1.13.6)).
58. Draw a circle in the geometric center of one of the sectors between the ribs using Circle (6.9.20) ( ) from the **Draw** section of theQuad (2.2.3)**.**
59. Enter a radius of **50** and hit **Enter**.
60. Select the circle that was just created. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360
61. Select**Solid Extrude (6.12.13)** ( )from the **Model** section of theQuad (2.2.3)**.**
62. Enter a distance of **20**with **Subtract** option and hit **Enter**.
63. Select the hole edges and select**Fillet (6.12.14)** ( )from the**Model** section of theQuad (2.2.3)**.**
64. Enter a radius of **3** and hit **Enter**.
65. Select the depression that was just created.
66. Select the**Copy Faces (6.12.8)** ( )on the **Solid** | **Edit**panel of the Ribbon (1.13.6).
67. In the **DMCOPYFACES** menu select the **Multiple** option. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articlea
68. Select the **geometric center** of the sector as the base point of copying (hit SHIFTto fix the dynamic UCS).
69. Select the **geometric center** of all other sectors as points of insertion. (\vref {fig:C:/Dev/Task-2448/images/helpbricsysco
70. Select the **Deflector Sheave** and change **Color** on the Properties panel (1.12.4) to **RGB:255,191,0**.
71. Save the Deflector_Sheave_2.dwg file.
72. Switch to the **Geared_Traction_Machine_2_PMI.dwg**file.
73. Select **Deflector_Sheave_2** in theMechanical Browser (3.2.2).
74. Click **Update (6.12.33)** ( ) on the right-click menu.
75. Save **Geared_Traction_Machine_2_PMI.dwg**file. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachm
76. Switch to the **Geared_Traction_Machine_2_PMI_assembly.dwg**file.
77. Select **Geared_Traction_Machine_2_PMI** in theMechanical Browser (3.2.2).
78. Click**Update (6.12.33)** ( )on the right-click menu. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleatta

Figure 3.10

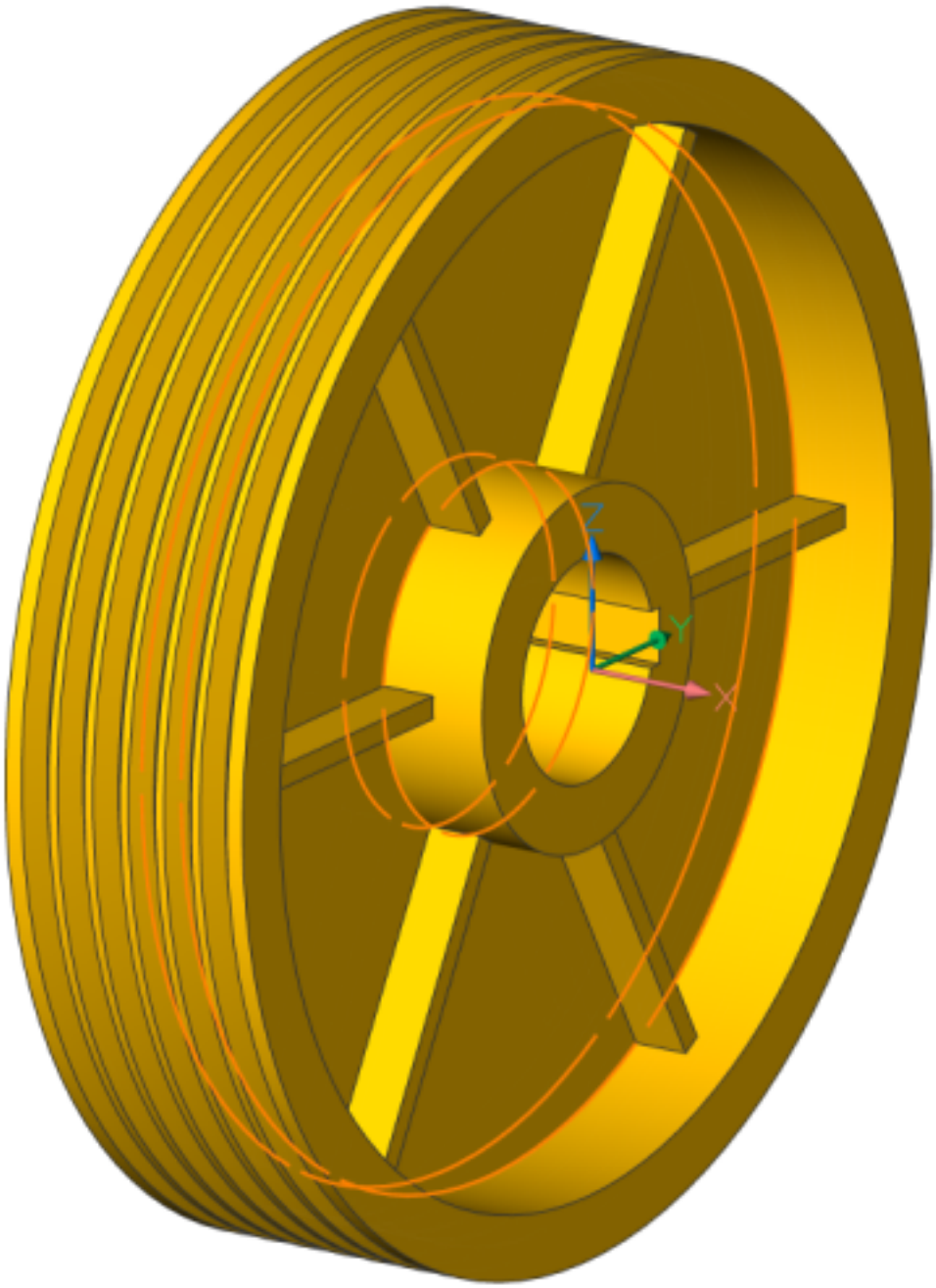


Figure 3.11



Figure 3.12

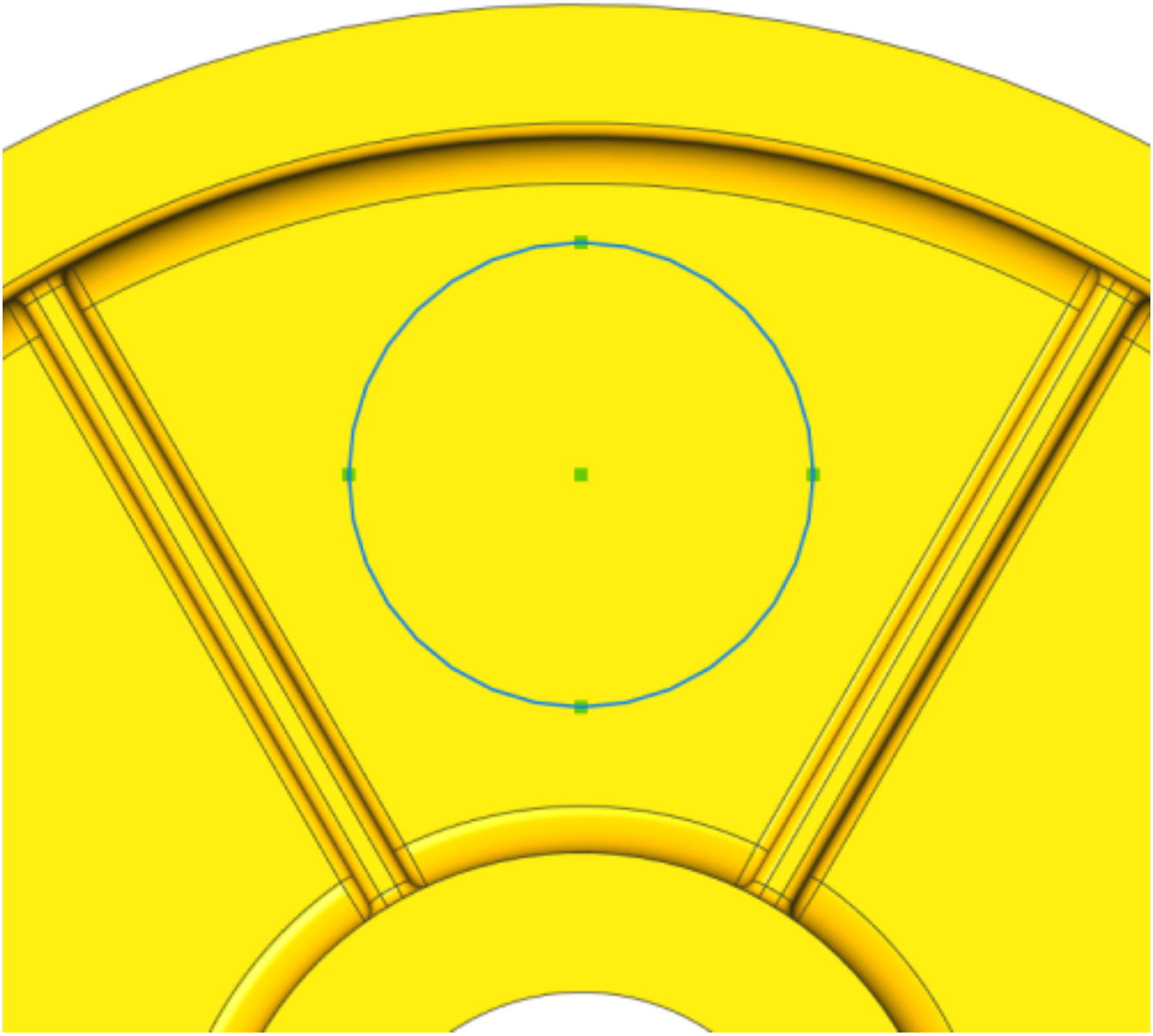


Figure 3.13

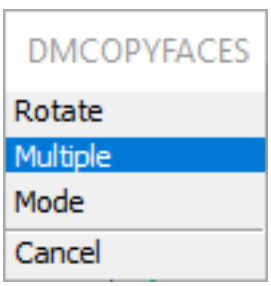


Figure 3.14

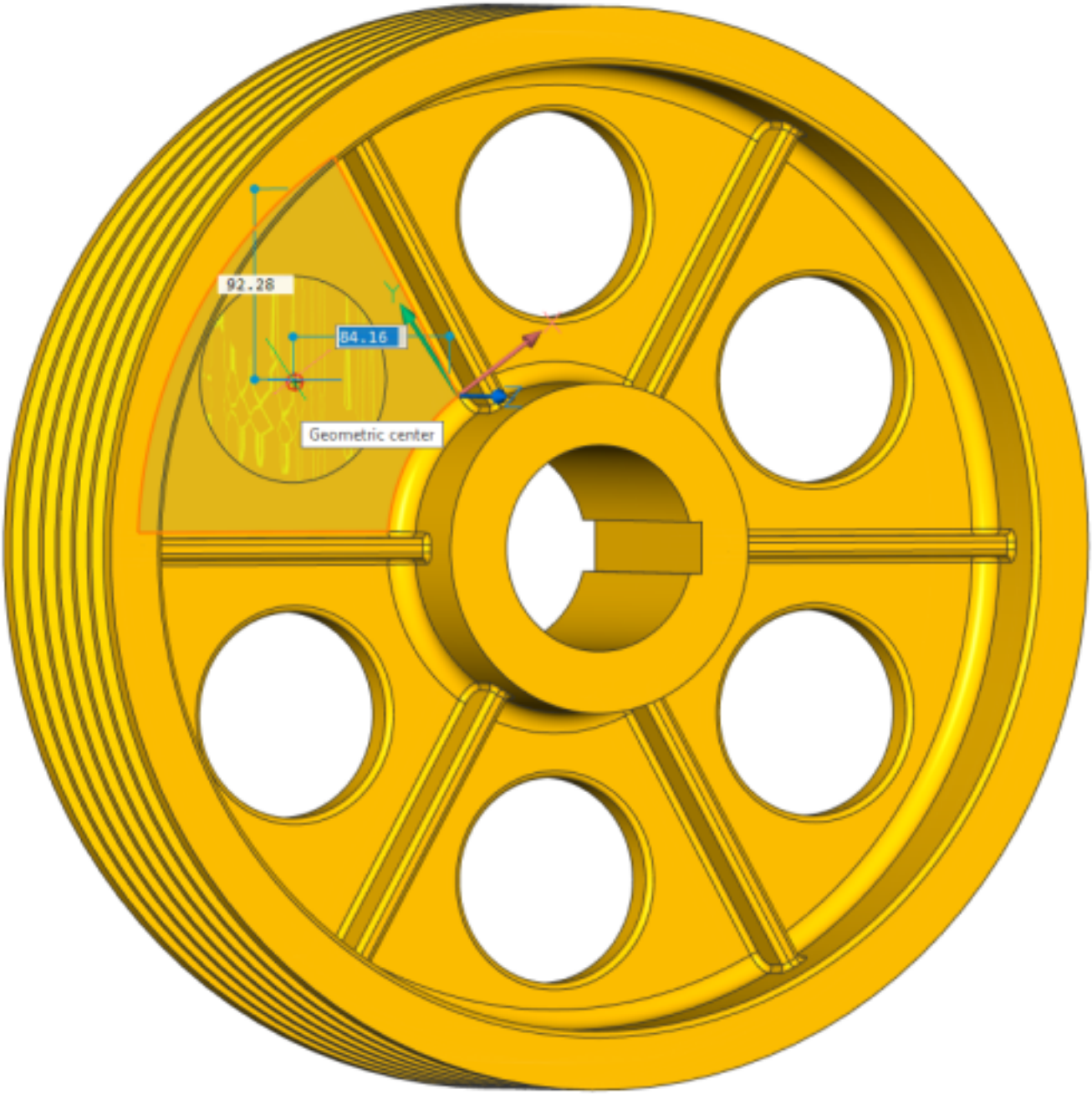


Figure 3.15

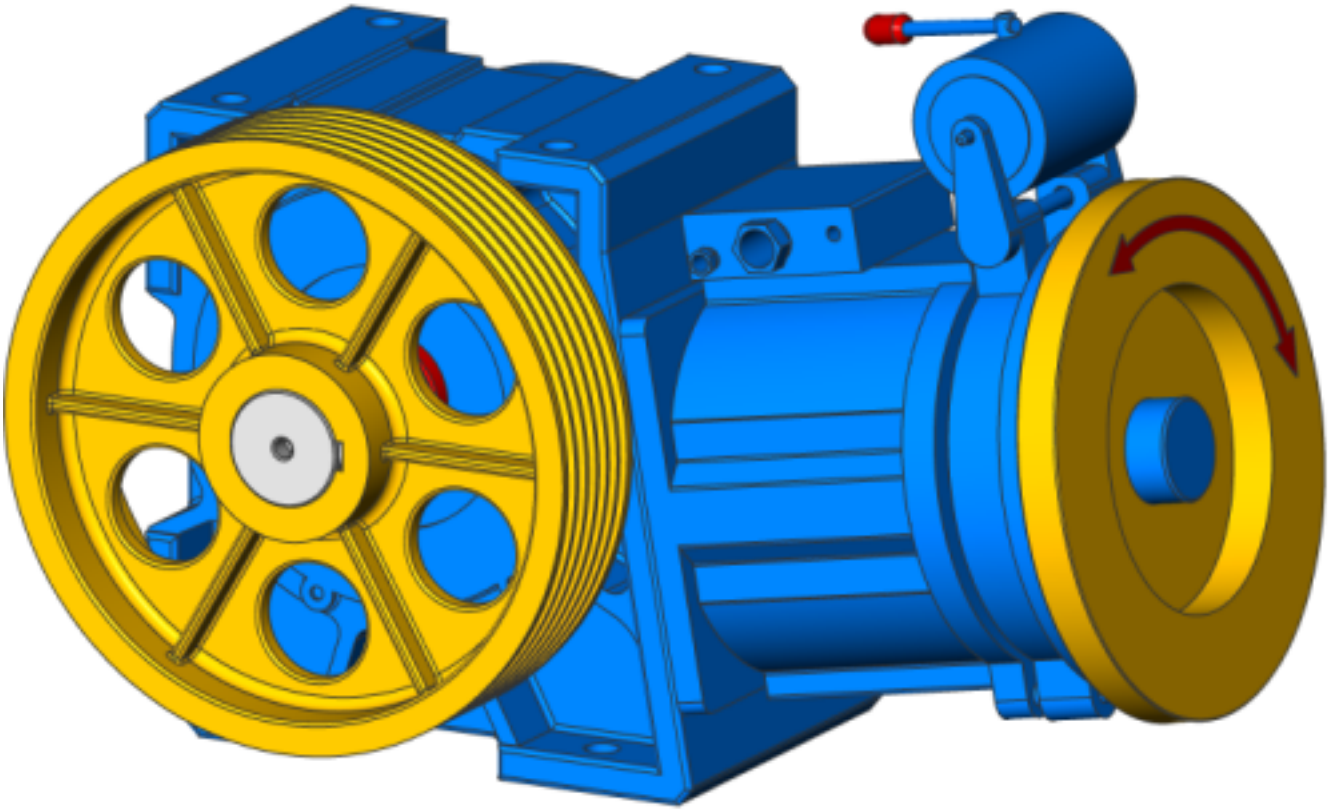
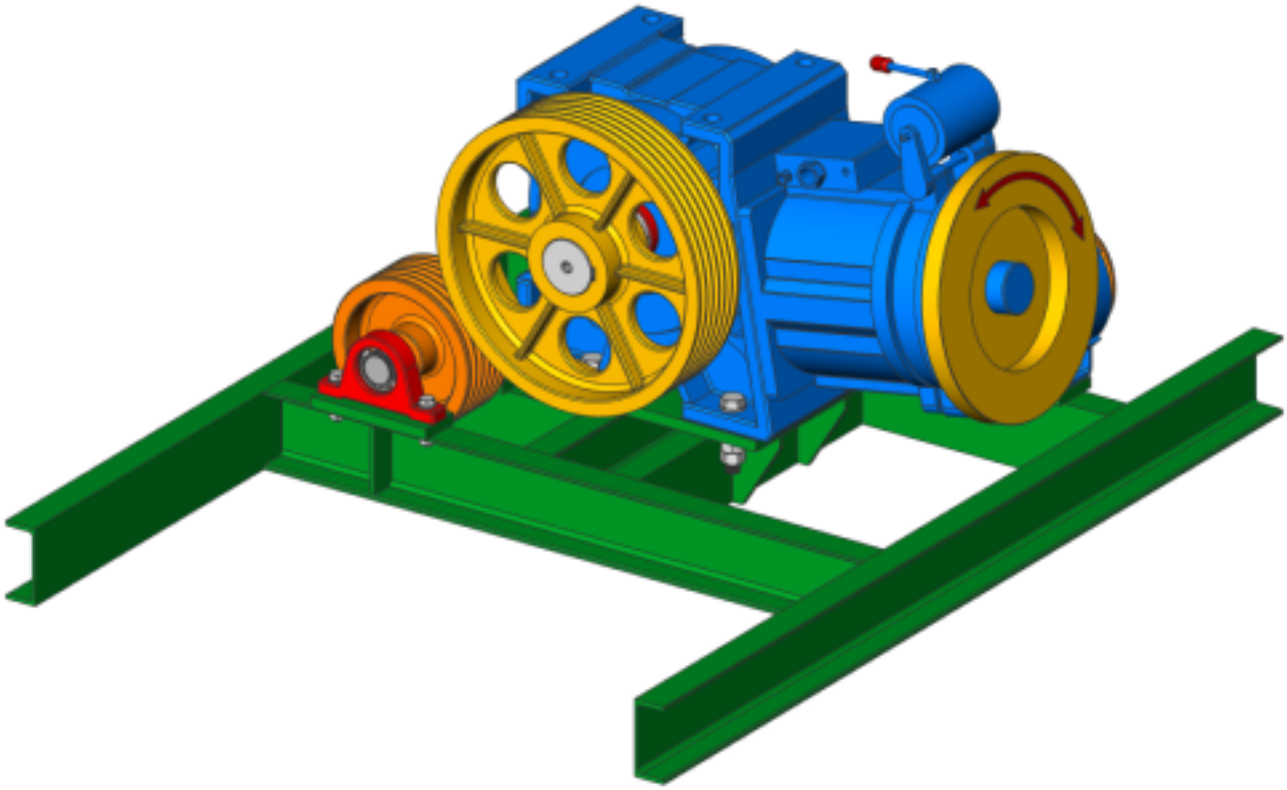


Figure 3.16



3.1.2 Shaft support editing

3.1.3 Step 1. Resize

1. You need a**BricsCAD Platinum**license for this tutorial.
2. Open the **Shaft_support.dwg** file from Shaft_support_files.zip.
3. Select **Bearing_support** in the Mechanical Browser (3.2.2).
4. Click**Open (6.23.8)** ( )on the right-click menu.
5. Click the **Same Radius fillets (6.12.26)** ( ) button on the **Solid** | **Select** panel in the Ribbon (1.13.6)and click any fillet face. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039112573/mceclip0.p
6. Delete all fillets with the**Erase (6.13.8)** ( )button on the **Solid** | **Edit** panel in the Ribbon (1.13.6)or the **Modify** section of the Quad (2.2.3).
7. Delete the chamfer on the front cylindrical edge with the **Erase (6.13.8)** ( ) button in the **Solid** | **Edit** panel in the Ribbon (1.13.6)or the **Modify** section of the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/ar
8. Select the two inclined faces. Select**Push/Pull (2.4.12)** ( ) from the **Model** section of theQuad (2.2.3)**.**
9. Enter a distance of **10** and hit **Enter.** (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36003
10. Select the outer top cylindrical face. Select **Push/Pull (2.4.12)** ( ) from the **Model** section of theQuad (2.2.3)**.**
11. Enter a distance of **70** and hit **Enter**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36003
12. Select the larger of the two internal cylindrical faces. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleatta
13. Select**Push/Pull (2.4.12)** ( ) from the **Model** section of theQuad (2.2.3)**.**
14. Enter a distance of **55** and hit **Enter**.
15. Select the smaller of the two internal cylindrical faces. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleat
16. Select**Push/Pull (2.4.12)** ( ) from the **Model** section of theQuad (2.2.3)**.**
17. Enter a distance of **50** and hit **Enter**.
18. Increase the Overall Depth: select the front and back vertical faces. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom
19. Select**Push/Pull (2.4.12)** ( ) from the **Model** section of theQuad (2.2.3)**.**
20. Enter a distance of **3.5** and hit **Enter**.
21. Increase the place for the bearing: select the inner vertical circular face. Select**Push/Pull (2.4.12)** ( ) from the **Model** section of theQuad (2.2.3)**.**
22. Enter a distance of **3.5** and hit **Enter**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/3600

Figure 3.17

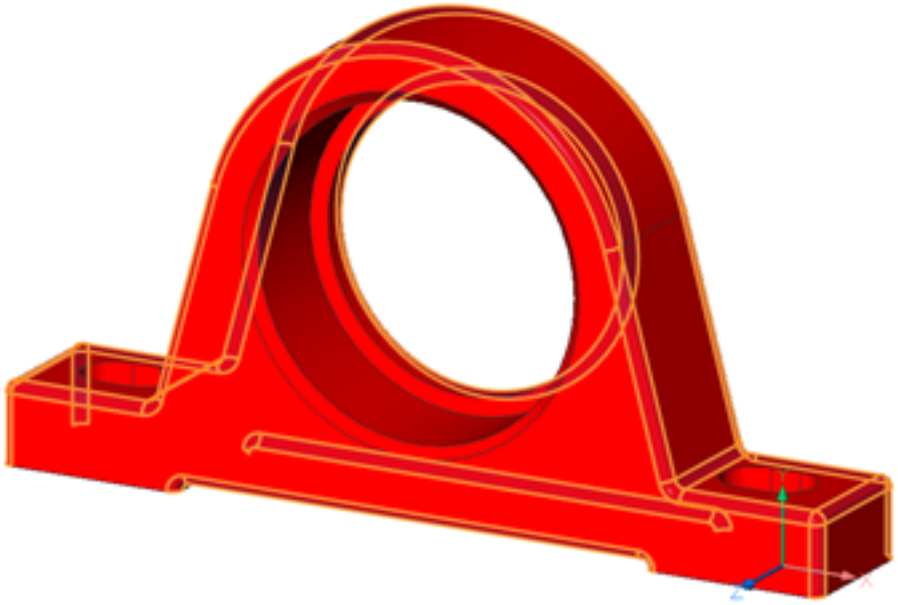


Figure 3.18

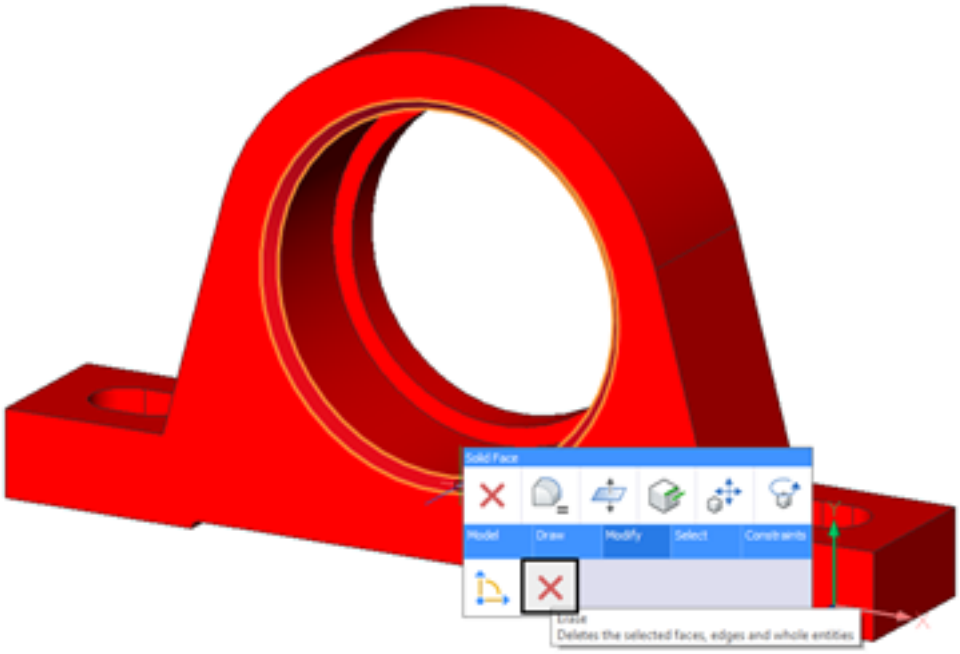


Figure 3.19

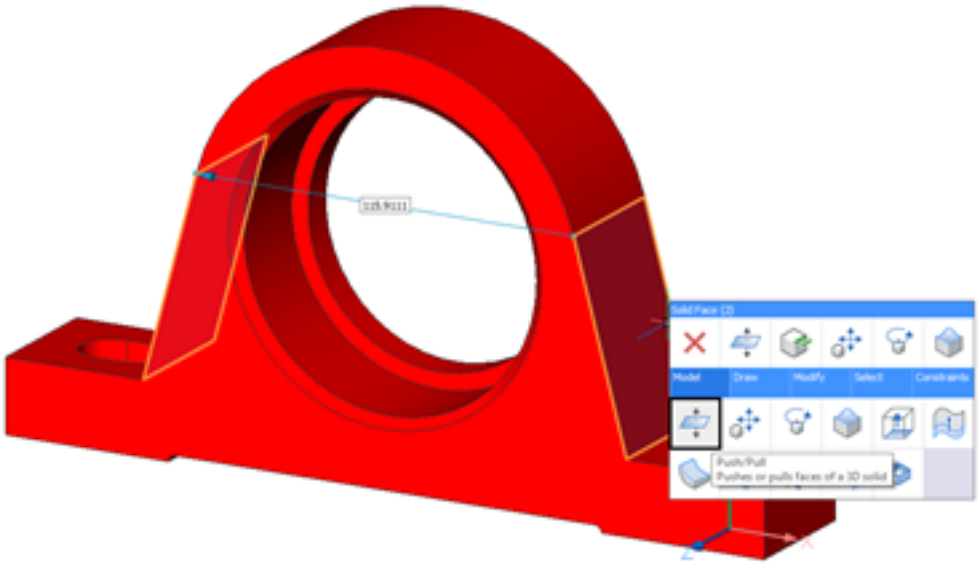


Figure 3.20

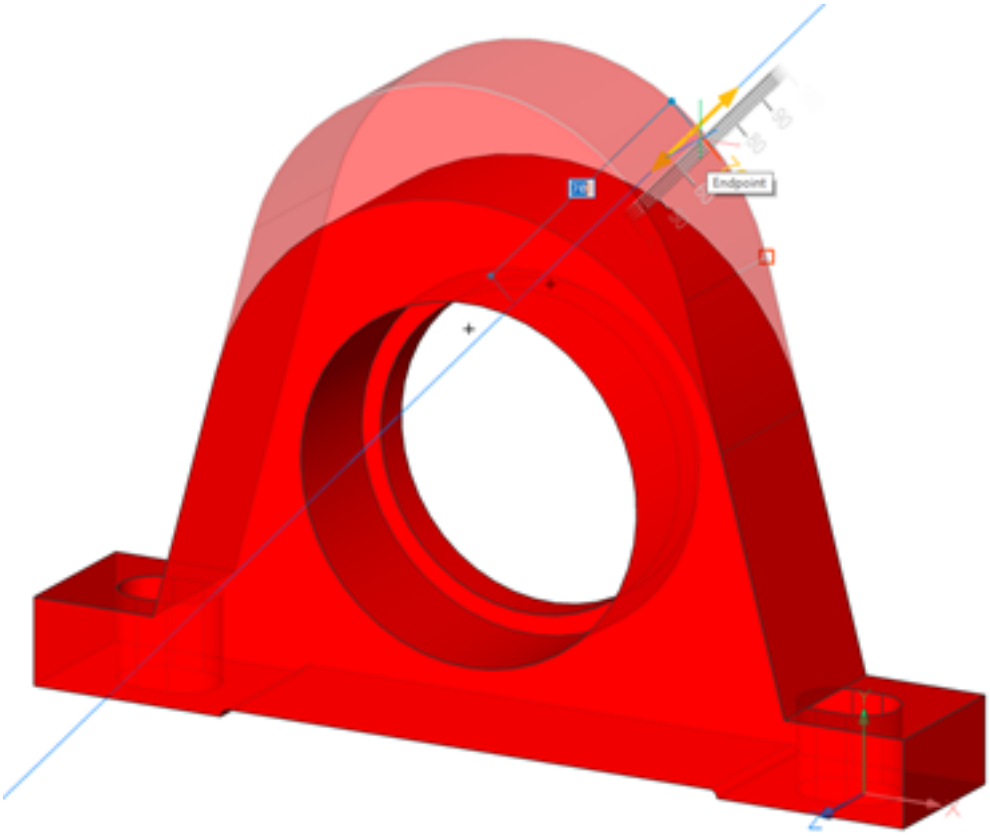


Figure 3.21

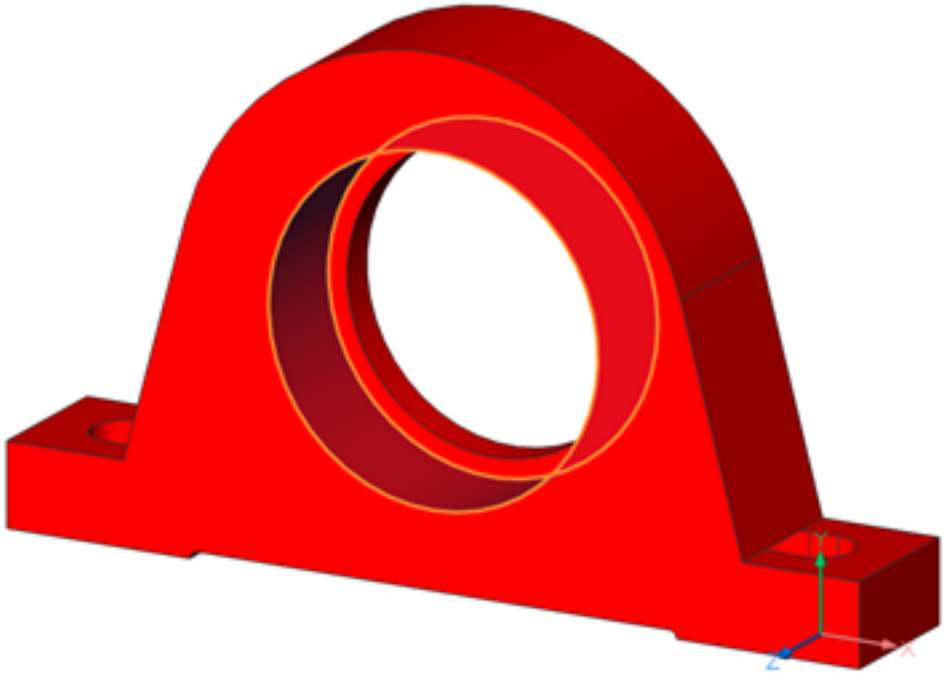


Figure 3.22

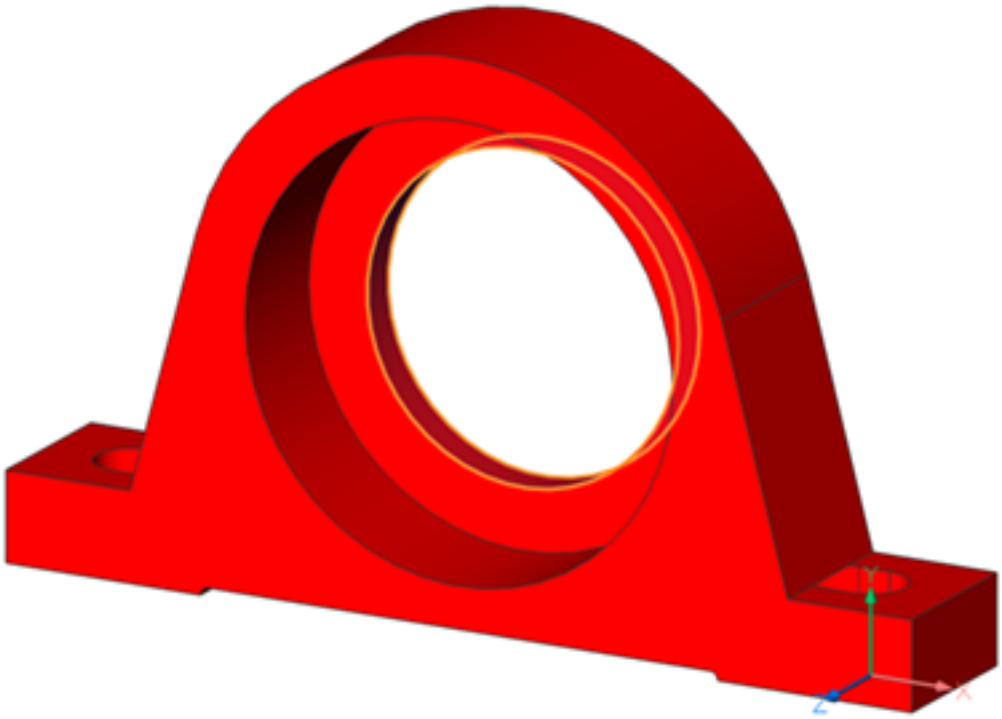


Figure 3.23

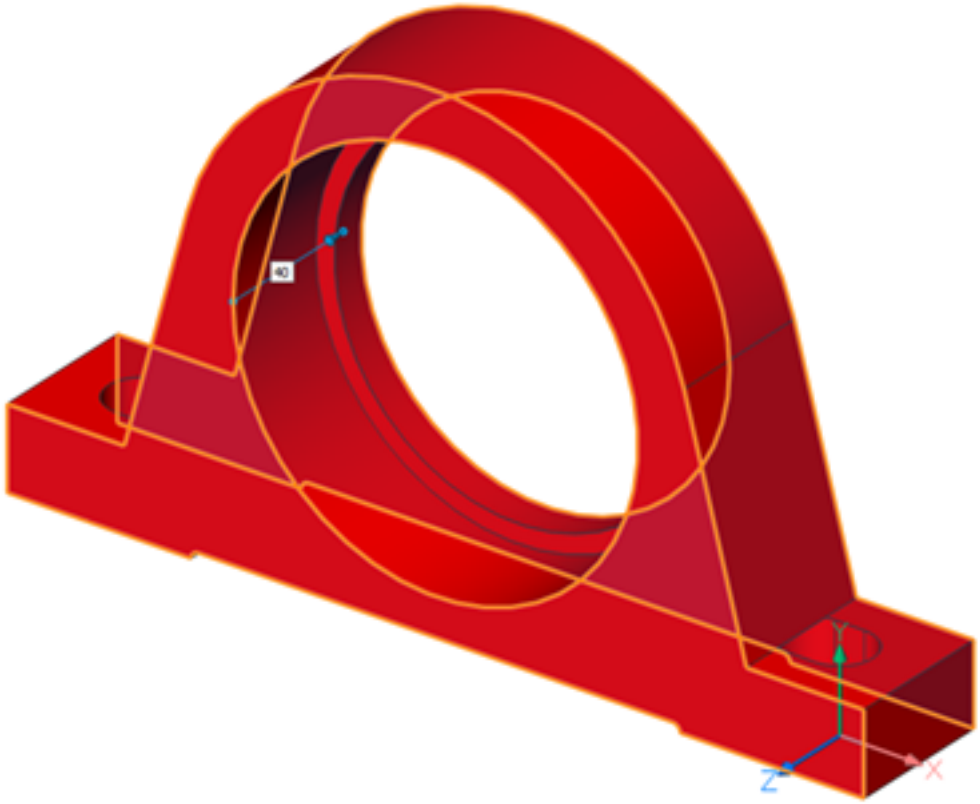
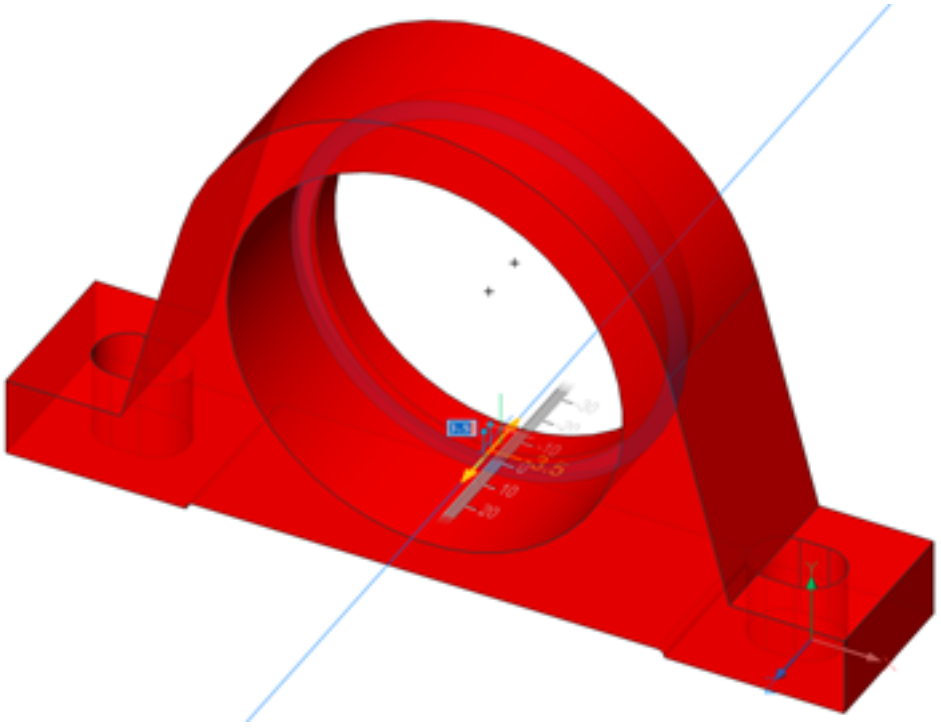


Figure 3.24

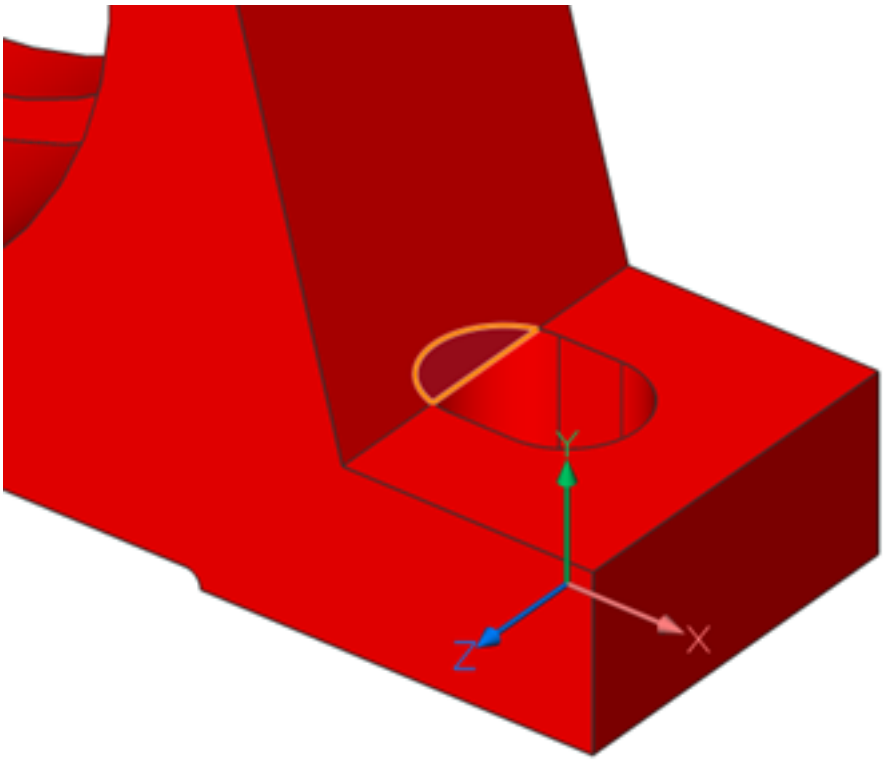


3.1.4 Step 2. Editing the base platform

1. Select the faces as shown in the picture below: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachme

2. Select**Solid Extrude (6.12.13)** () from the **Model** section of theQuad (2.2.3).Move the mouse up using the **Subtract** option. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038279314/mceclip3

Figure 3.25



3. Click the **Slice (6.27.36)** ( ) button on the **Solid** | **Edit** panel in the Ribbon (1.13.6), select the solid and hit **Enter**.
4. Select the **Object** option. Specify the top face of the supporting platform and hit **Enter**.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038279414/mceclip4.png}) (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038279434/mceclip5.png})
5. On one side, change the mounting cut to a distance of **18** using**Push/Pull (2.4.12)** ( ) from the **Model** section of theQuad (2.2.3)**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038279454/**
6. Select the right vertical edge on the front face. Select **Offset (6.23.3)**( ) from the **Draw** section of the Quad (2.2.3). Enter an offset distance of **50**, and select the front face for a parallel copy. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038279594/mceclip7.png})
7. Repeat for this on the back face.
8. Select the two new boundaries. Select**Solid Extrude (6.12.13)** ( ) from the **Model** section of theQuad (2.2.3)and move in an outward direction, with a distance of **8**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/a
9. Select the two bodies and select **Union (6.30.6)** ( ) from the **Model** section of the Quad (2.2.3).

Figure 3.26

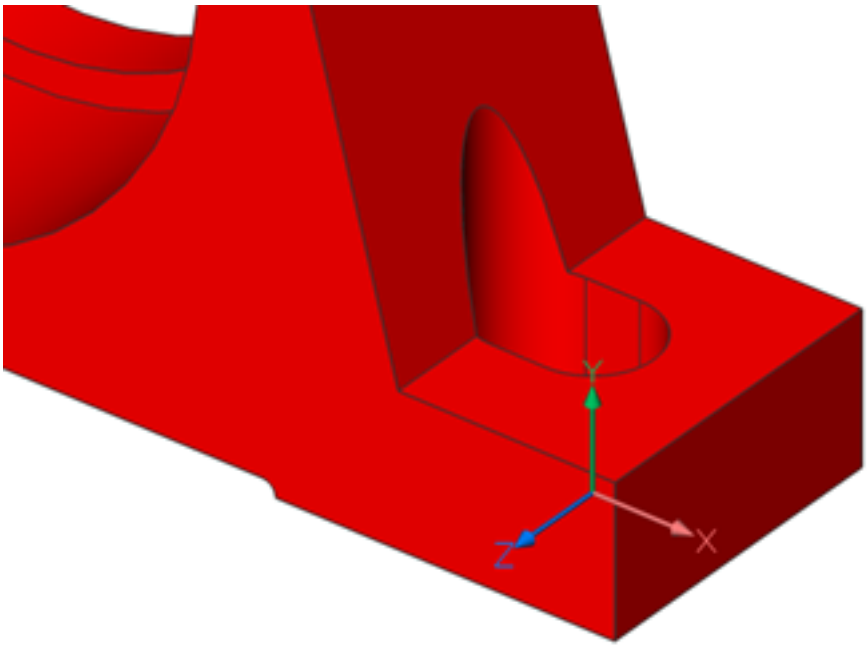


Figure 3.27

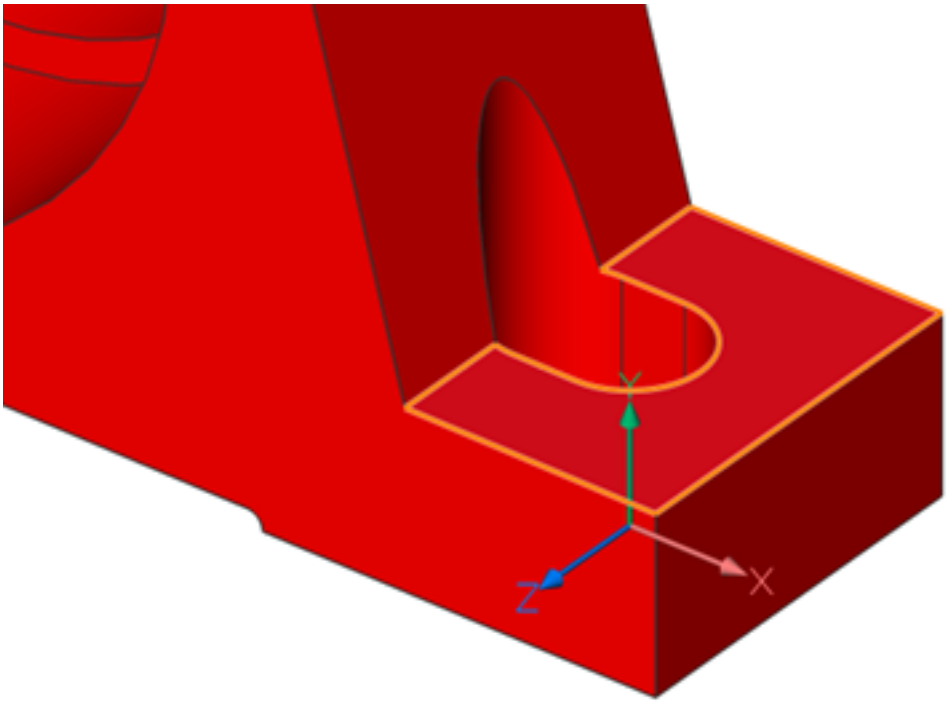


Figure 3.28

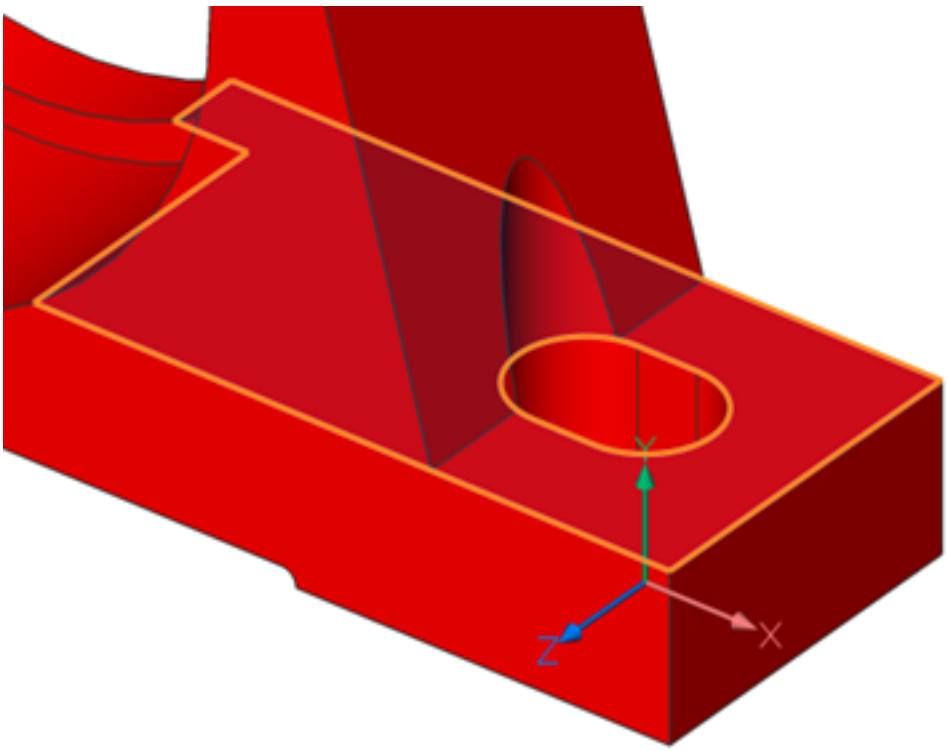


Figure 3.29

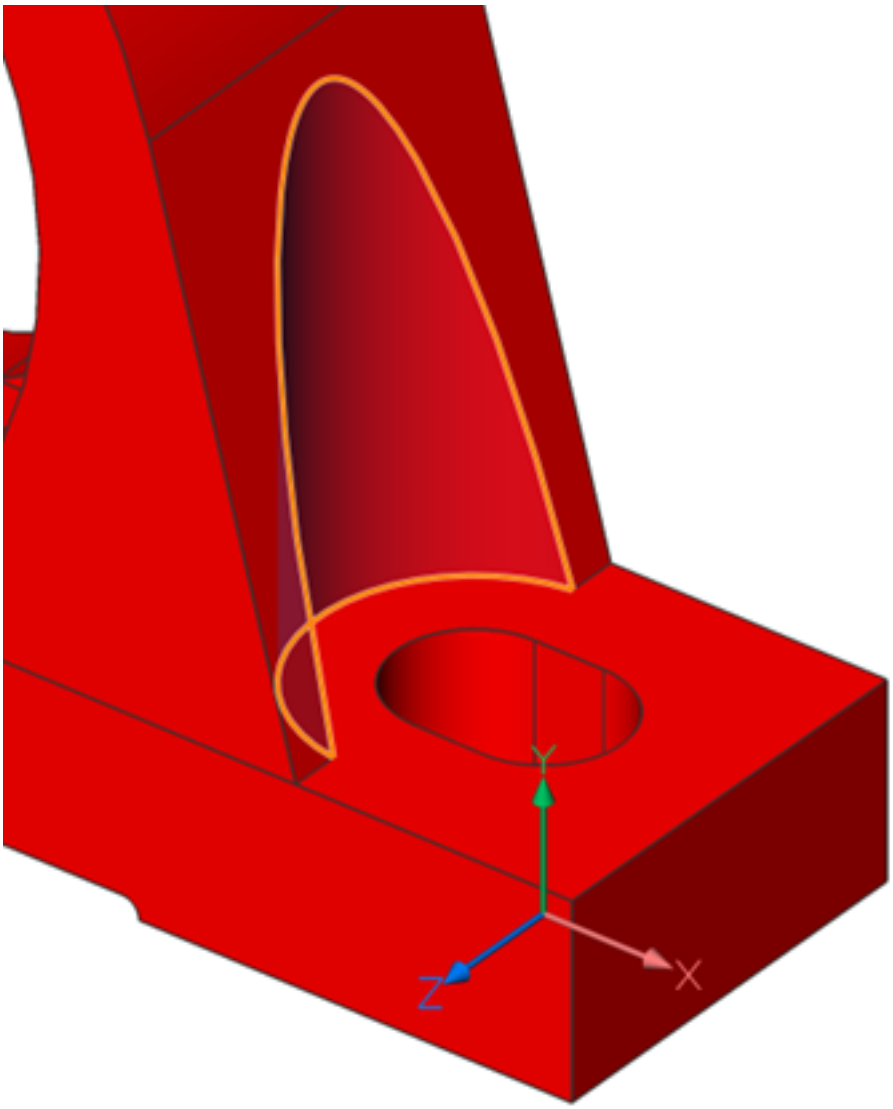


Figure 3.30

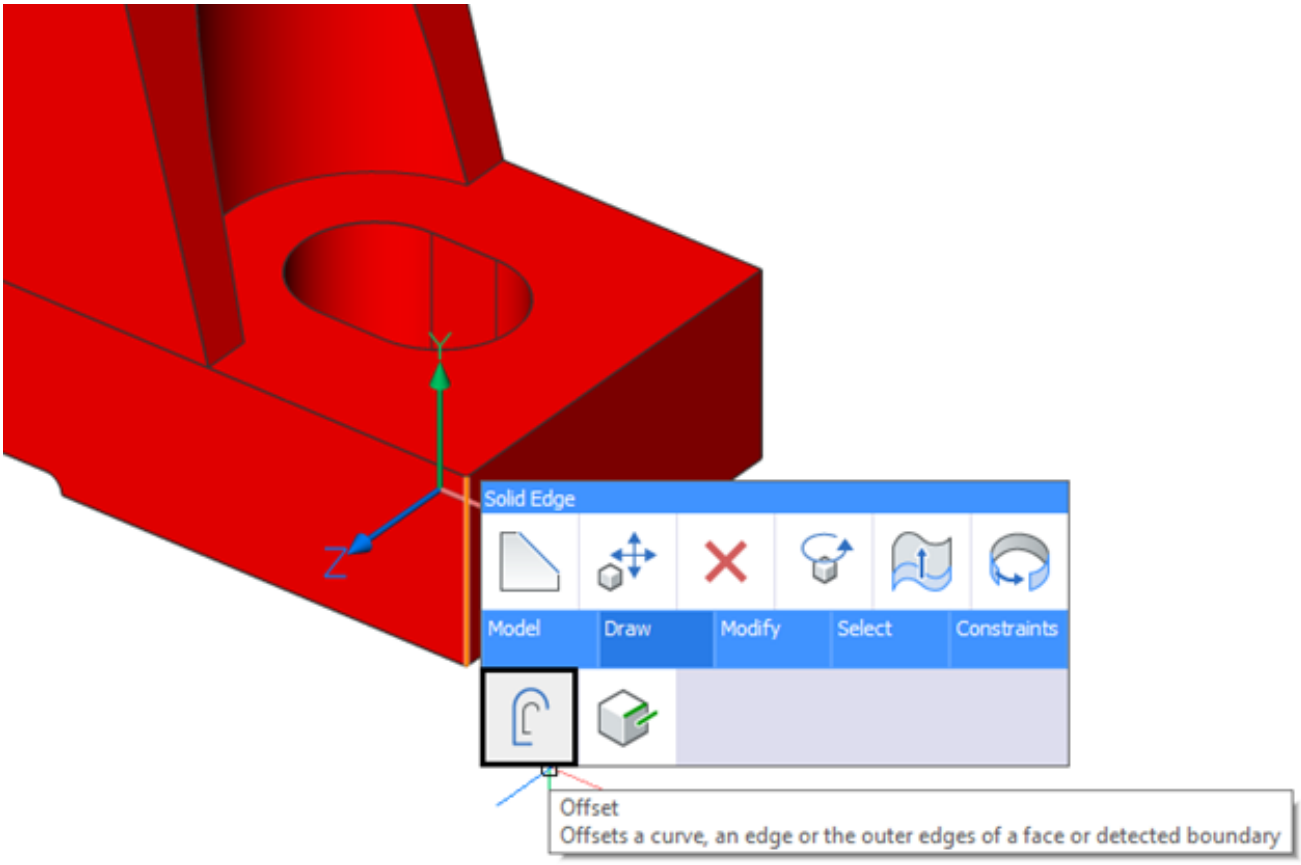
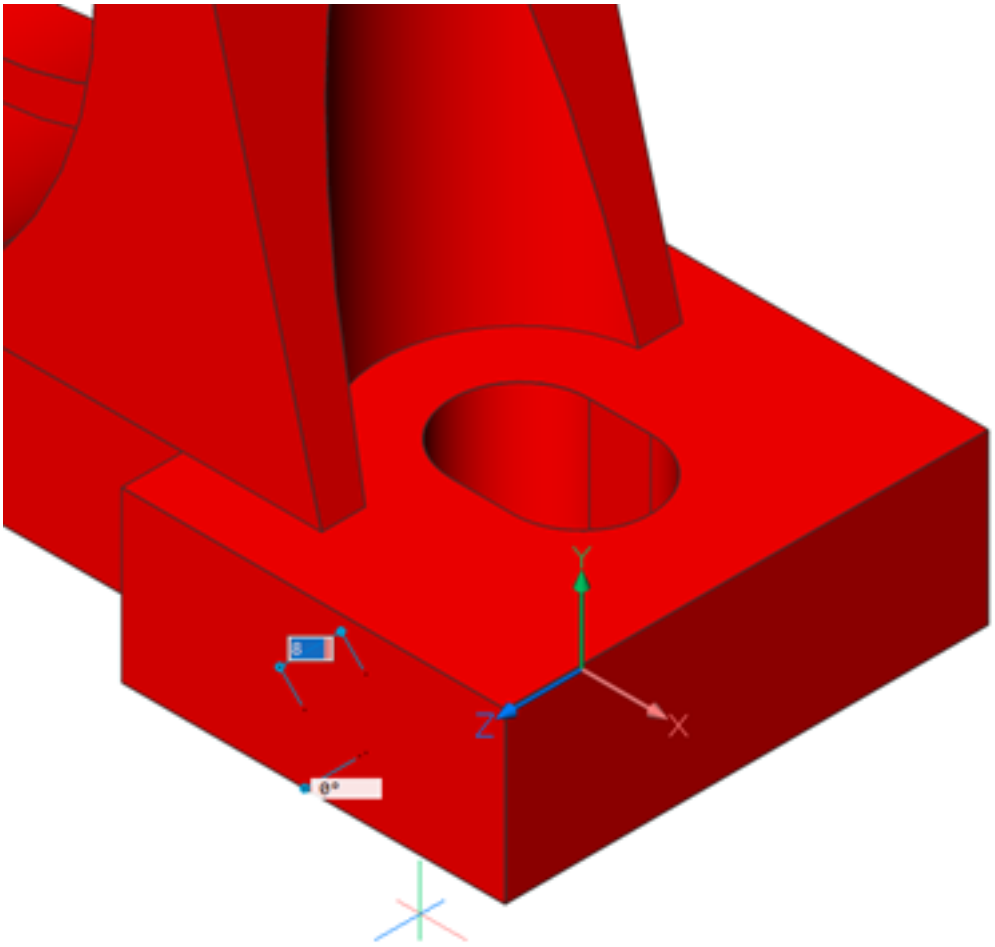

Solid Edge
Model
Draw
Modify
Select
Constraints
Offset
Offsets a curve, an edge or the outer edges of a face or detected boundary

Figure 3.31



3.1.5 Step 3. Copy faces

1. Select the faces of the updated base platform and then select**Copy Faces (6.12.8)** () from the **Model** section of theQuad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038283674/m

2. Select the **midpoint** of the right horizontal edge as a base point: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc

3. Select the **Rotate** option and enter a rotation angle of **180**.
4. Select the **midpoint** of the left horizontal edge as the insertion point. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscon

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038281954/mceclip9.png})

Figure 3.32

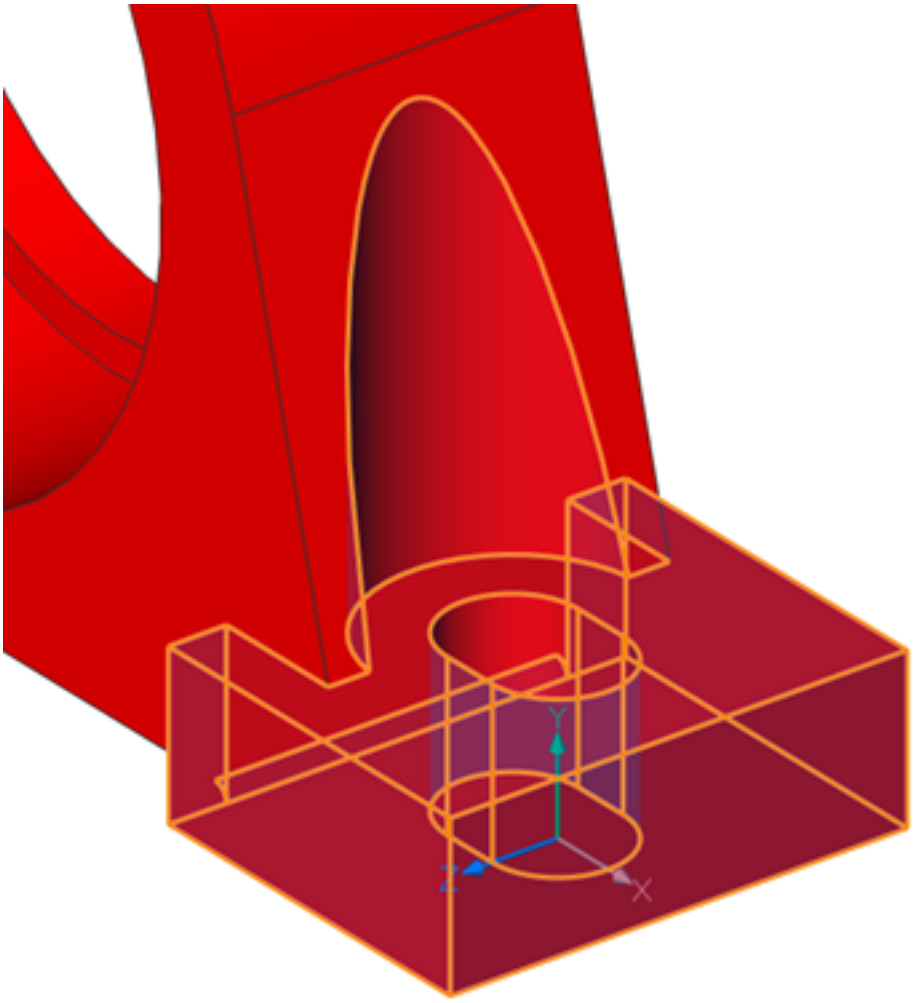


Figure 3.33

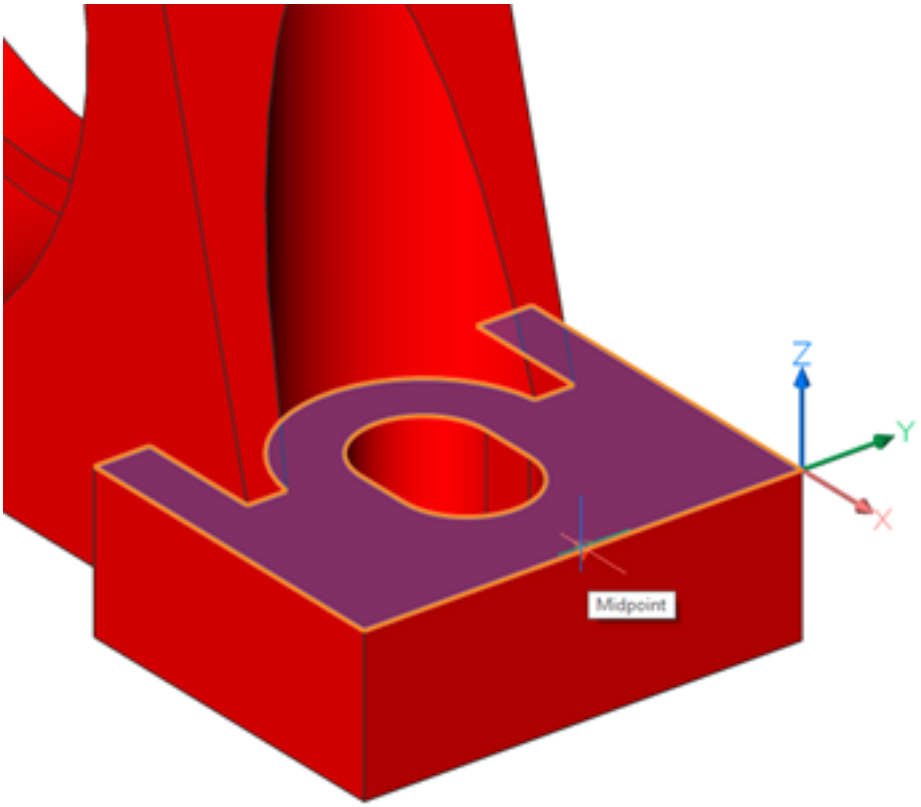


Figure 3.34

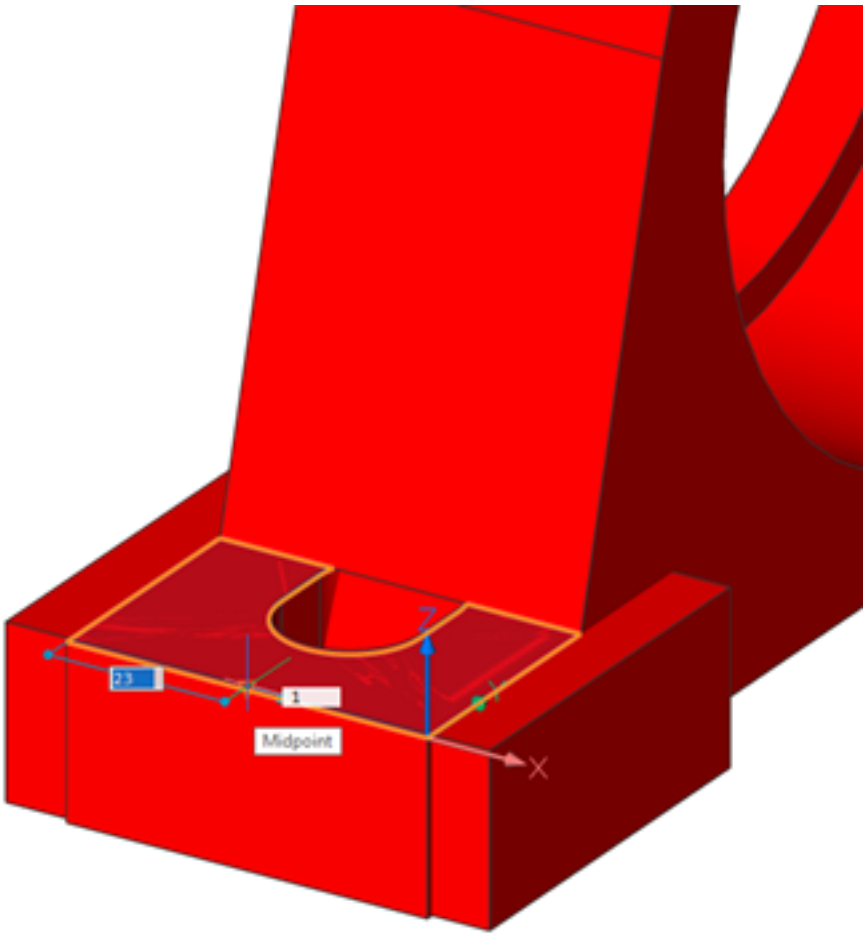
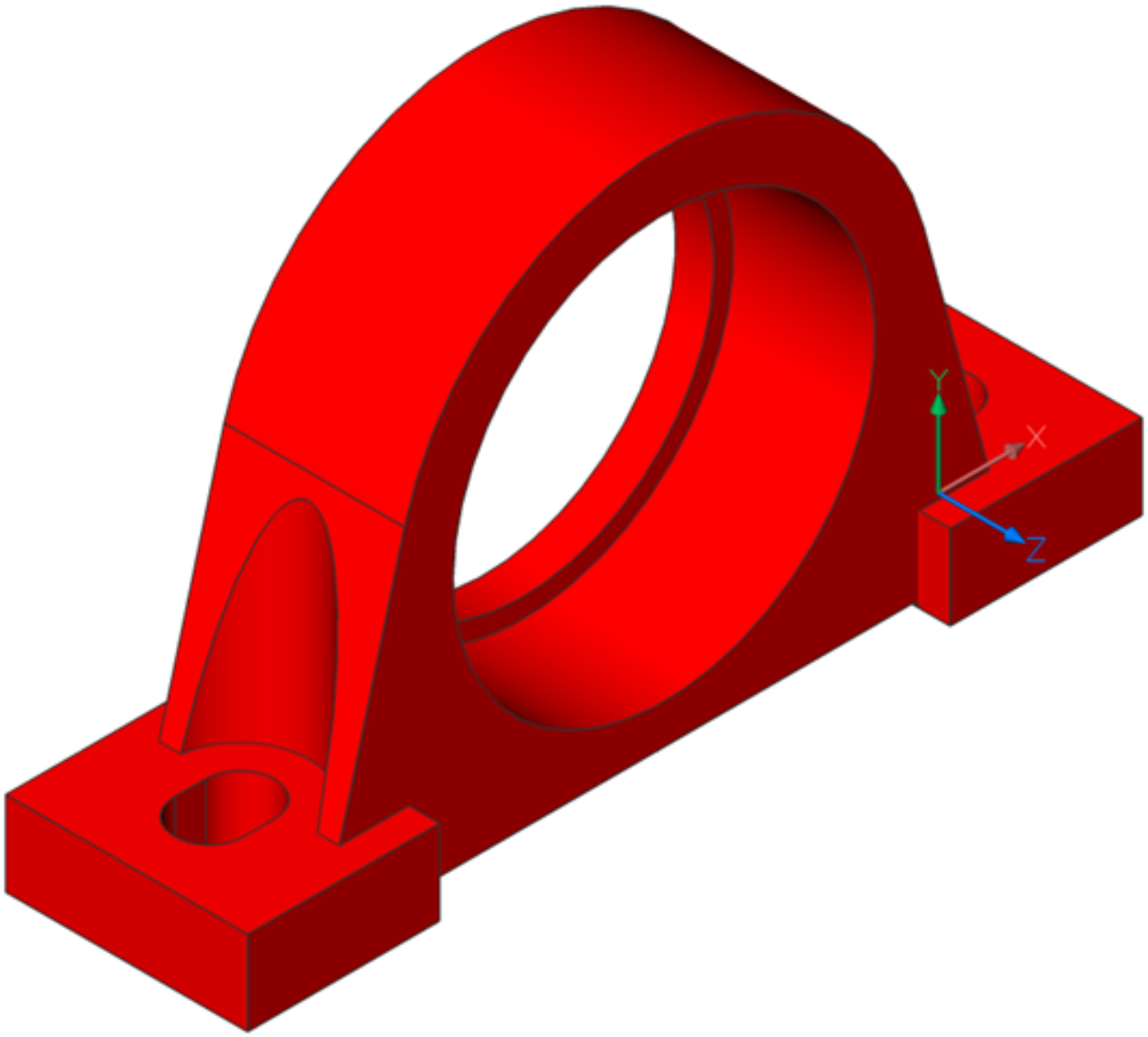


Figure 3.35



3.1.6 Step 4. Edge smoothing

1. Select all vertical edges and the two edge networks. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleatta
2. Select**Fillet (6.12.14)** ( ) from the **Model** section of theQuad (2.2.3). Specify a fillet radius of **3**.
3. Select the larger circular edge and select **Chamfer (6.12.4)**( ) from the **Model** section of the Quad (2.2.3). Specify the edge distance as **2.5**.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039118173/mceclip12.png})

Figure 3.36

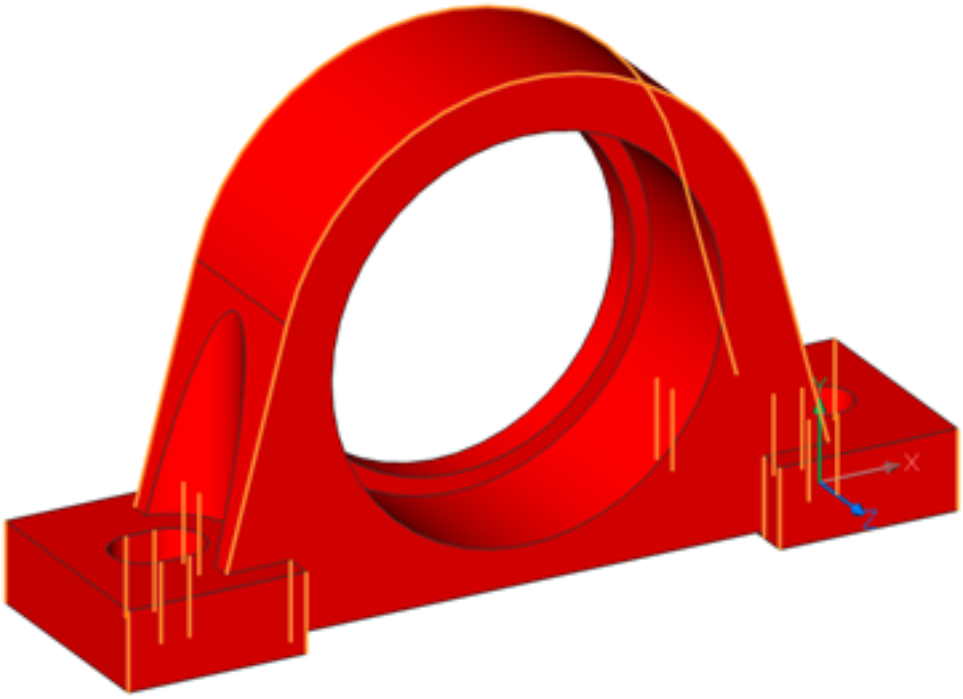
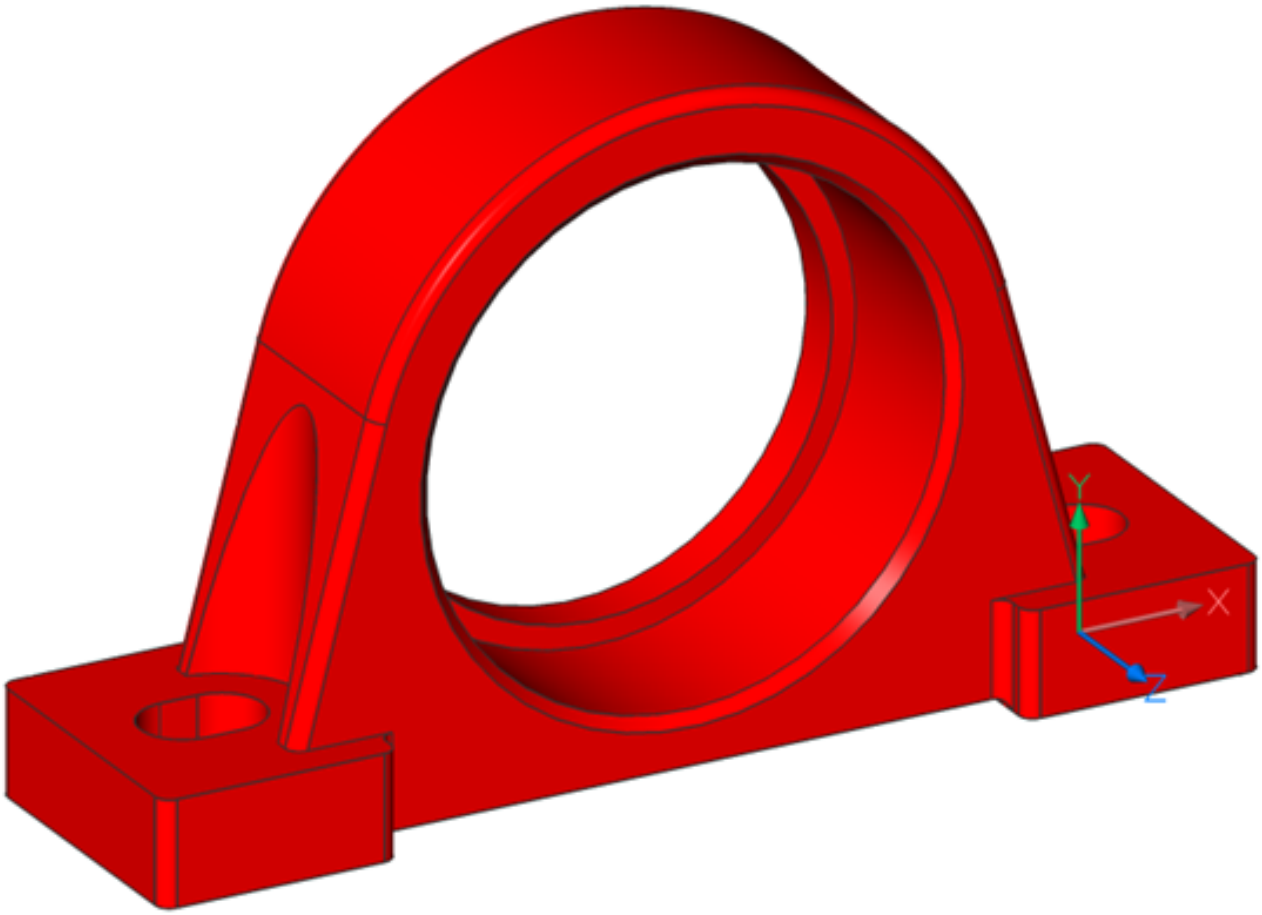


Figure 3.37



3.1.7 Step 5. Update in assembly

1. Save the file with the new name of Bearing_support_110.dwgand switch to the **Shaft_support.dwg** file.
2. In the Mechanical Browser (3.2.2)**,** replace **Bearing_support** with the new **Bearing_support_110.dwg**file.
3. In the Mechanical Browser (3.2.2), replace **BBRG-22x50 MM** with the ball_bearing.dwgfile.
4. Change the **OutsideDiameter** parameter of the **Ball_bearing** component to **110mm.**

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039118253/mceclip13.png})

Figure 3.38

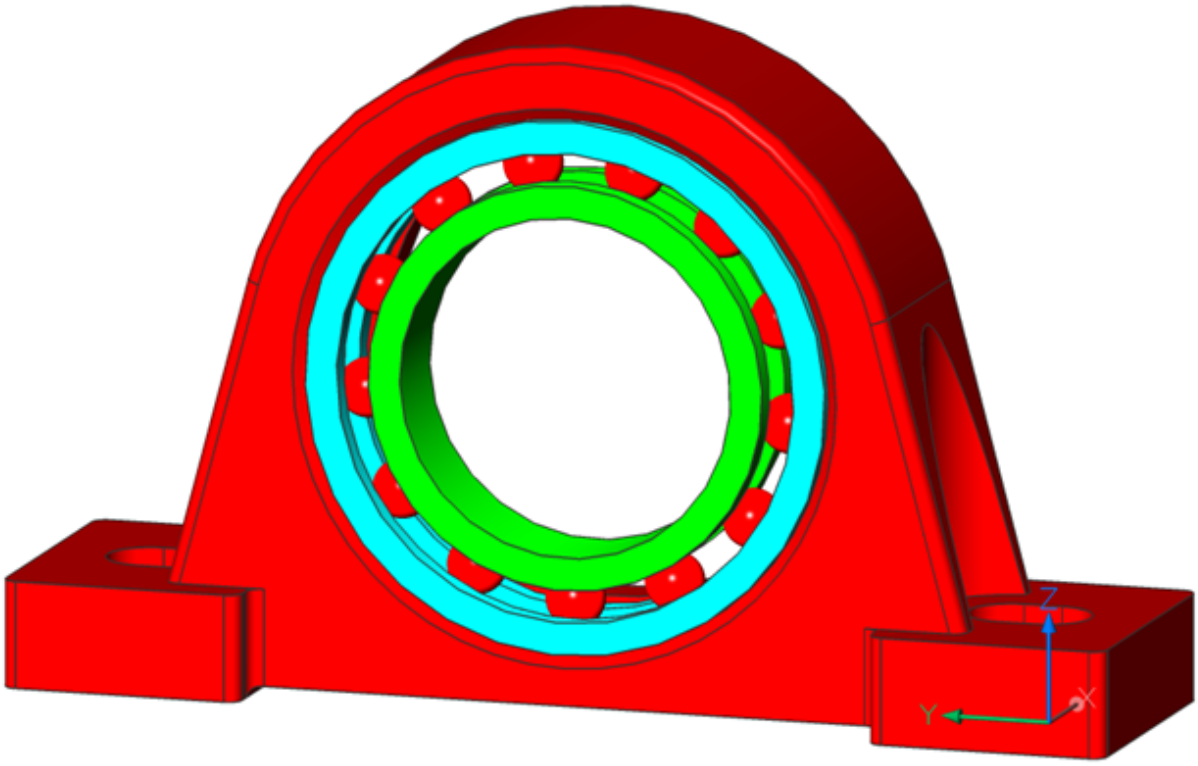
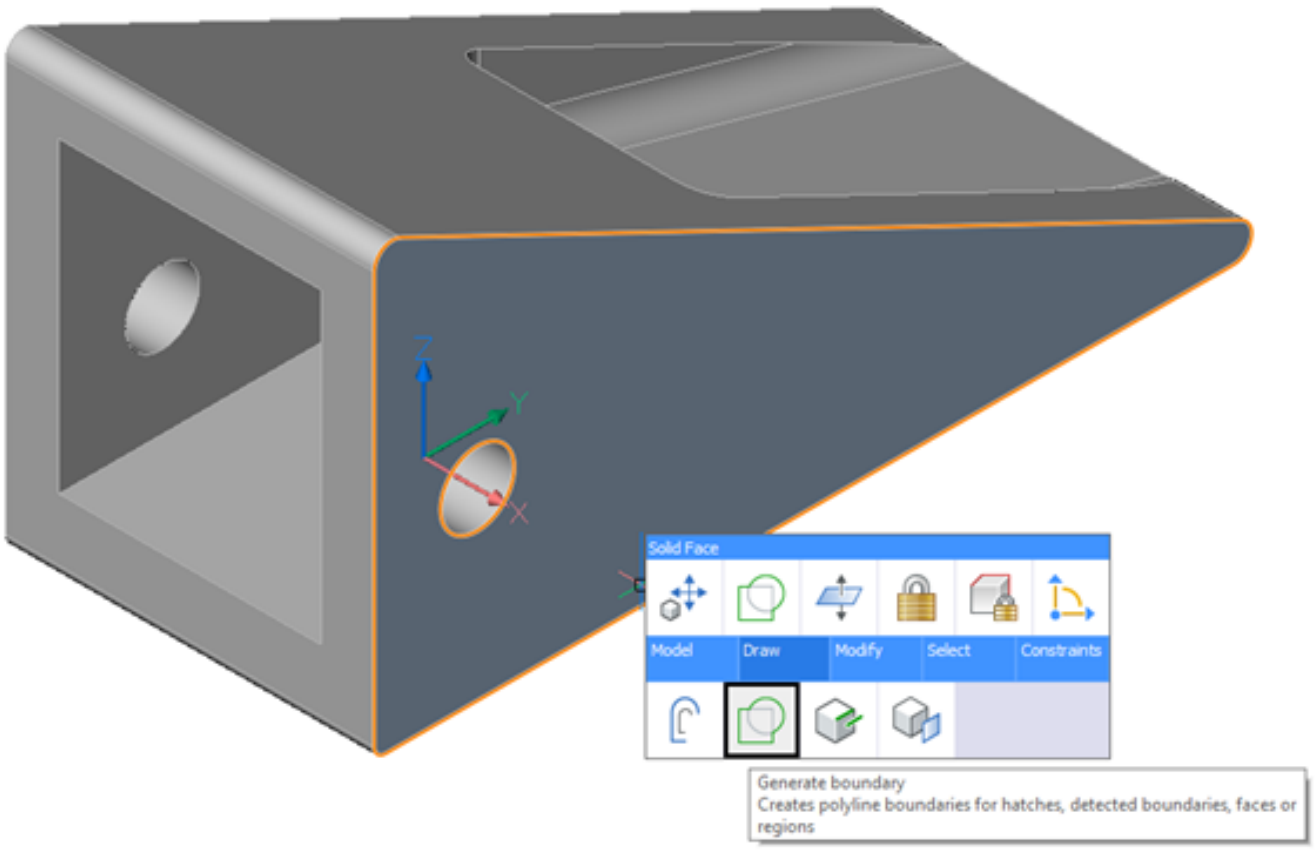


Figure 3.39



3.1.8 Bucket tooth editing

In this tutorial we will edit the teeth of the excavator bucket.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360037398173/mceclip0.png}) Before	(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360037868 After

1. You need a **BricsCAD Platinum**license for this tutorial.
2. Downloadand open the **Bucket_Assembly.dwg**file.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattach
3. Select **Crown** in the Mechanical Browser (3.2.2).
4. Click Open a copy (6.8.18) ( ) in the right click menu.
5. Save as **Crown_new.dwg.**
6. Hover over the side face and select Generate boundary (6.15.14) ( ) from the **Draw** section of the Quad (2.2.3)**.** (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360037398393/mceclip0.png})
7. Hover over the solid and select Move (6.12.17) ( ) from the **Model** section of the Quad (2.2.3)and move the solid along the X-axis by a distance of **200**.
8. Hover over the solid and select Hide entities (6.16.11) ( ) from the **Select** section of the Quad (2.2.3)**.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360037399833/mceclip8.png})**
9. Delete the circle.
10. Select the polyline and select Extrude (6.12.13) ( ) from the **Model** section of the Quad (2.2.3)**.** Enter an extrusion distance of **120**.
11. Hover over the filet and select Select same or less radius fillets (6.12.26) ( ) from the **Select** section of the Quad (2.2.3)**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360036565034/mceclip12**
12. Delete all fillets.
13. Select the blade edge. Select Move (6.12.17) ( ) from the **Model** section of the Quad (2.2.3)**.**

Figure 3.40

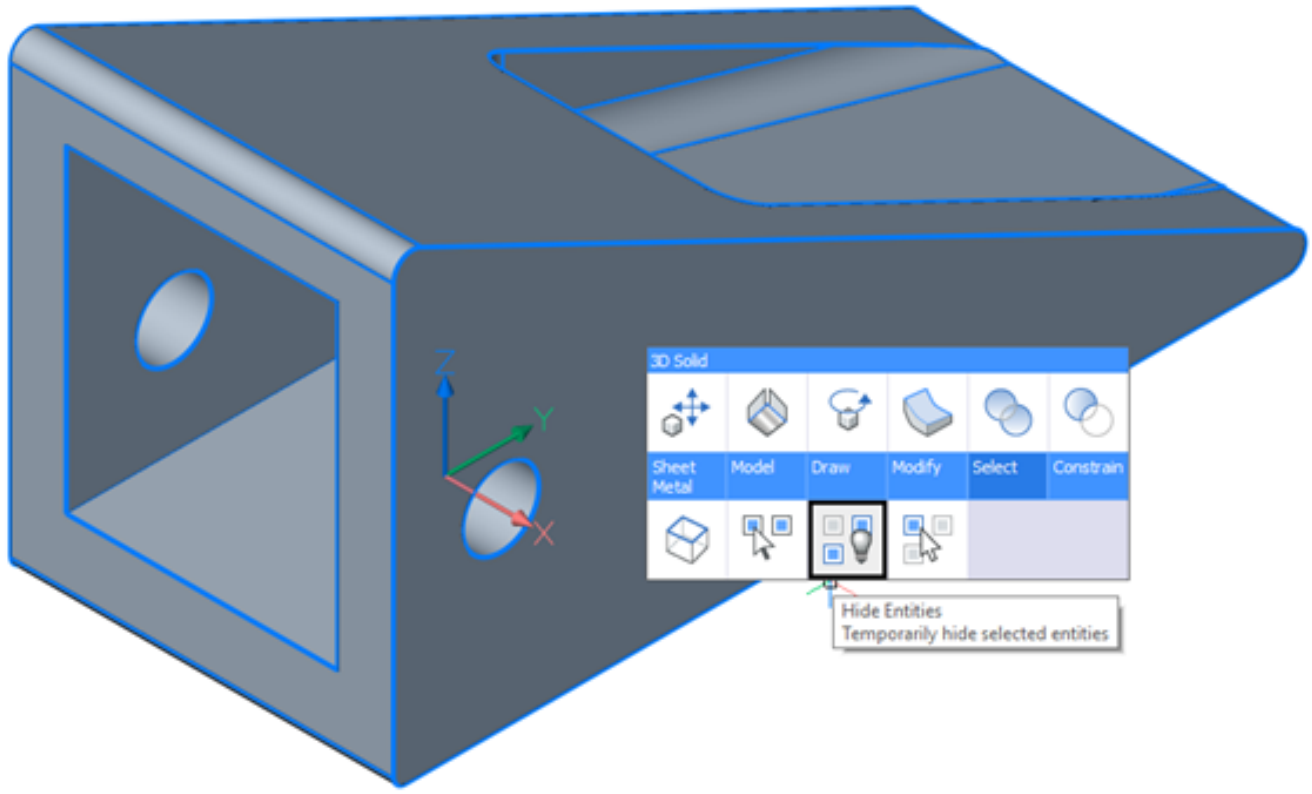


Figure 3.41

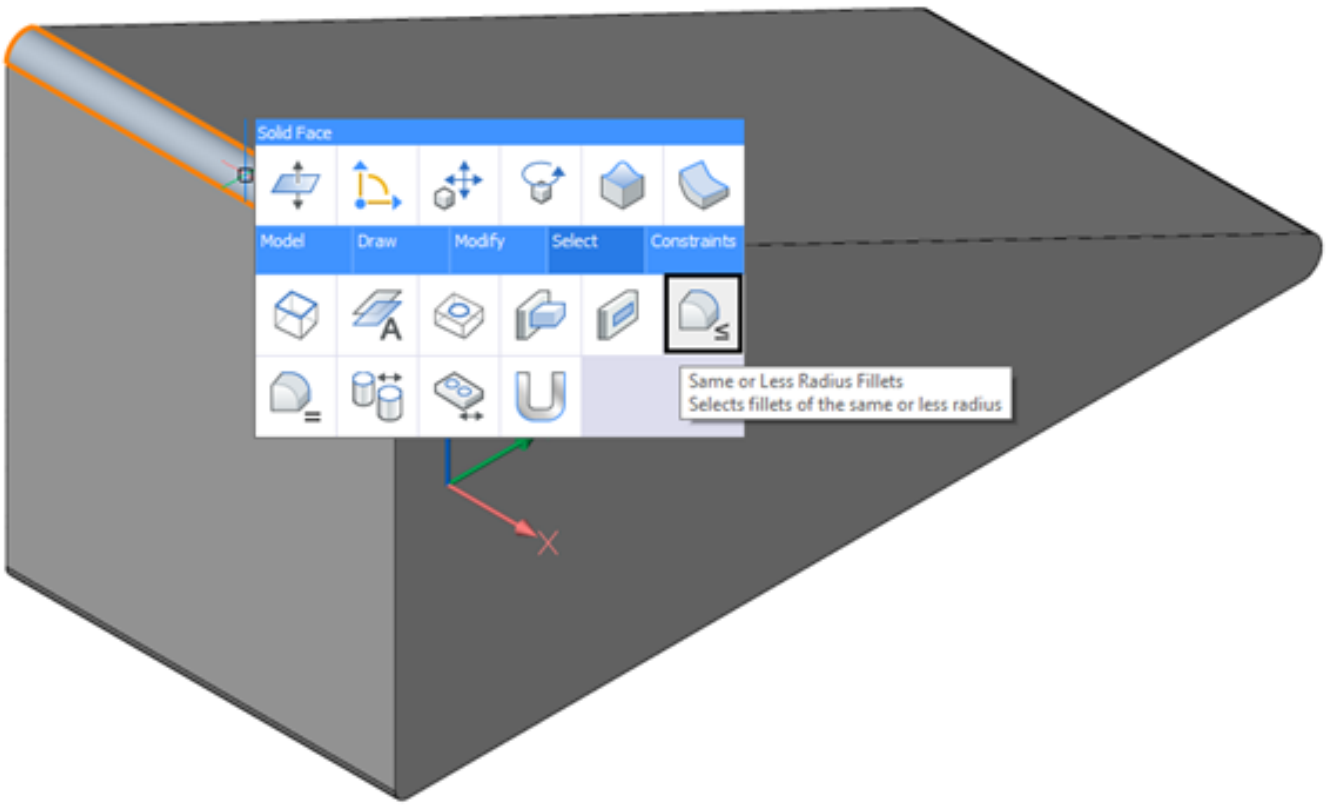
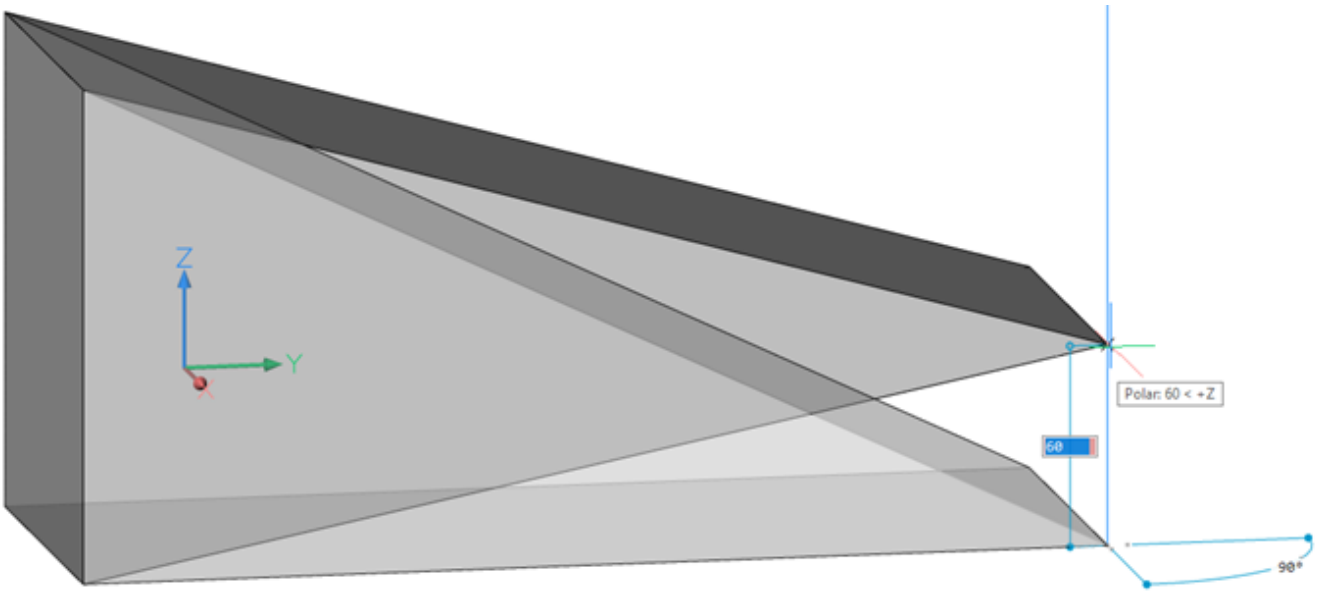


Figure 3.42



14. Click to select a base point, move the edge along the Z-axis. Enter a distance of **60**.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360036564254/mceclip0.png})
15. Hover over the back face and hold down the left mouse button longer than ¼ second to launch the manipulator widget. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360036564294/mceclip1.png})
16. Place the cursor over the Y-axis. Hold down the CTRLkey and click.
17. Move the cursor to copy the entity along the Y-axis. Set a distance of **100**.(\vref {fig:C:/Dev/Task-2448/images/helpbricsy
18. Repeat step 16 to create another surface with a distance of **60.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc**
19. Click the Imprint (6.17.12) ( ) button on **Solid** | **Edit** panel of the Ribbon (1.13.6).
20. Select the solid and the first surface. Enter **Yes** to delete the source surface.
21. Select the second surface. Enter **Yes** to delete the source surface and finish the command.
22. Enable the Edges between coincident faces (1.18.3) ( ) tool button on the **Solid** | **Design Intent** panel of the Ribbon (1.13.6).
23. Select the 2 triangular blade faces and select PushPull (6.12.20) ( ) from the **Model** section of the Quad (2.2.3)**.** Move the cursor out a distance of **30**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachmer
24. Hover over the blade edge. Select Add Fixation (6.12.15) ( ) from the **Constraint** section of the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360037405293/mceclip18.png})
25. Hover over the second top face. Select Rotate (6.12.24) ( ) from the **Model** section of the Quad (2.2.3)**.**
26. Select the edge between the second and third top faces as the rotate axis. Move the cursor down and enter a rotation angle of **5**.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360036569294/mceclip

Figure 3.43

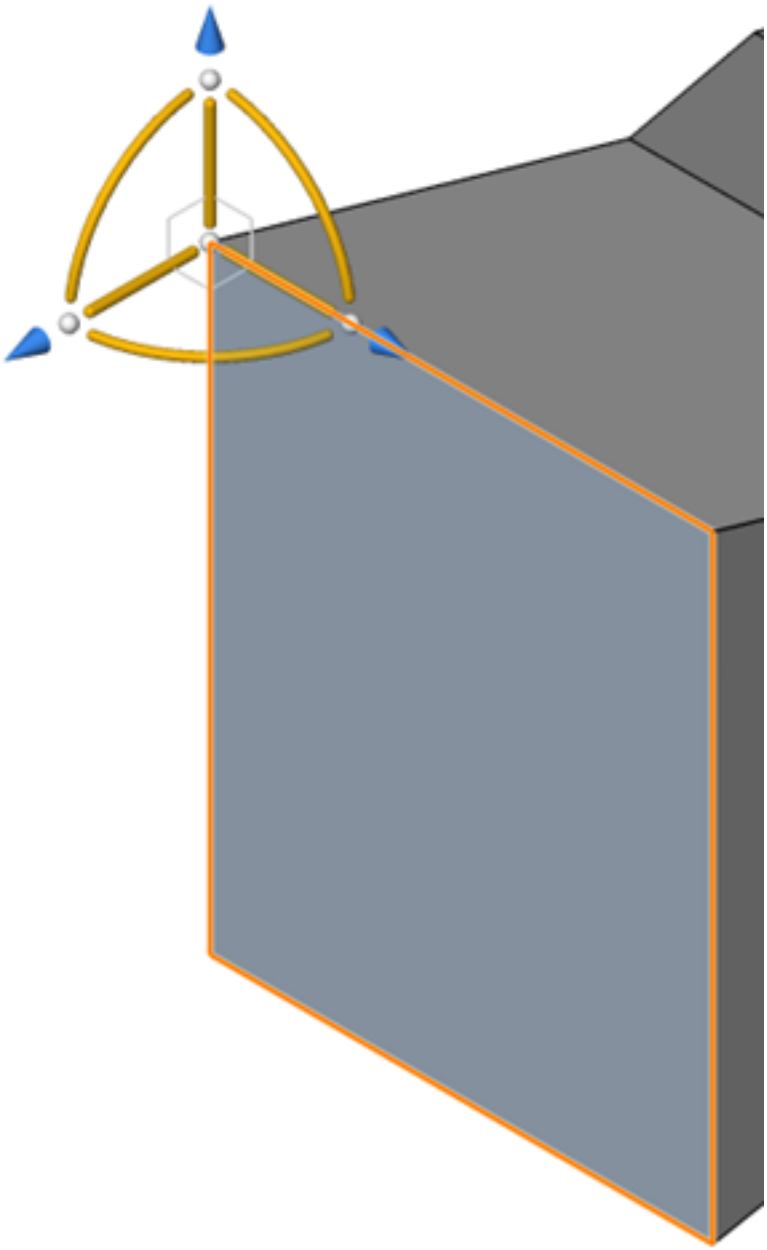


Figure 3.44

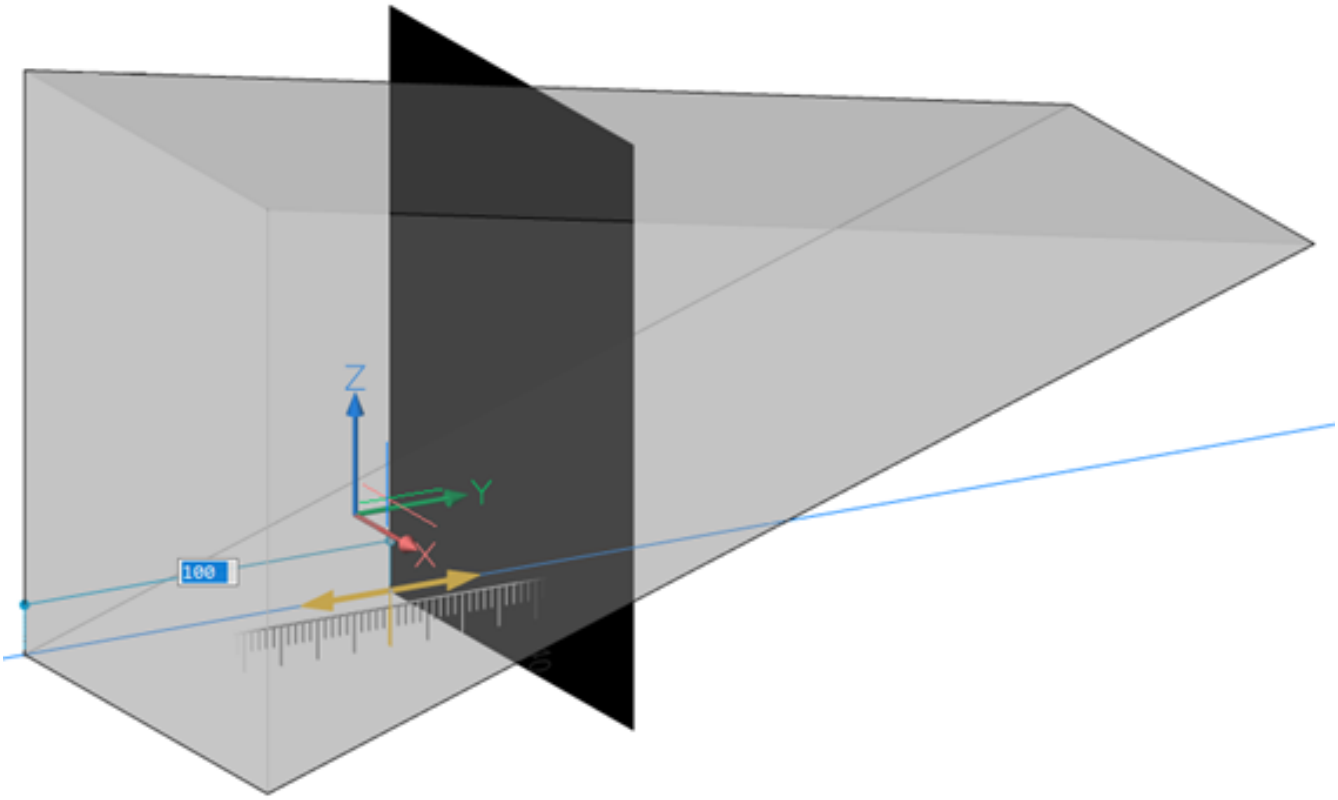


Figure 3.45

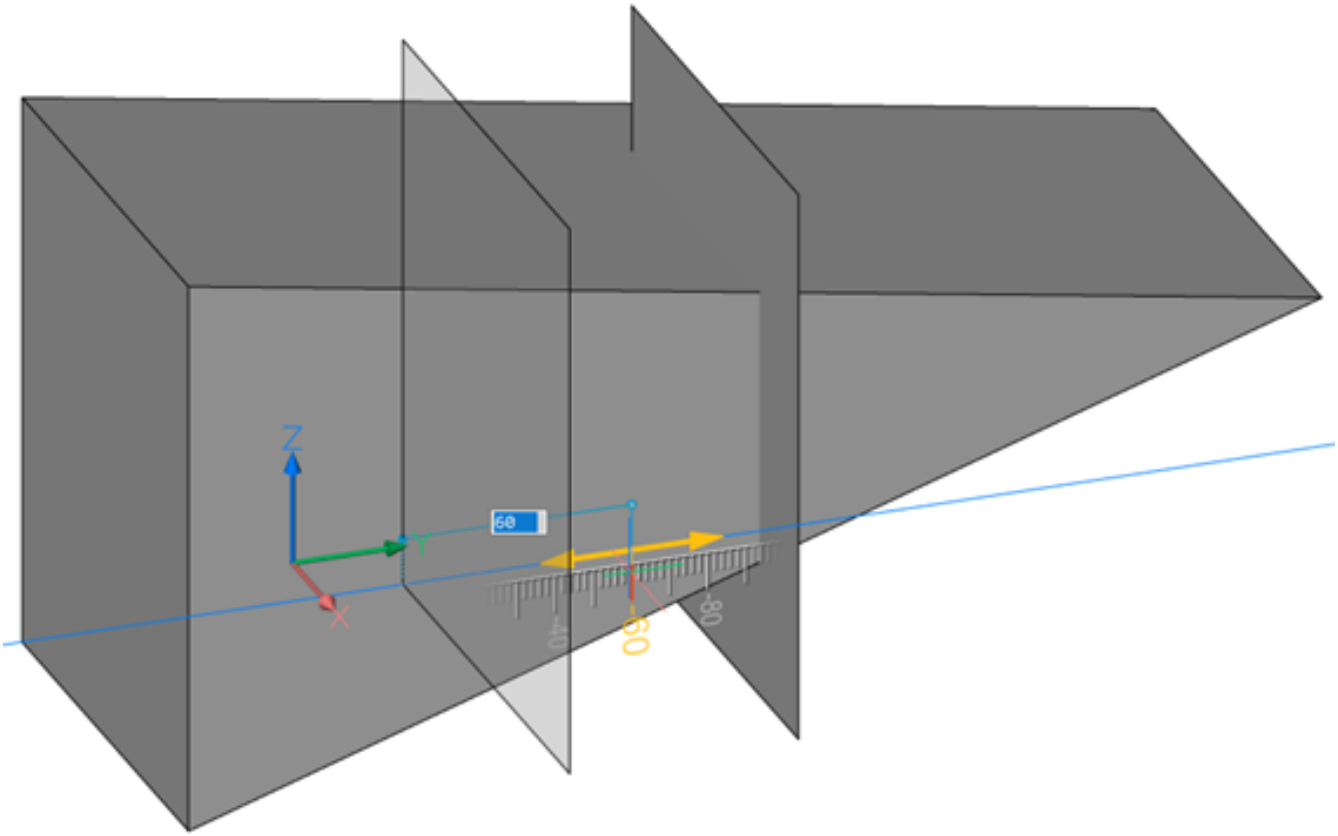


Figure 3.46

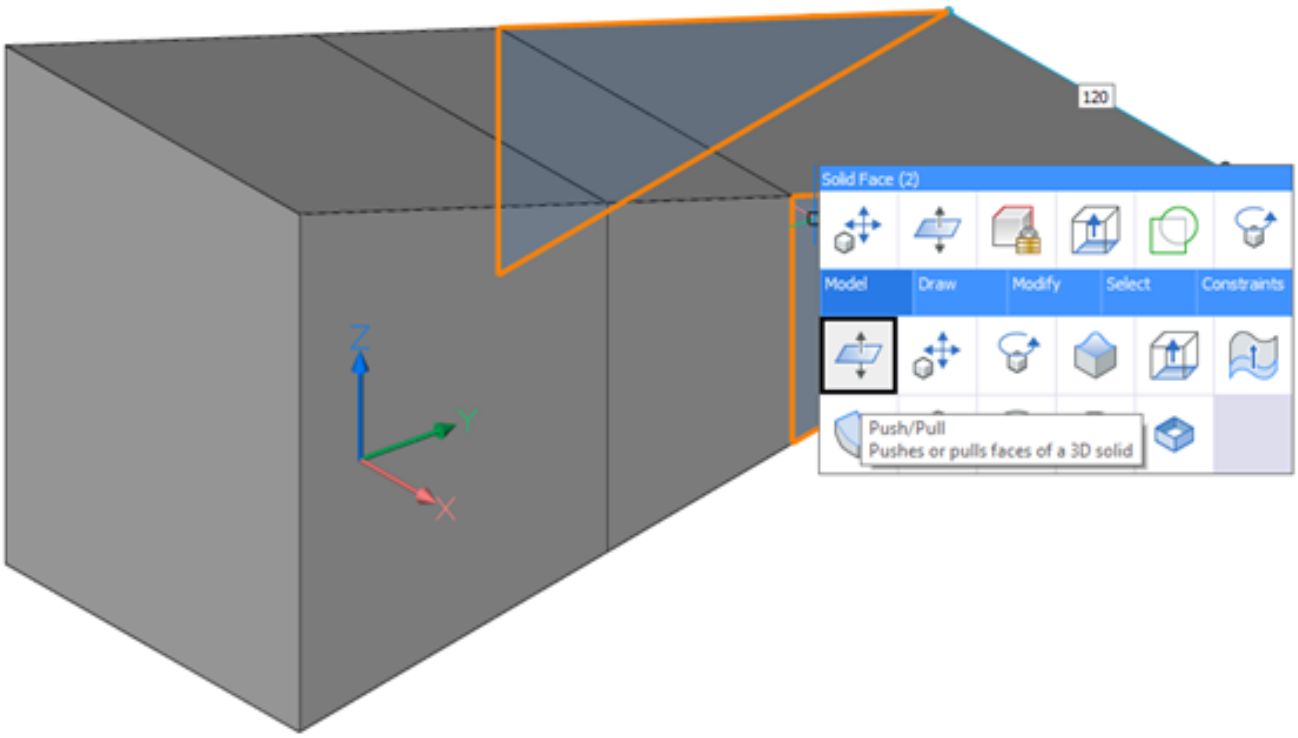
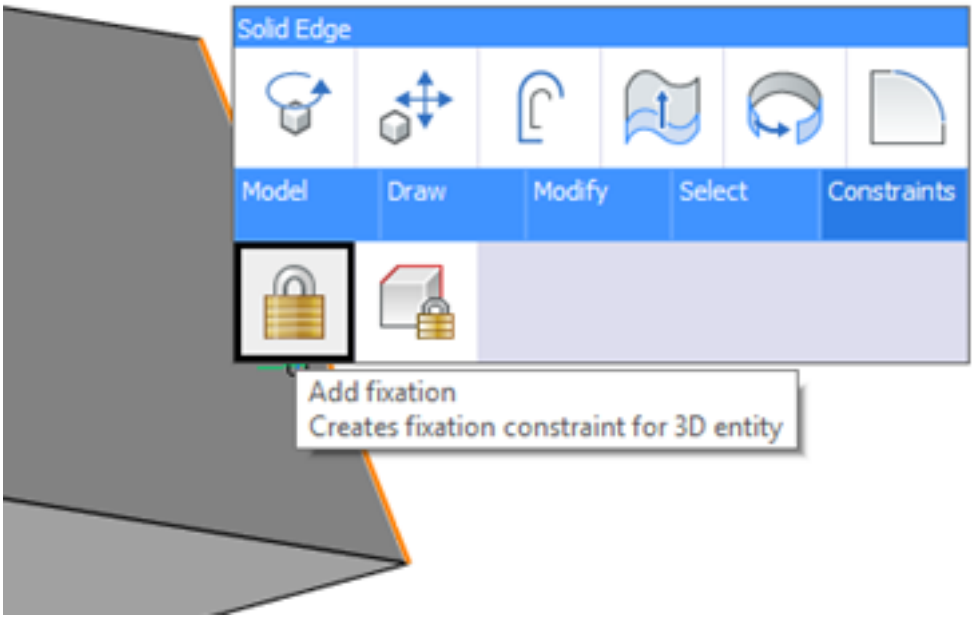


Figure 3.47



27. Rotate the second bottom face symmetrically to up **5** degrees.
28. Select all horizontal edges. Select Fillet (6.12.14) ( ) from the **Model** section of the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360037405273/mceclip16.png})
29. Enter a radius of **6**.
30. Enable Snap to Midpoint (6.21.16) ( ) in the **Home | Entity Snap** panel of the Ribbon (1.13.6).
31. Click theRectangle (6.26.6) ( ) tool button in the **Solid | Draw** panel of the Ribbon (1.13.6).
32. Create a rectangle of dimensions **20x200**. Snap to theMidpointof the blade edge of the top face.

NOTE	To lock Dynamic UCS (1.15.18) to the face, hover over the blade top face and press the SHIFT key during the drawing command.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360037405373/mceclip21.png})

33. Hover over the rectangle. Select Extrude (6.12.13) ( ) from the **Model** section of the Quad (2.2.3).
34. Enter an extrusion height of **20**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36003657
35. Hover over the vertical left face of the new extrusion. Select Extrude (6.12.13) ( ) from the **Model** section of the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360036572174/mceclip
36. Enter an extrusion distance of **20** to make the rib symmetrical.
37. Hover over the top front face of the rib.
38. Select Rotate (6.12.24) ( )from the **Model** section of the Quad (2.2.3).
39. Set the edge between the selected face and the fillet face as the rotation axis.
40. Move the cursor inwards. Enter a rotation angle of **60**.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleatt
41. Repeat steps 37-40 to rotate the top back face of the rib inwards by an angle of **80**.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360037678434/mceclip1.png})
42. Hover over a face of the rib and select Select Protrusion (6.12.26) ( ) tool button on the **Select** section of theQuad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360037678954/mceclip2
43. Click the CopyFaces (6.12.8) ( ) tool button on **Solid | Edit** panel of the Ribbon (1.13.6). Select the main face's middle point as the base point for the copy.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachr
44. Use the **Rotate** option of the command, rotate the rib **180** degrees around a vertical axis.
45. Use the **Mode** option of the command, to change the cutting mode to **No cut** and insert the rib symmetrically with the edge midpoint as the insertion point.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachment
46. Use theRotate (6.12.24) ( )of the **Model** section of theQuad (2.2.3).
47. Rotate side faces by an angle of **15**.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36003

Figure 3.48

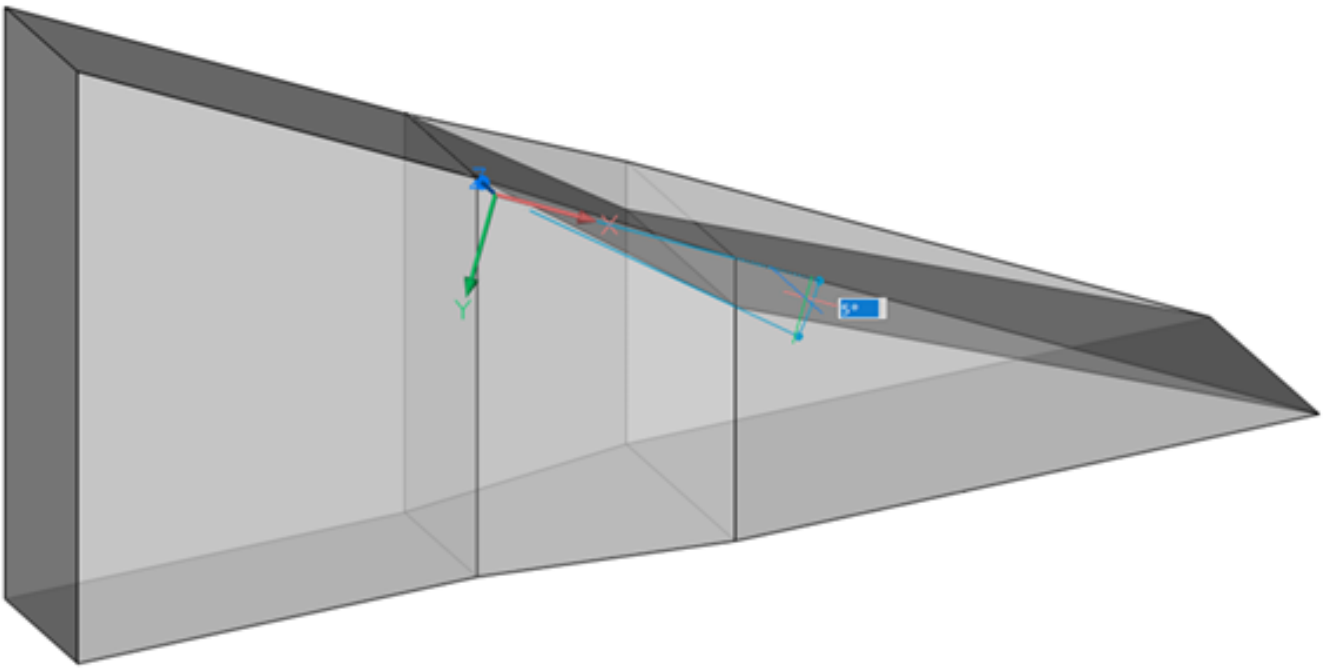


Figure 3.49

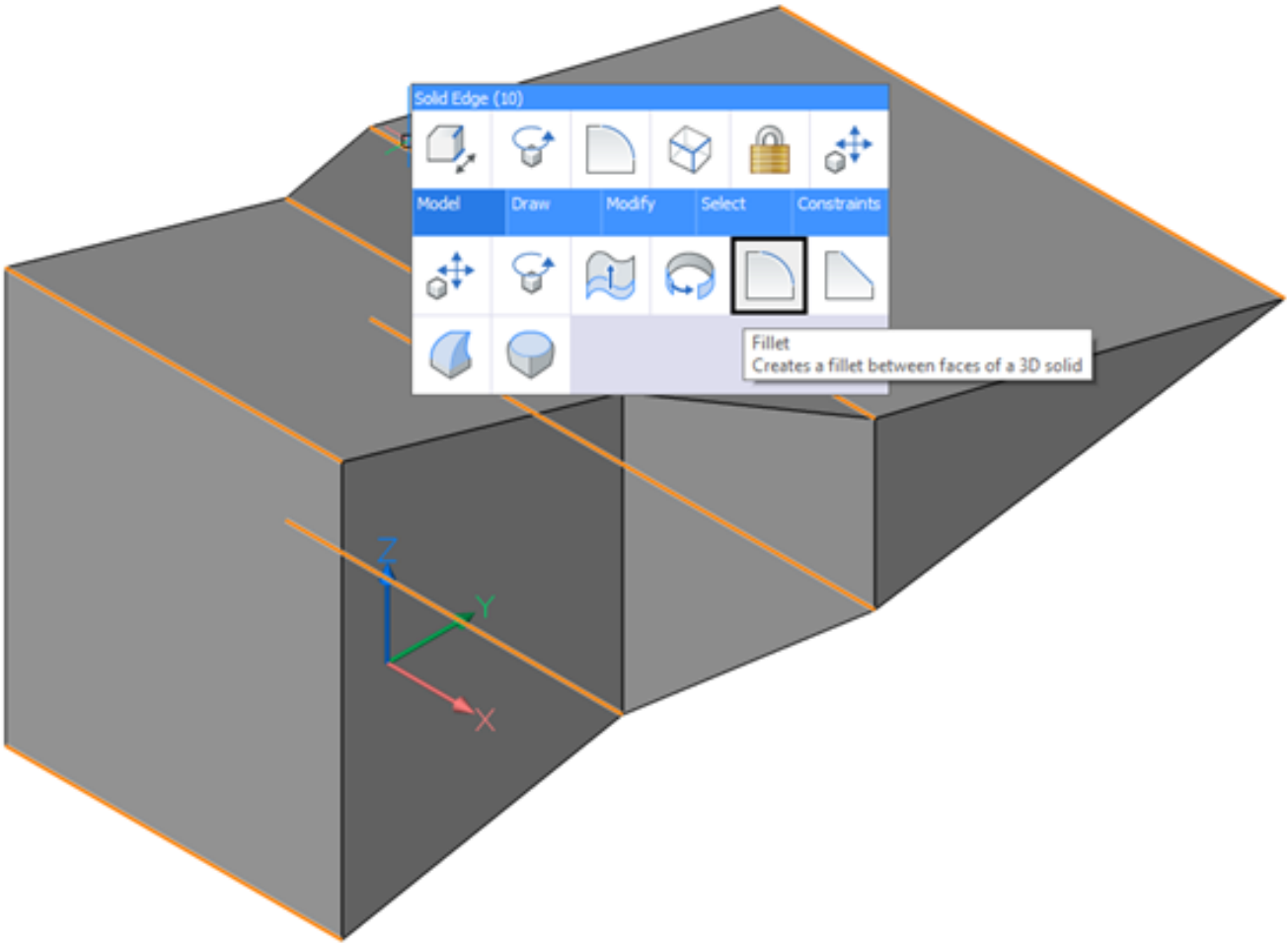


Figure 3.50

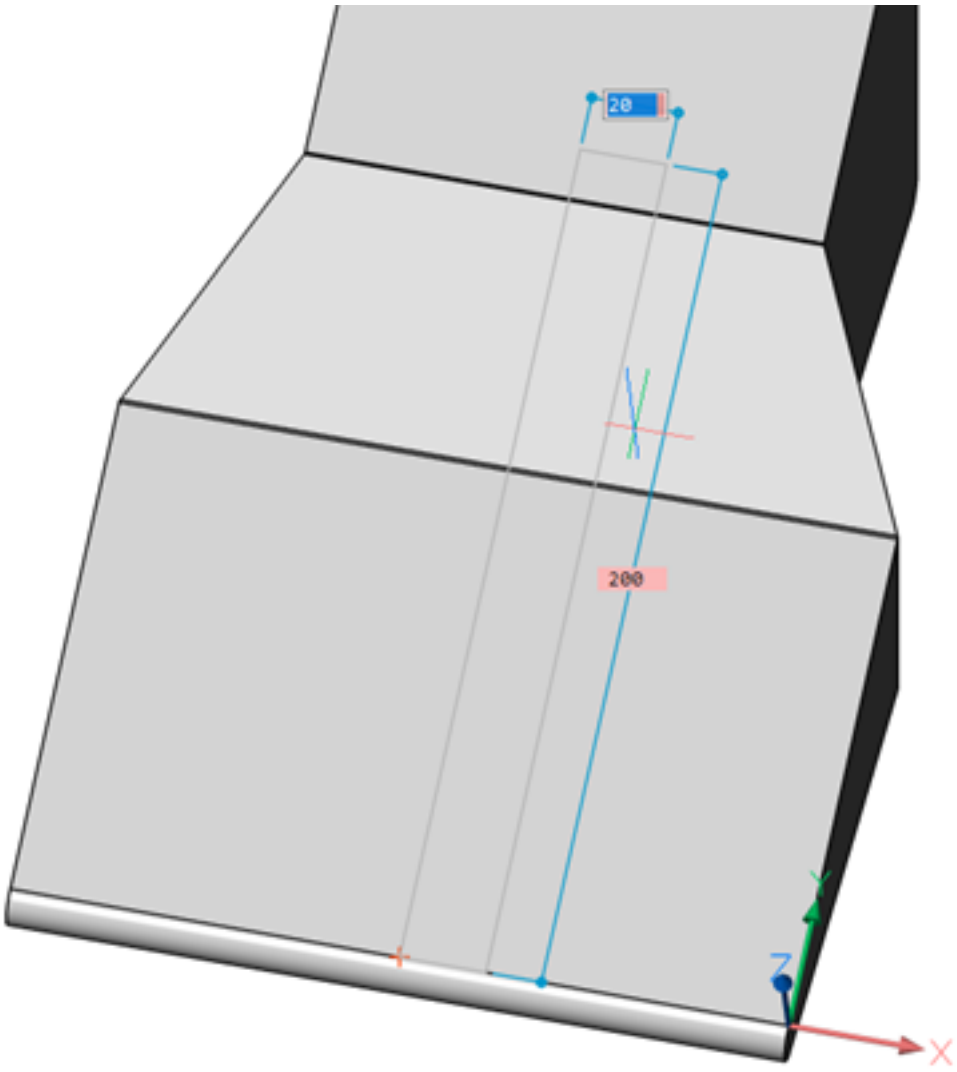


Figure 3.51

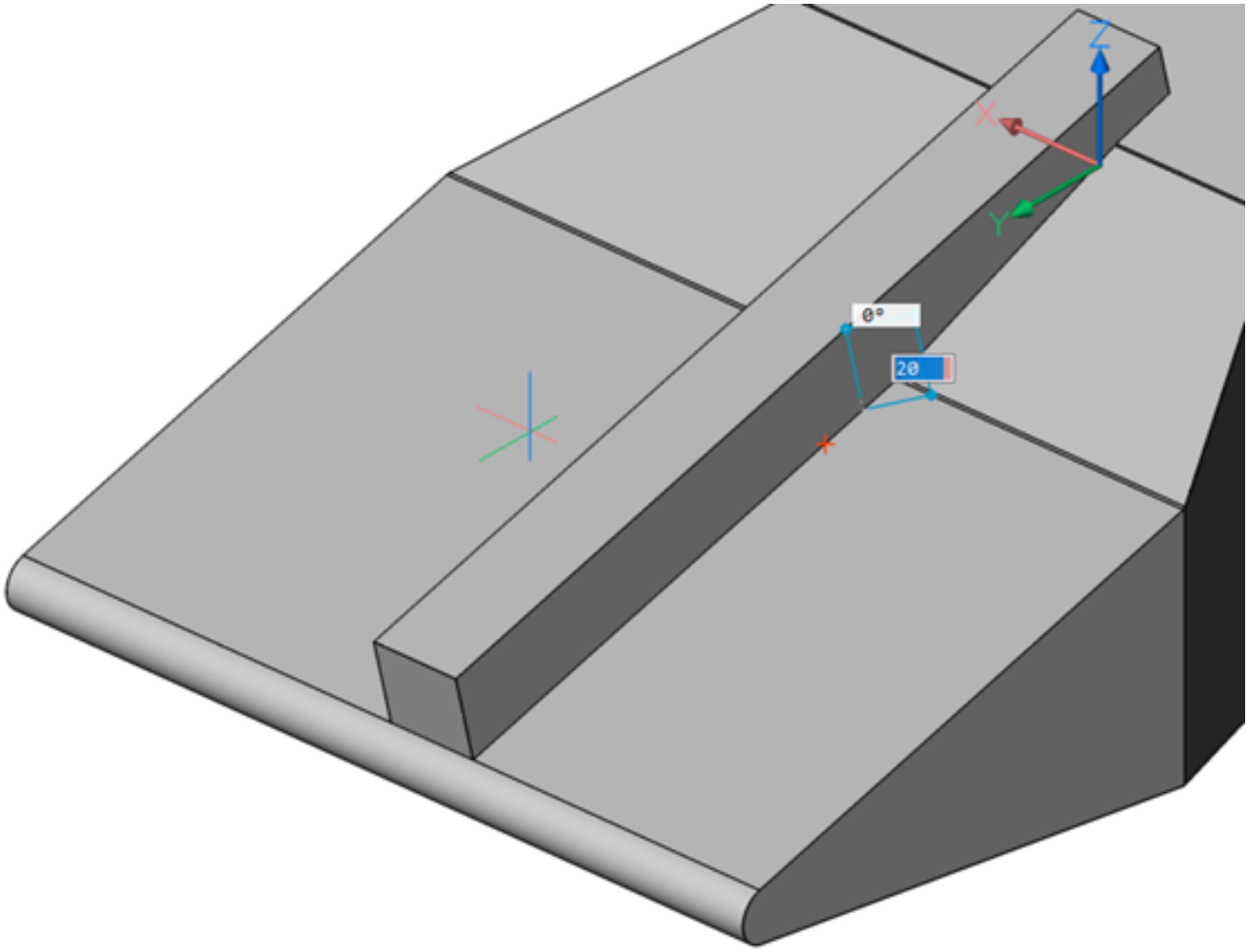


Figure 3.52

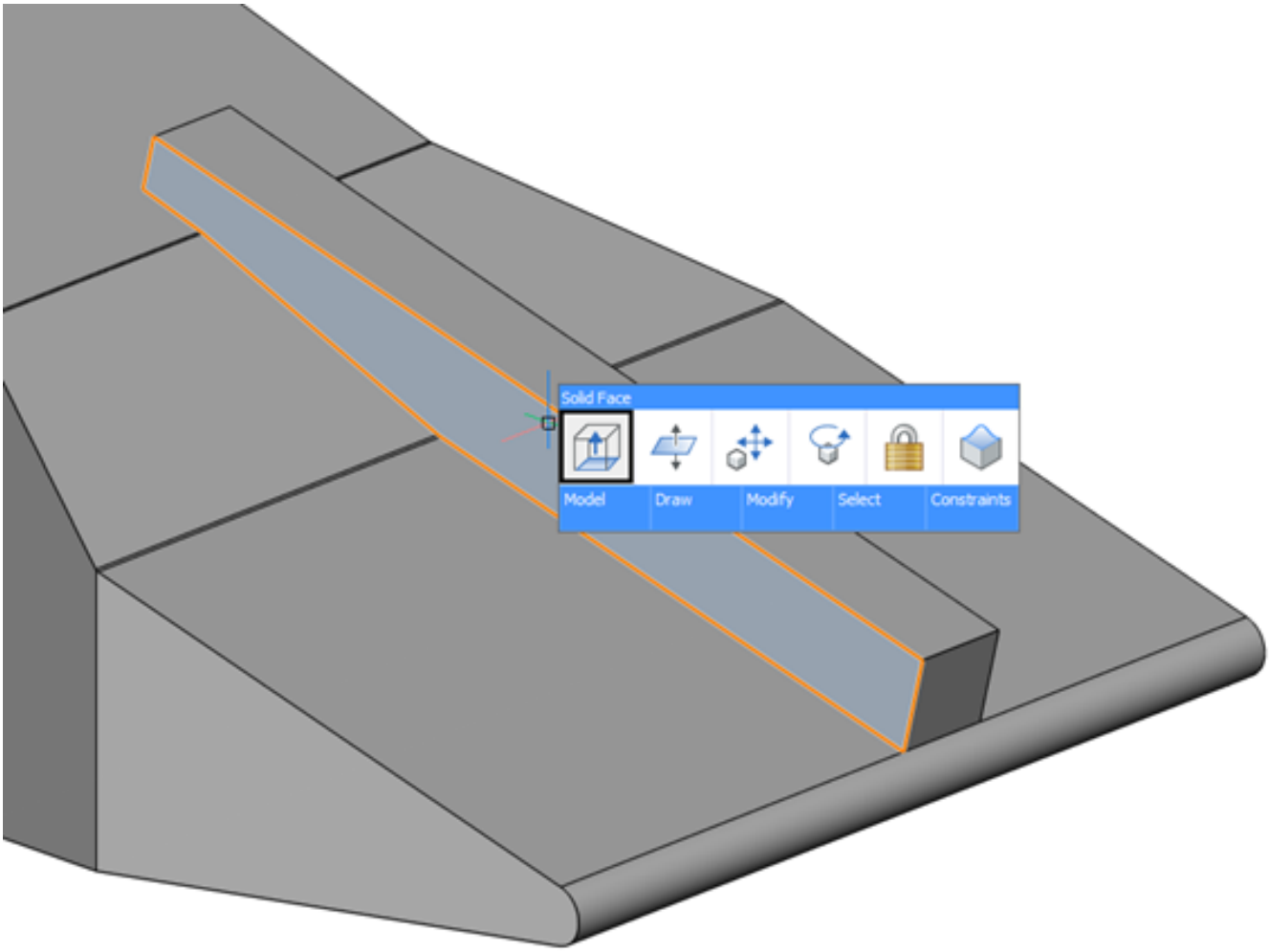


Figure 3.53

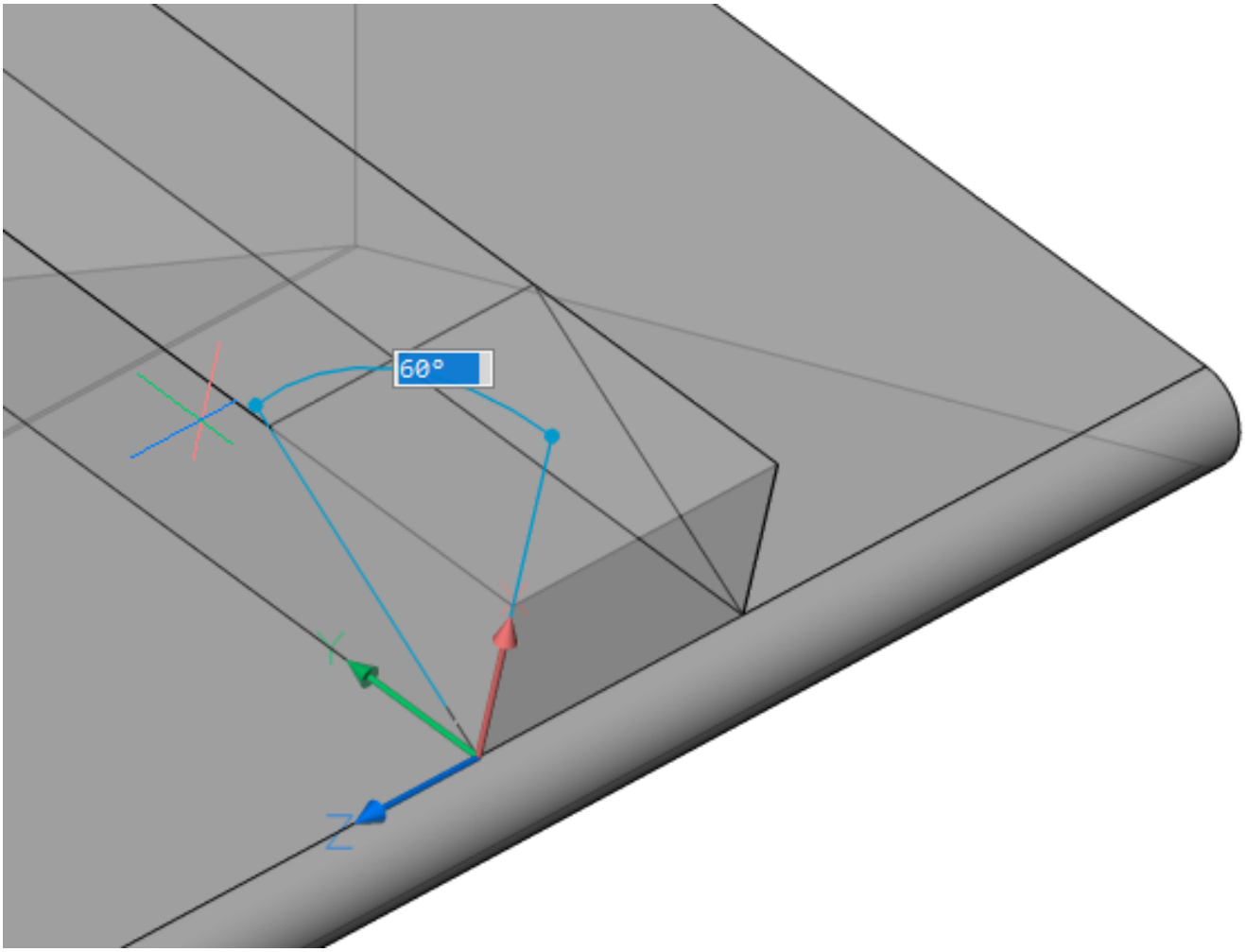


Figure 3.54

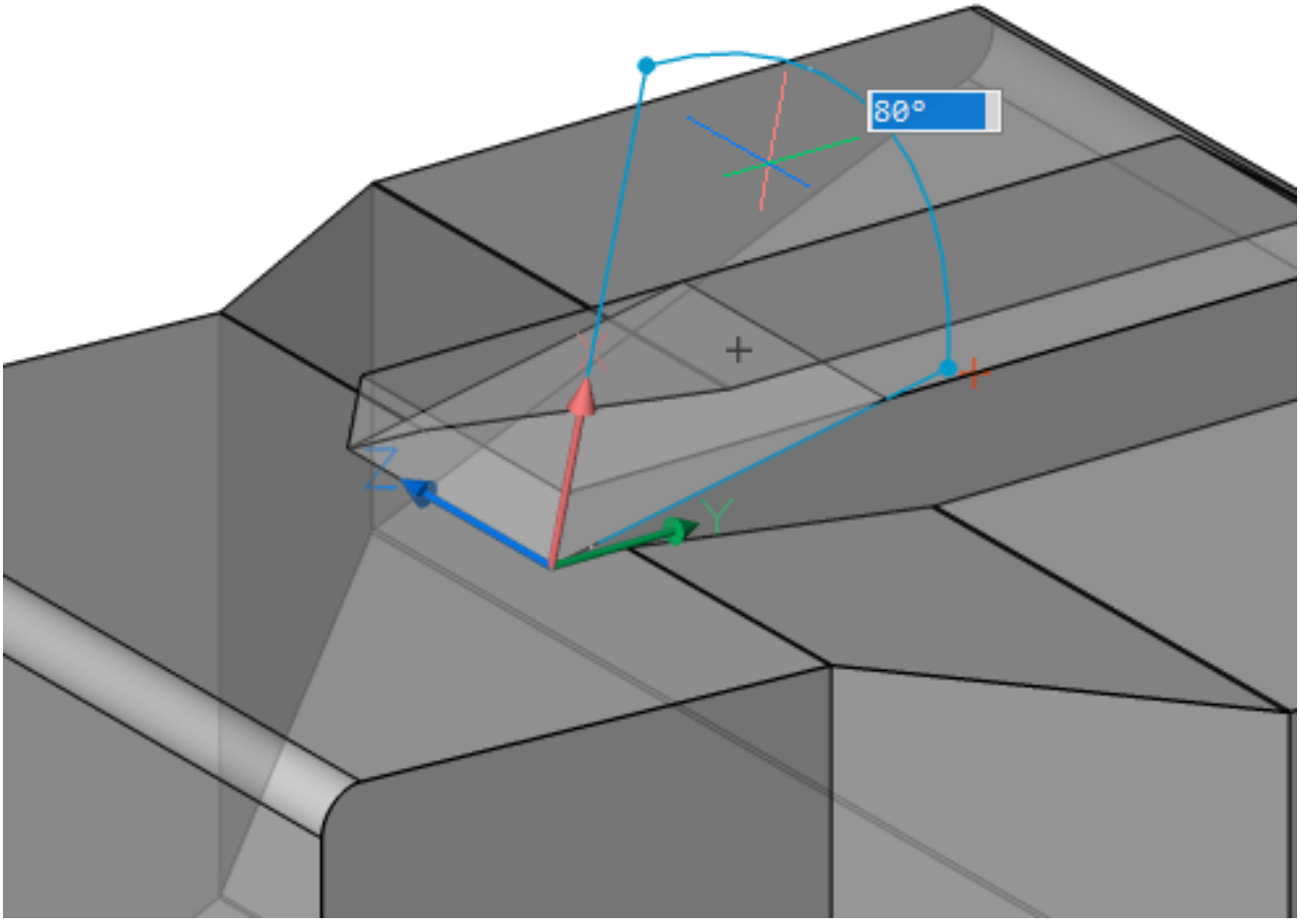


Figure 3.55

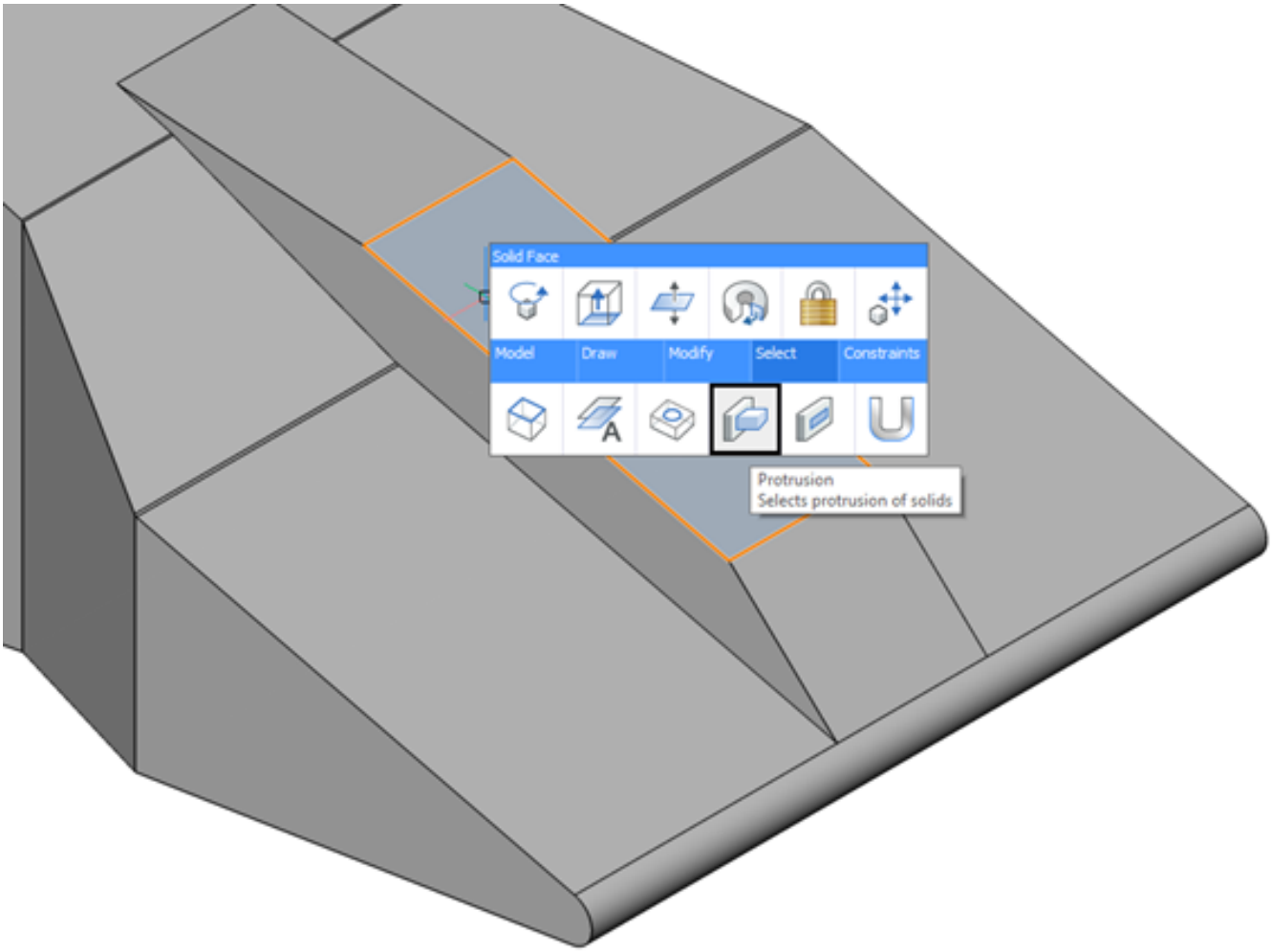

Solid Face
Model
Draw
Modify
Select
Constraints
Protrusion
Selects protrusion of solids

Figure 3.56

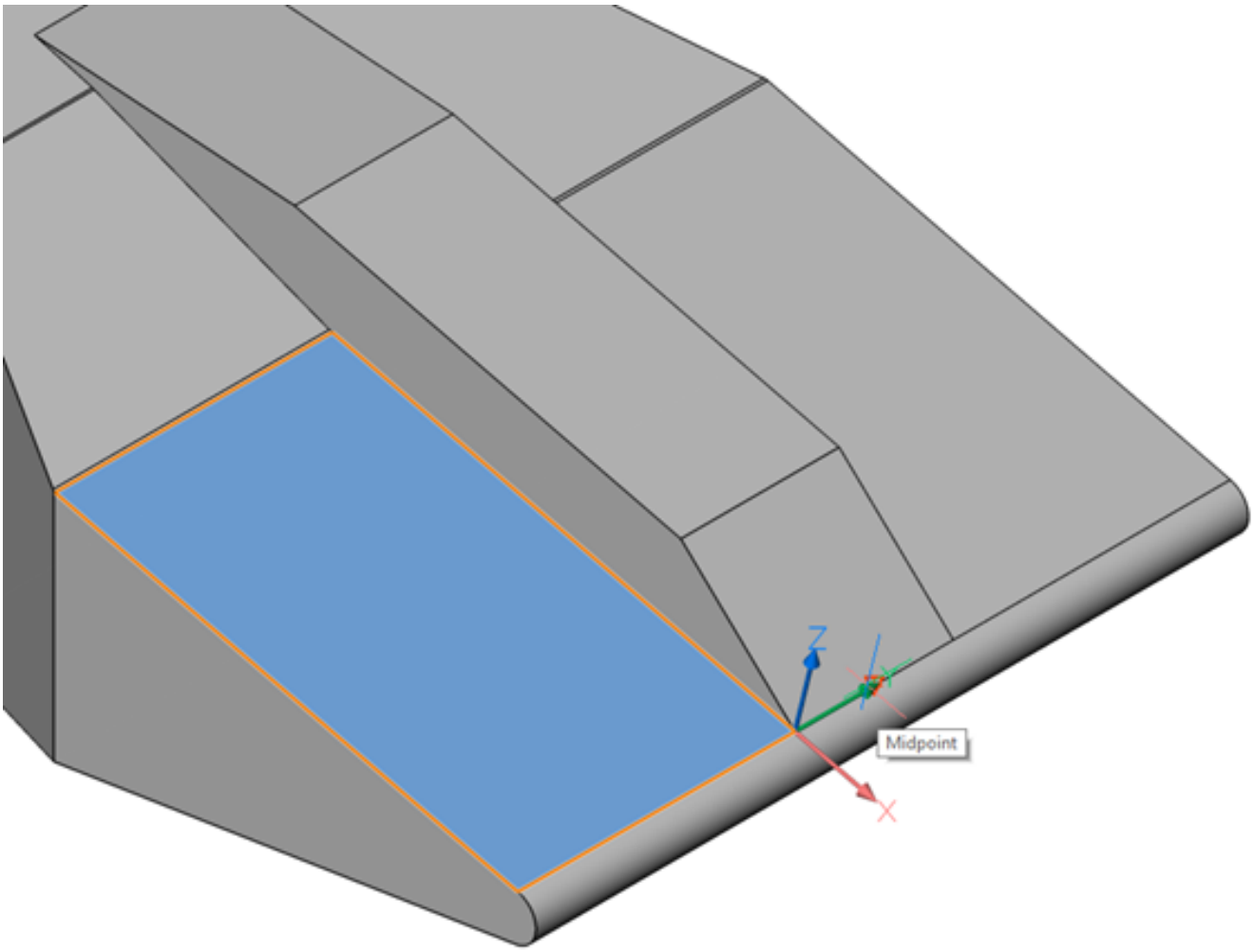
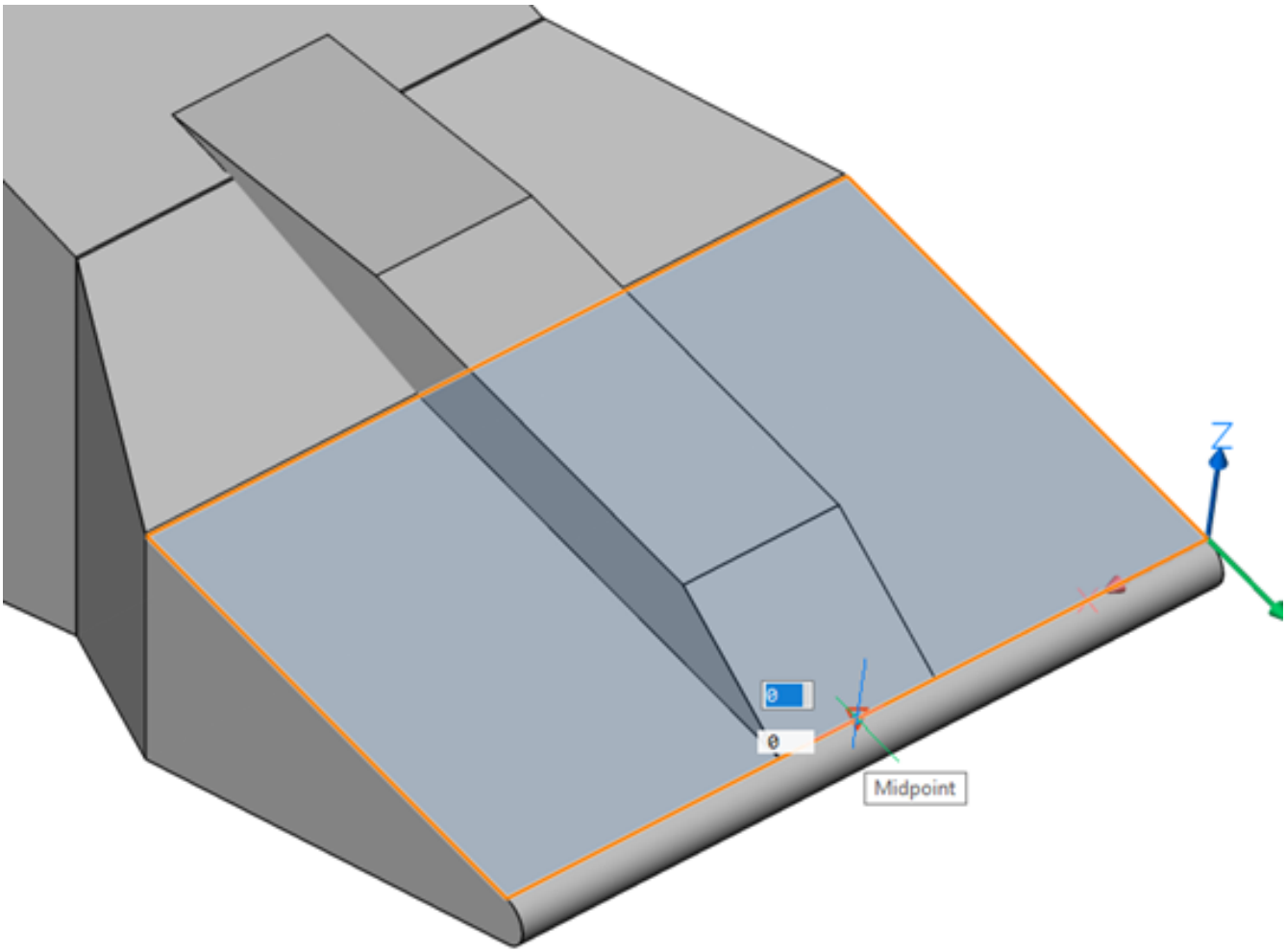


Figure 3.57



48. With nothing selected, right-click to call theQuad (2.2.3). Select Show Entities (6.30.7) ( ) from the **General** section of the Quad (2.2.3).
49. Using the Select Depression (6.12.26) ( ) tool from **Solid | Select** panel of the Ribbon (1.13.6), select the depression and two holes as show below.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360
50. Click theCopyFaces (6.12.8) ( )tool button on **Solid | Edit** panel of the Ribbon (1.13.6). Select the midpoint of the left vertical edge of the back face of the old crown. Set the insertion point to the midpoint of the left vertical edge of the back face of the new crown.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/
51. Select Erase (6.13.8) ( ) from the **Modify** section of the Quad (2.2.3). Delete the old crown.
52. Save file**Crown_new.dwg**.
53. Switch to the **Bucket_Assembly.dwg** file.
54. Select **Crown** in Mechanical Browser (3.2.2).
55. Click Replace all inserts (6.8.21) ( ) in the right click menuand select the **Crown_new.dwg** file to use as a replacement.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360037868834/mceclip2.png})

Figure 3.58

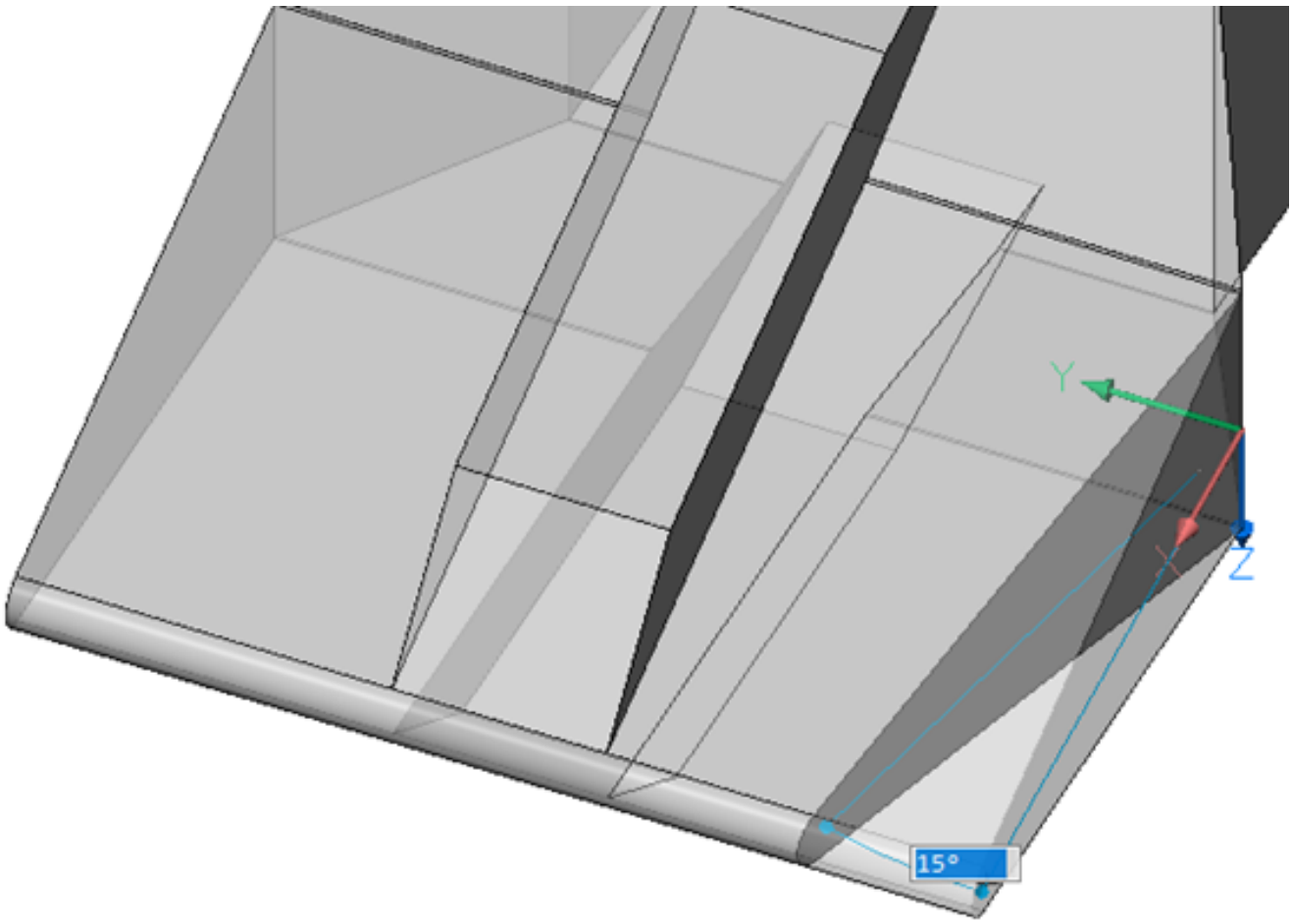


Figure 3.59

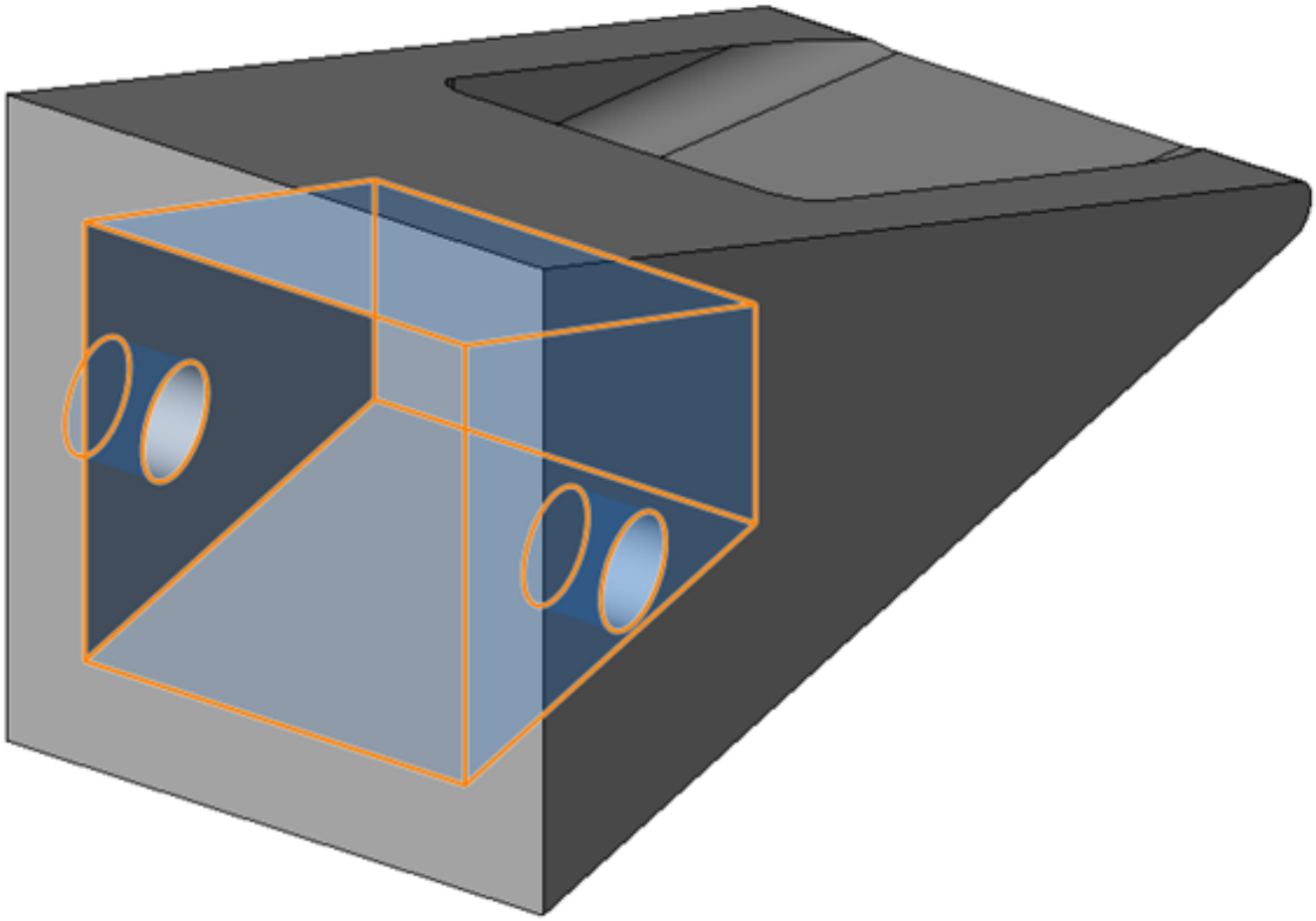


Figure 3.60

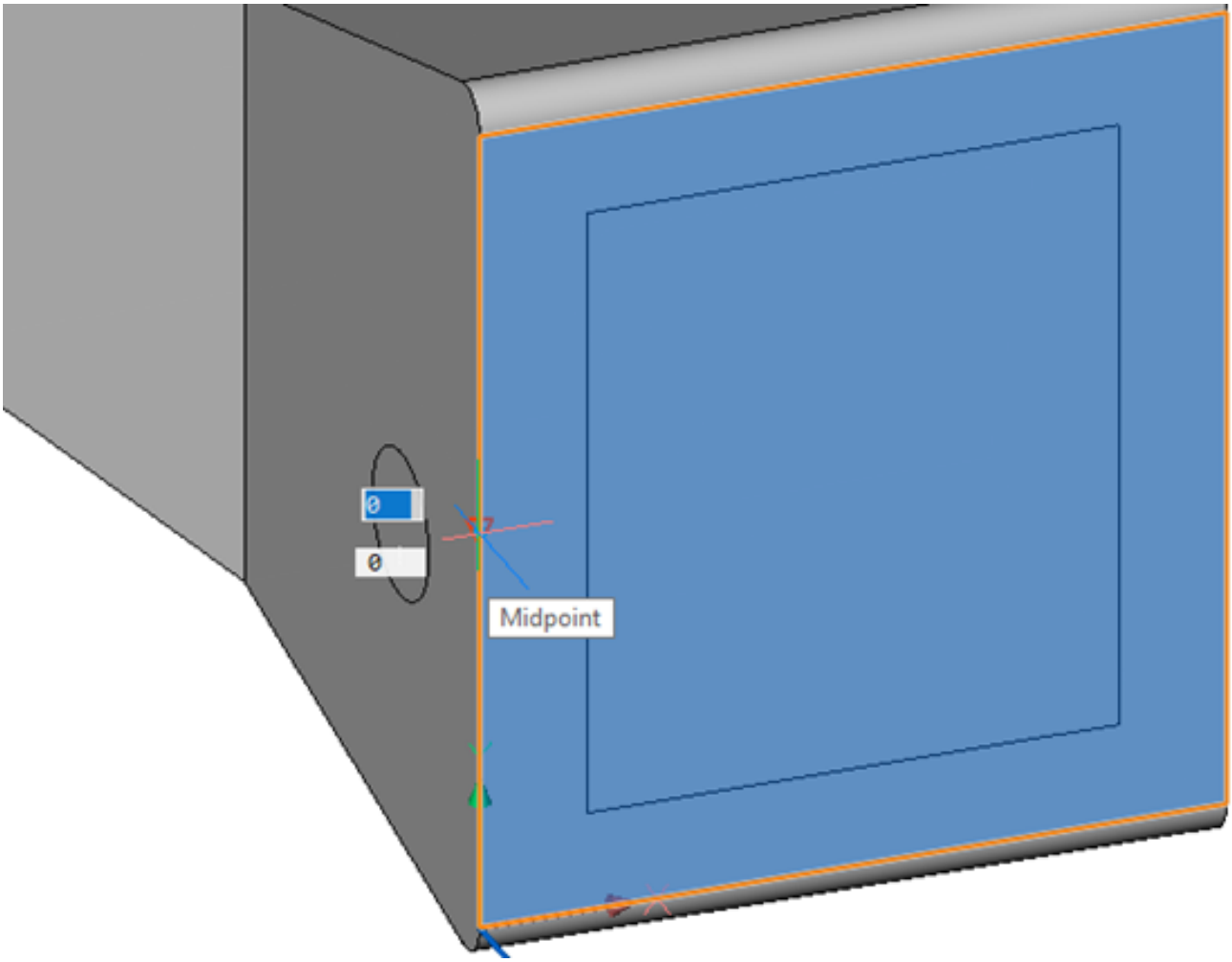


Figure 3.61

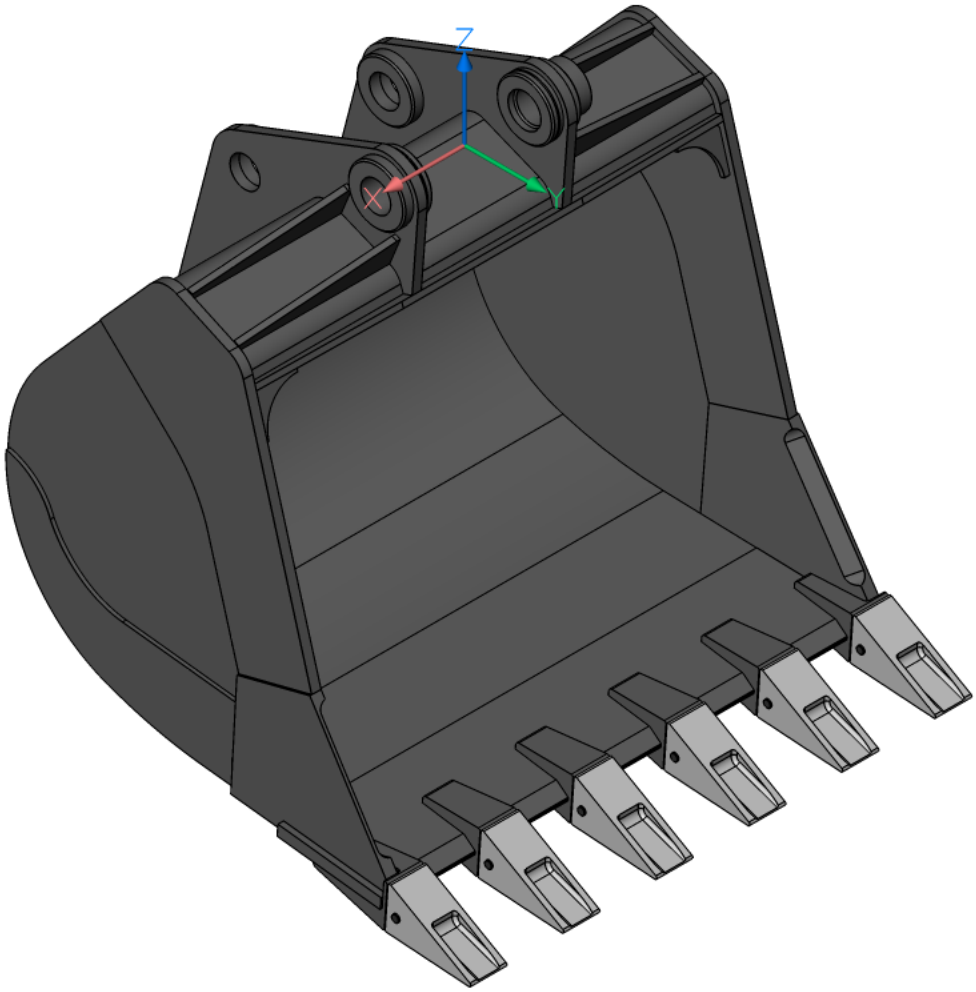


Figure 3.62

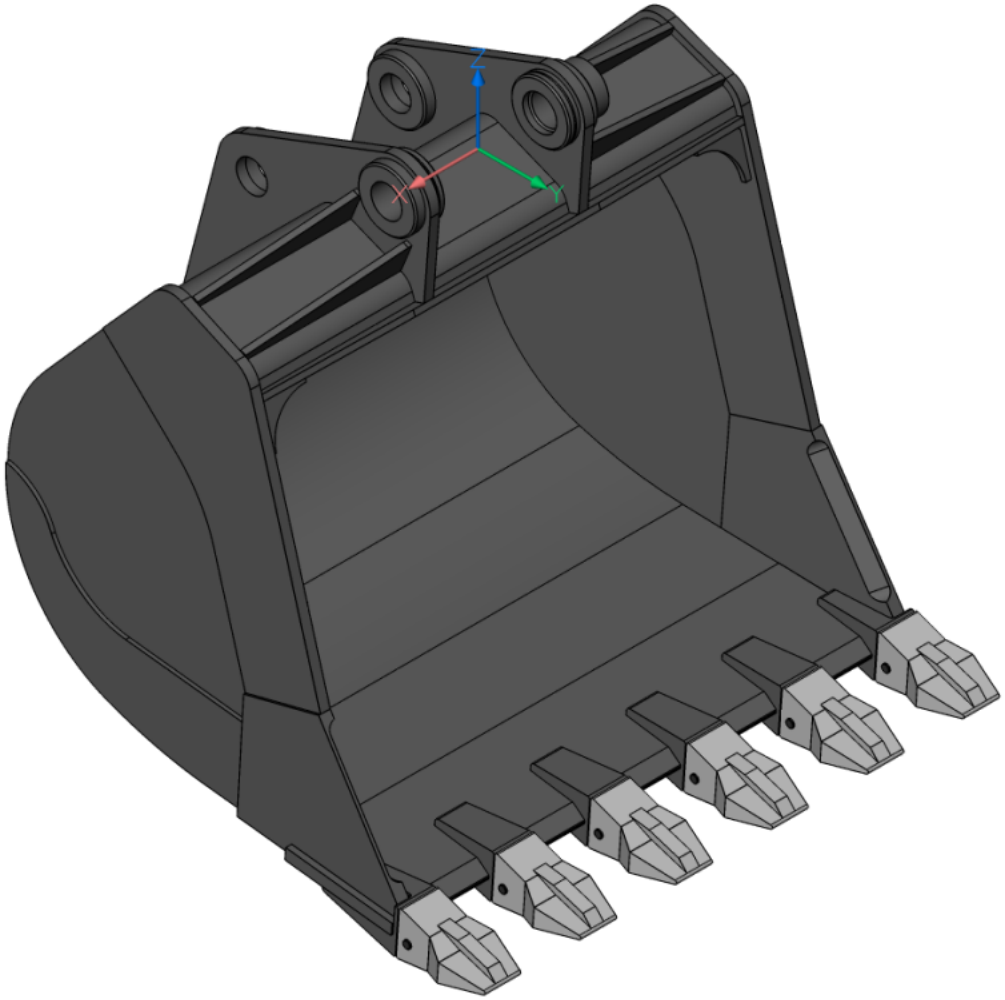


Figure 3.63

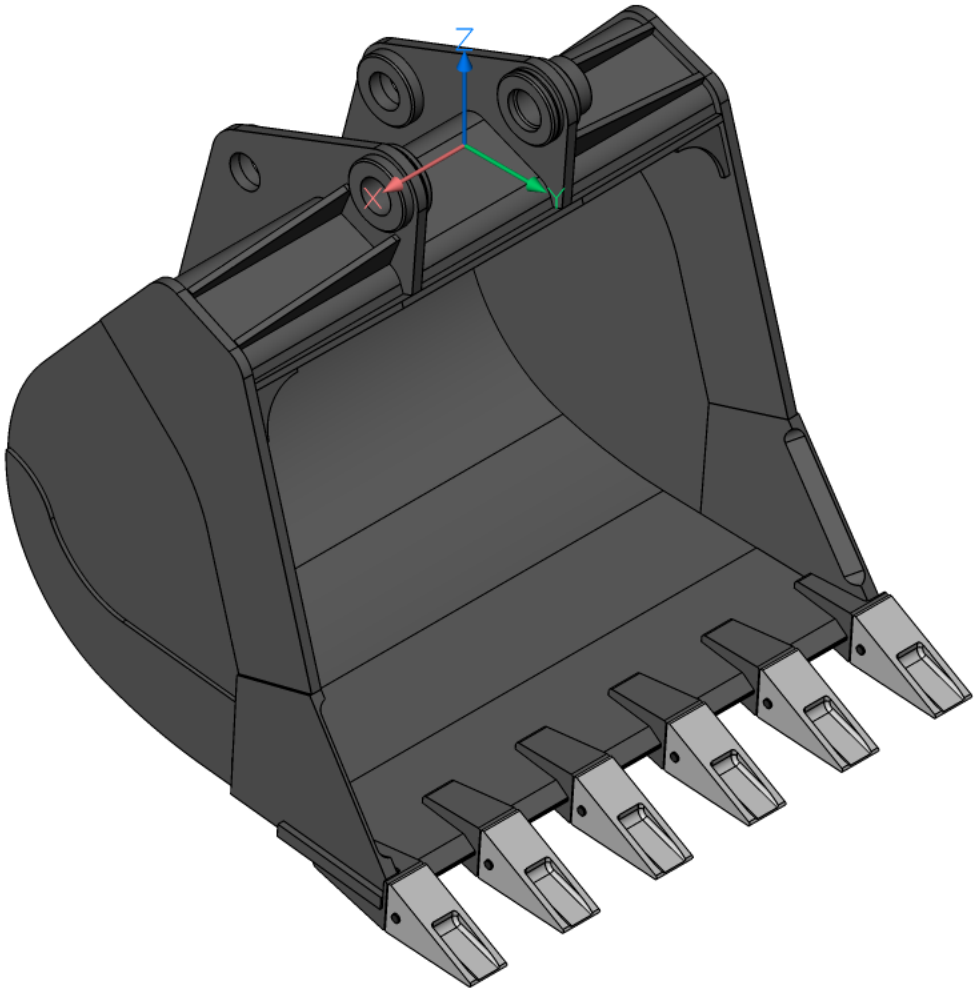
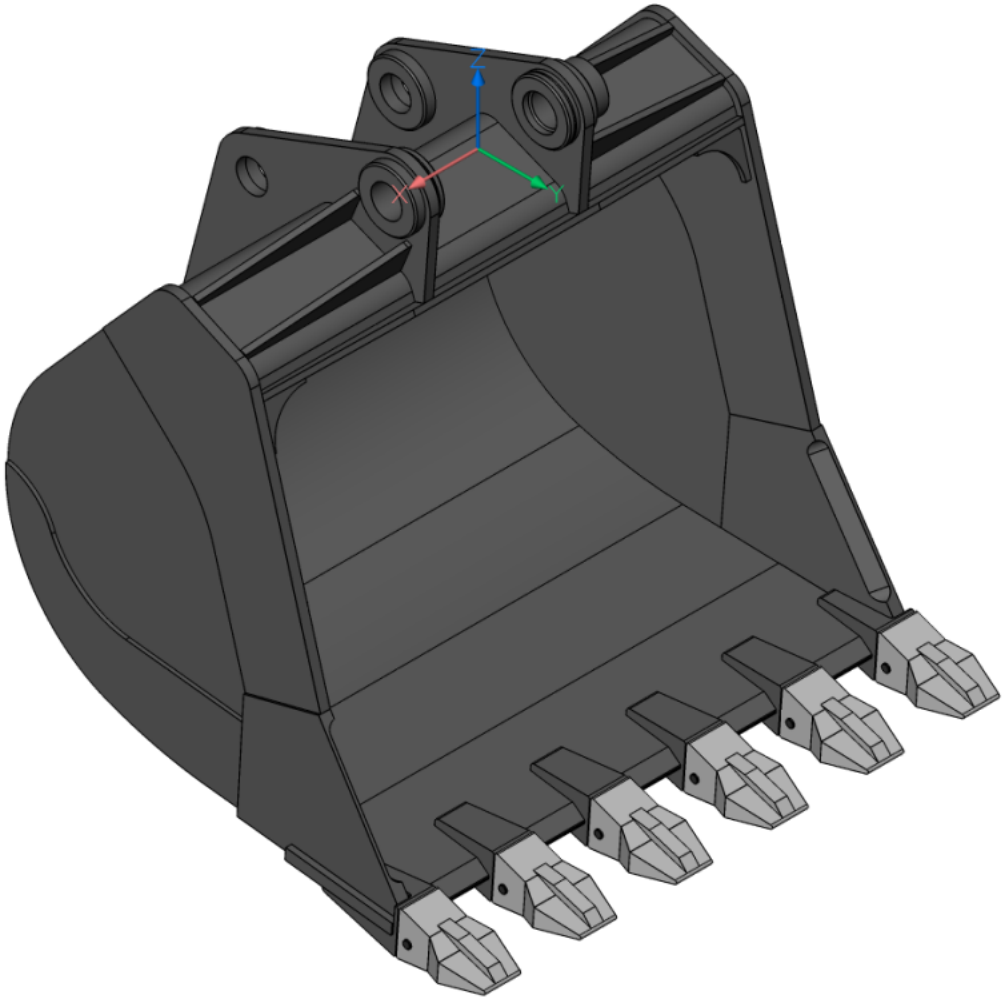


Figure 3.64



3.1.9 Ball bearing creating

3.1.10 Step 1. Preparing for the exercise

You need a**BricsCAD Platinum**license for this tutorial.

1. Create a new file in the **Mechanical**workspace.
2. Make sure that the **UCS orthographic** option in the Settings (6.27.23)dialog is switched on (the UCSORTHO system variable is 1). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039137253/mce

3. Set the **Facet resolution** option in the Settings (6.27.23)dialog (FACETRES system variable) to **3**.
4. Make sure that the Quad (2.2.3)cursor menu is active.
5. Select the **Solid** panel of the Ribbon (1.13.6). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachmen

Figure 3.65

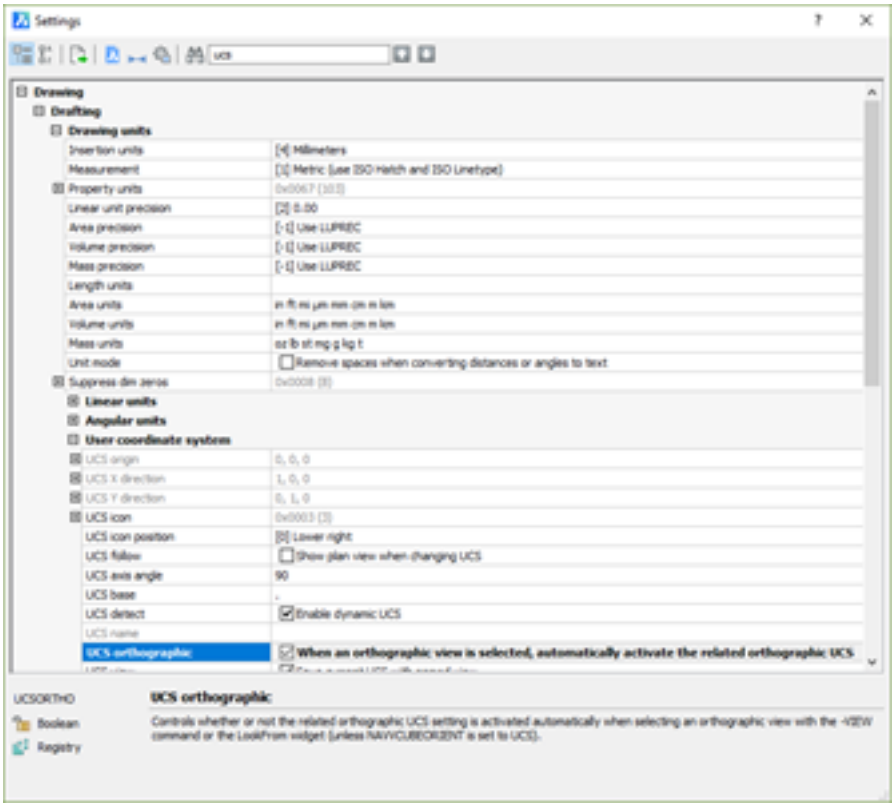
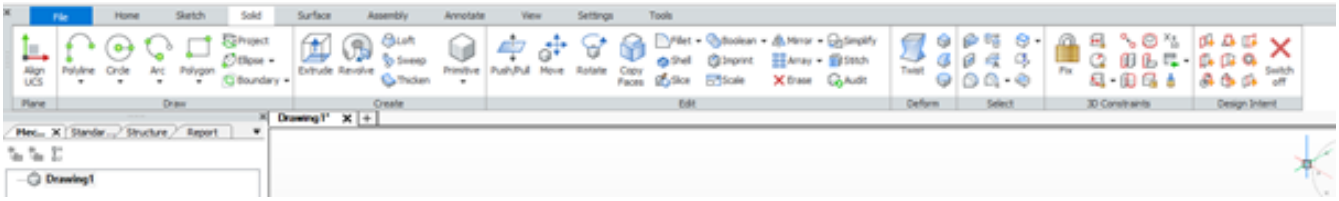


Figure 3.66



3.1.11 Step 2. Design external and internal rings without grooves

1. Click the **Cylinder (6.9.60)** ( ) tool button on the **Solid** | **Create** panel in the Ribbon (1.13.6). Set **0,0,-15** as the center point, **115** as the radius, and **30** as the height. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/article
2. Select the upper edge of the cylinder.
3. Select **Offset (6.23.3)**( ) from the **Draw** section of the Quad (2.2.3).
4. Enter a distance of **10** and hit **Enter**.
5. Hover over the circle.
6. Select**Solid Extrude (6.12.13)** ( )from the **Model** section of theQuad (2.2.3). Move the mouse in a negative direction. Extrude until the through-hole appears. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattach
7. Select the upper internal edge.
8. Select**Offset (6.23.3)**( ) from the **Draw** section of theQuad (2.2.3).
9. Enter a distance of **10** and hit **Enter**.
10. Select the circle.
11. Select**Solid Extrude (6.12.13)** ( )from the **Model** section of theQuad (2.2.3).
12. Move the mouse in a negative direction. Enter a distance of **30** and hit **Enter**. (\vref {fig:C:/Dev/Task-2448/images/helpbri
13. Select the upper edge of the internal solid.
14. Select**Offset (6.23.3)**( ) from the **Draw** section of theQuad (2.2.3).
15. Enter a distance of **10**, click **Enter**.
16. Select the circle you just created.
17. Select**Solid Extrude (6.12.13)** ( )from the **Model** section of theQuad (2.2.3).
18. Move the mouse in a negative direction. Extrude until the through-hole appears (you should now have two rings). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039165213/mceclip7.png})
19. Select the outer ring.
20. In the **Properties Panel (1.12.4)** click the **Color** field and change the value to **Cyan.** (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038306634/mceclip2.png})
21. Repeat steps 19 and 20 to change the color of the inner ring to **Green**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsysco

Figure 3.67

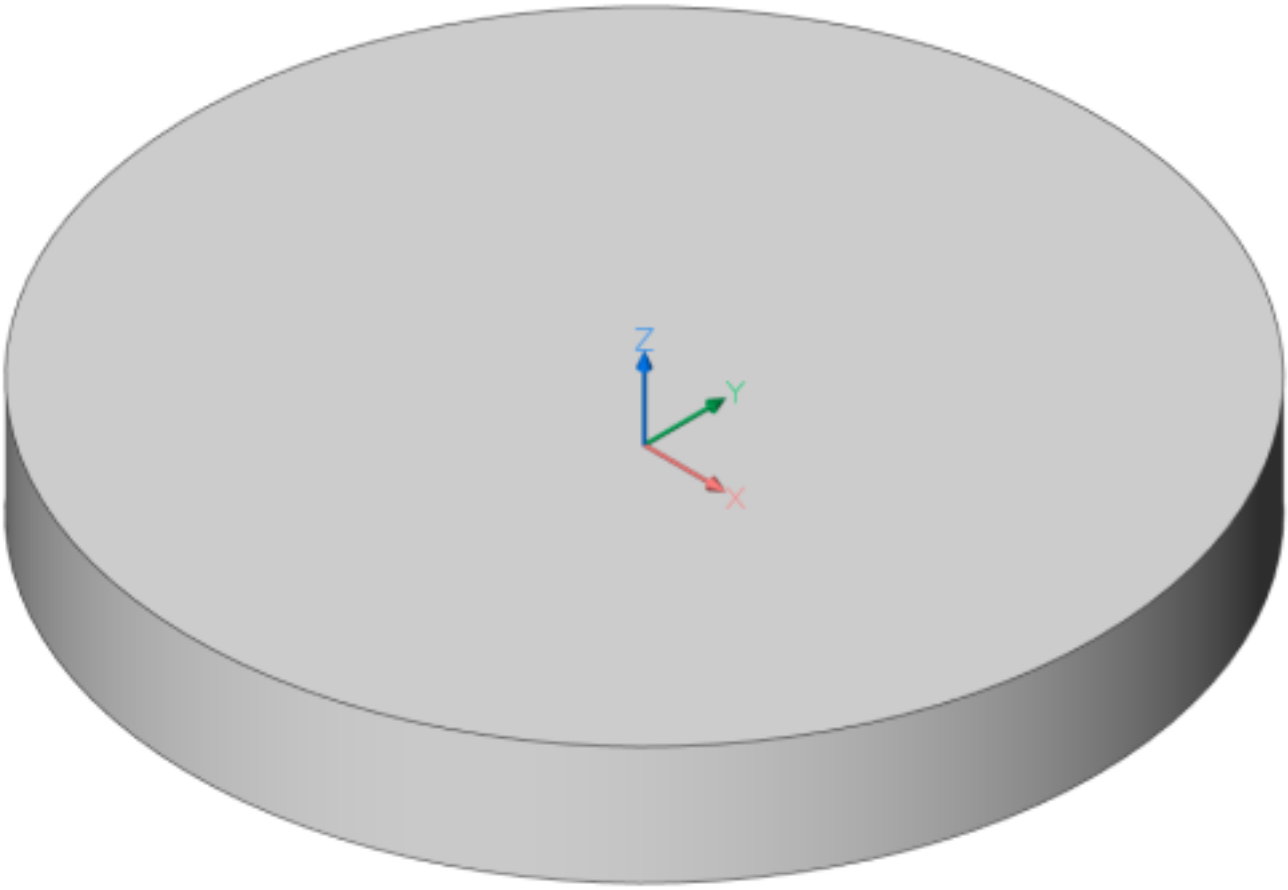


Figure 3.68

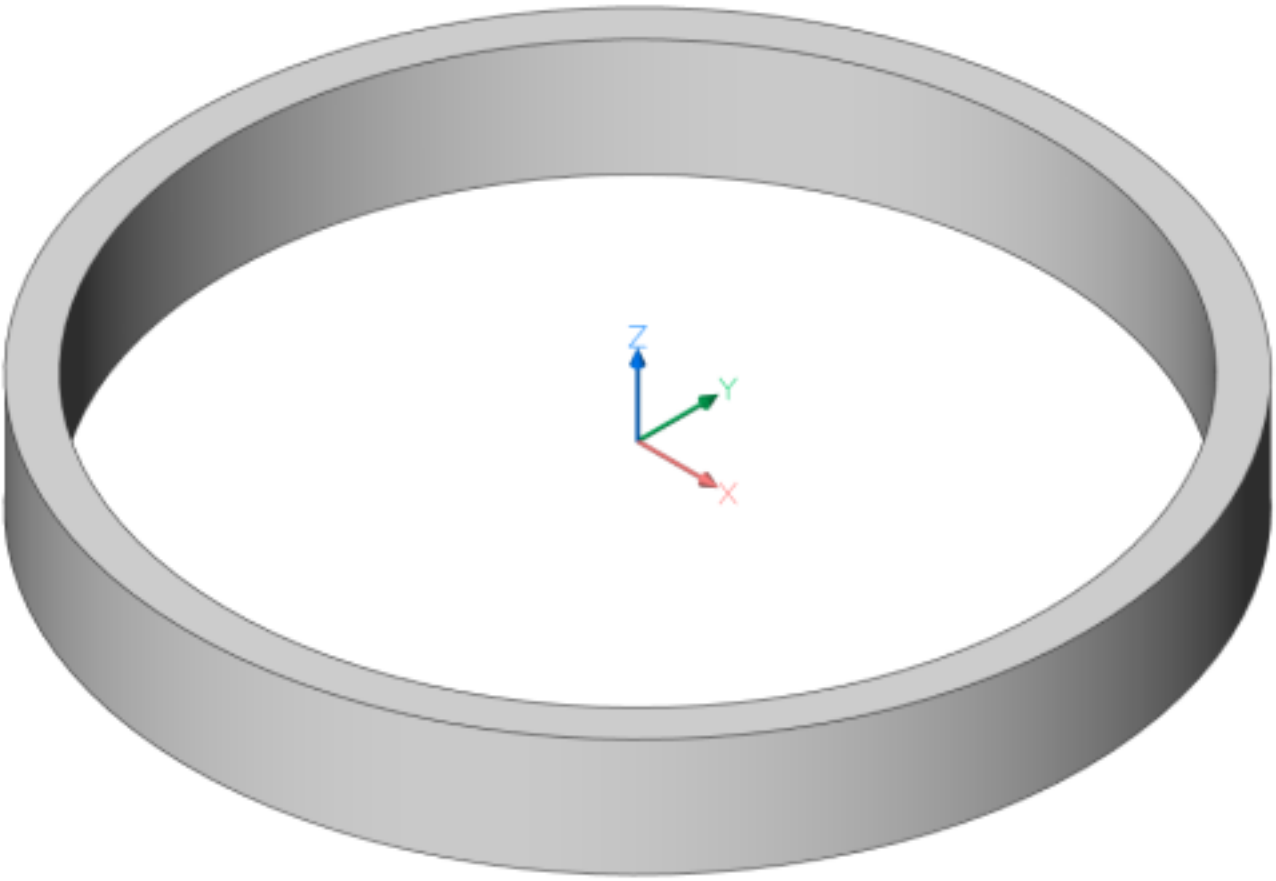


Figure 3.69

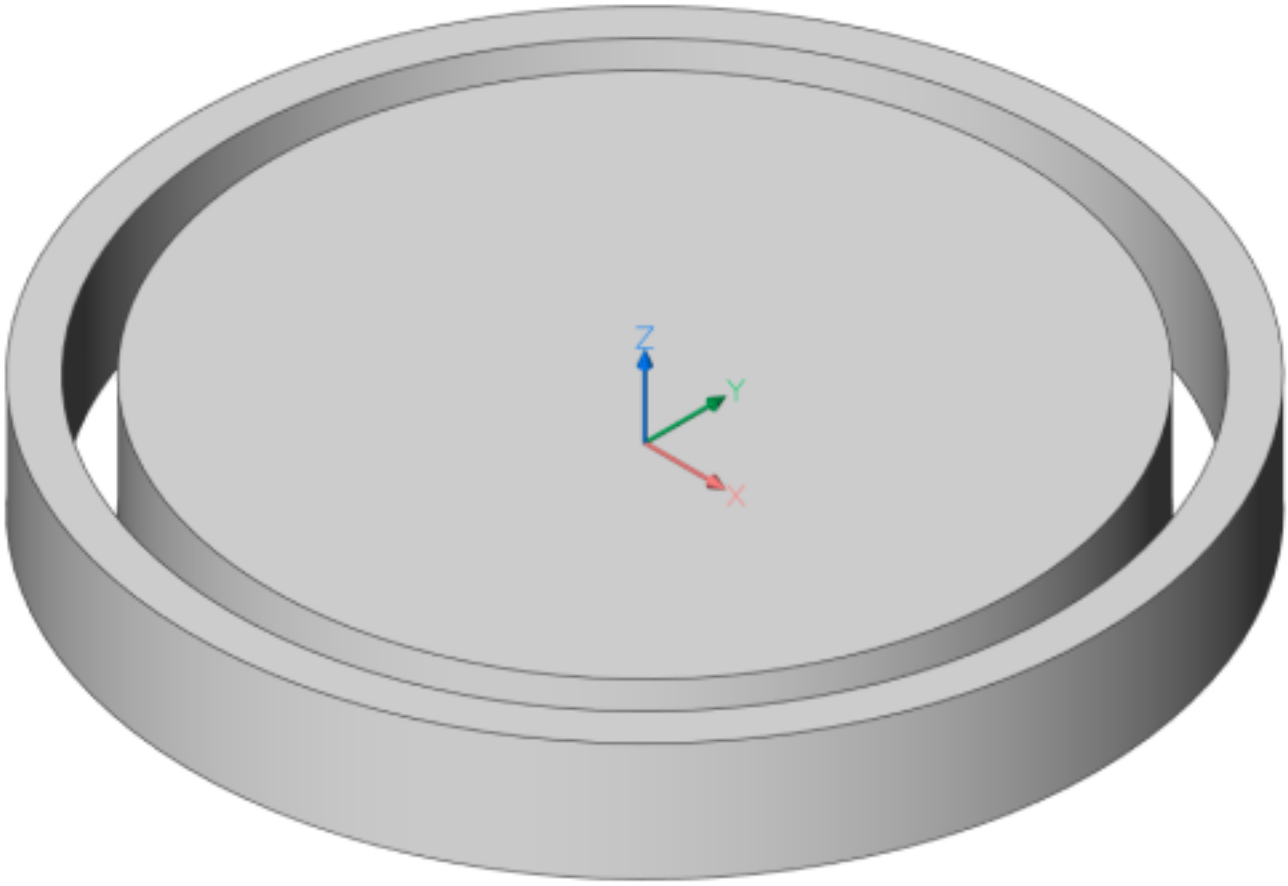


Figure 3.70

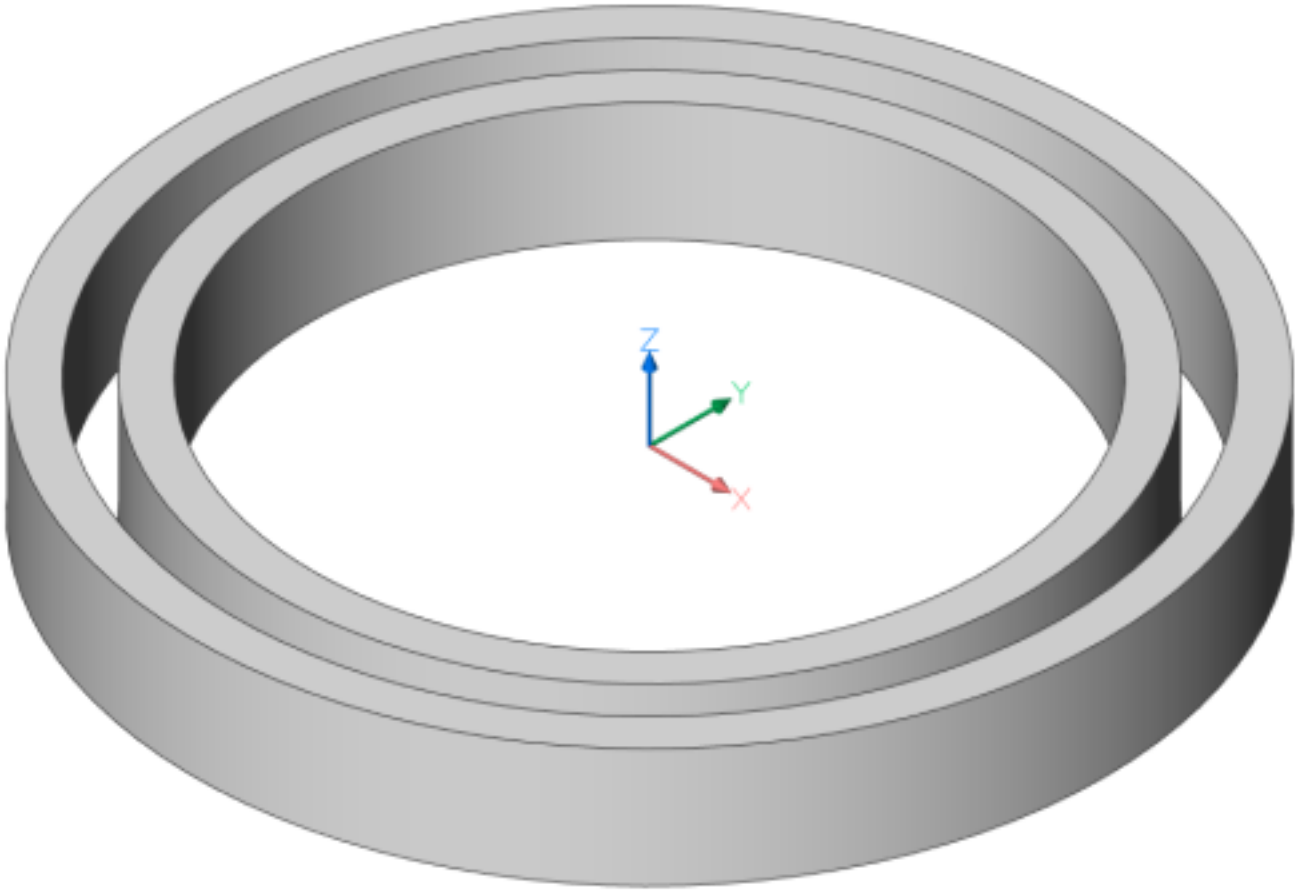


Figure 3.71

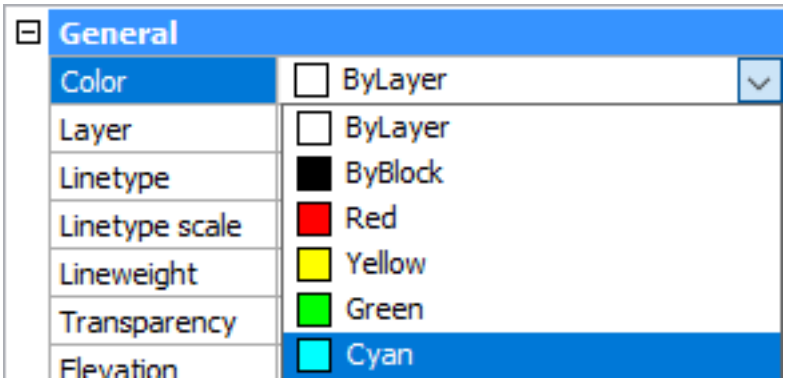
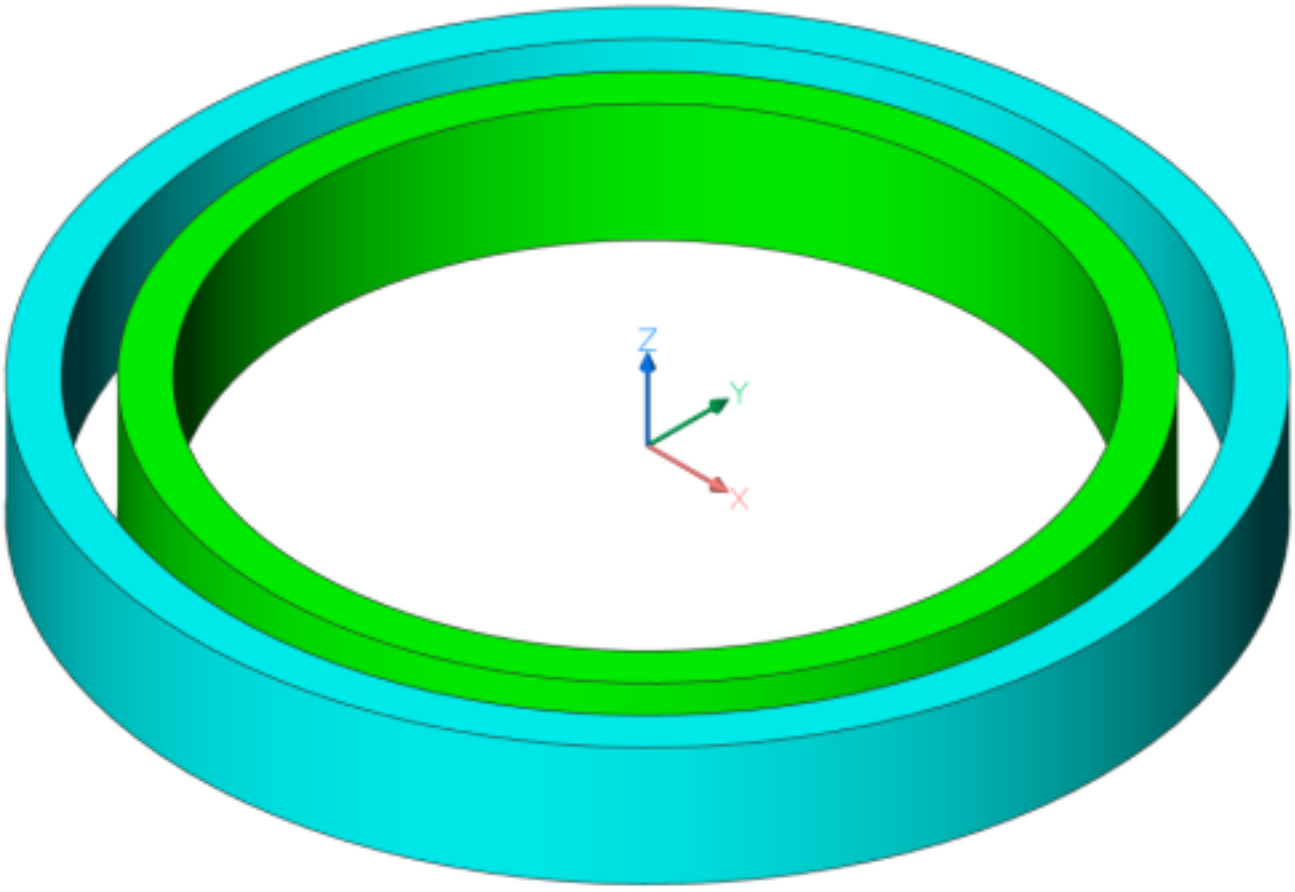


Figure 3.72



3.1.12 Step 3. Make both rings parametric

1. Call the context menu (right-click menu) on **Drawing*** component in the Mechanical Browser (3.2.2). Click the **Add new parameter (6.24.5)** ( ) button**.**
2. Select the new parameter (**v***) in the Mechanical Browser (3.2.2). In the bottom part of the Mechanical Browser (3.2.2), click the **Name** field and rename the parameter: **Bore.**
3. In the bottom part of the Mechanical Browser (3.2.2), click the **Expression** field and change parameter value to **170**.
4. In the bottom part of the Mechanical Browser (3.2.2), click the **Units** field and change parameter value to **mm**. **(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038310754/mceclip3.png})**

5. Repeat steps 3.1-3.4 to create the other parameters:
   - **OutsideDiameter = 230 mm**;
   - **OverallWidth = 30 mm**;
   - **Cover = (OutsideDiameter-Bore)/6**.

   (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039170513/mceclip9.png})

Figure 3.73

Parameter	
Name	Bore
Expression	170
Value	170
Description	
Exposed	Auto
Units	mm

6. Select the inside cylinder of the internal ring and select **Radius constraint (6.12.21)** ( ) from the **Constrain** section of the Quad (2.2.3)**.** Enter **Bore/2** as the parameter value.
7. Select the outside cylinder of the internal ring and select**Radius constraint (6.12.21)** ( )from the **Constrain** section of the Quad (2.2.3). Enter **Bore/2+Cover** as the parameter value.
8. Select the inside cylinder of the external ring and select**Radius constraint (6.12.21)** ( )from the **Constrain** section of the Quad (2.2.3). Enter **OutsideDiameter/2-Cover** as the parameter value.
9. Select the outside cylinder of the external ring and select**Radius constraint (6.12.21)** ( )from the **Constrain** section of the Quad (2.2.3)**.** Enter **OutsideDiameter/2** as the parameter value.
10. Select the top planar faces of the internal and external rings and click the **Coincident constraint (6.12.5)**( ) button tool in the **Solid | 3D constraints** panel of the Ribbon (1.13.6).
11. Select the bottom planar faces of the internal and external rings and click the**Coincident constraint (6.12.5)**( )button tool in the**Solid | 3D constraints**panel of the Ribbon (1.13.6).
12. Select the horizontal faces of the external ring.
13. Select **Distance constraint (6.12.12)**( ) from the **Constrain** section of the Quad (2.2.3)**.** Enter **OverallWidth** as the parameter value.
14. Select the**Distance constraint (6.12.12)**( )button tool in the **Solid | 3D constraints** panel of the Ribbon (1.13.6).
15. Select the top planar face of the external ring.
16. Click the **World | XY plane** buttons in the **Prompt menu**.
17. Enter **OverallWidth/2** as the parameter value. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachmer

18. Change the**Bore**,**OutsideDiameter**,and**OverallWidth**parameters to make sure they work.

Figure 3.74

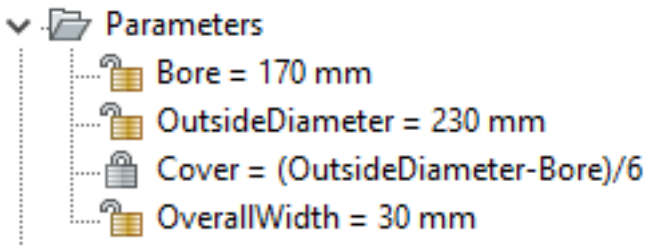
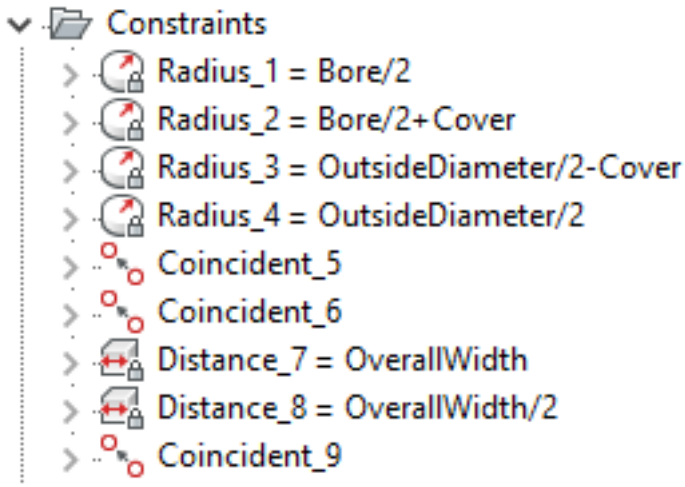


Figure 3.75



3.1.13 Step 4. Add parametric groves to rings

1. Switch to **Right** view in the LookFrom (6.20.39) widget by clicking the right triangle.
2. Select**Circle (6.9.20)** ( )from the **Draw** section of the Quad (2.2.3). Set **100,0,0** as a center point. Set a radius value of **10** and hit **Enter**.
3. Switch to **TopFrontRight** view in theLookFrom (6.20.39)widget to restore the vertical orientation of the Z-axis. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039178633/mceclip12.png})
4. Hover over the**circle.** Select **Revolve (6.12.22)** ( ) from the Quad (2.2.3)**.** Enter a rotation angle of **360°,** on the Z-axis. Use the **Subtract** option. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36003
5. Select the torus faces of the external and internal rings and click the**Coincident constraint (6.12.5)**( )button tool in the **Solid** | **3D constraints** panel of the Ribbon (1.13.6).
6. Enable the **Equal radius (1.18.3)**( ) option from the **Solid** | **Design Intent** panel in the Ribbon (1.13.6).
7. Repeat steps 3.1-3.4 to create a new parameter:**BallDiameter = 20 mm.**
8. Repeat step 3.6 to create a **Tube radius constraint** on the torus faces of the external and internal rings. Enter expressions of**BallDiameter/2**.
9. Hover over the torus face of the external ring and select**Radius constraint (6.12.21)** ( )from the **Constrain** section of the Quad (2.2.3)**.**
10. Select the **Rotation radius** option in **Prompt menu.**
11. Select the new Rotation radius constraint (**RotationRadius_***) in the Mechanical Browser (3.2.2). In the bottom part of the Mechanical Browser (3.2.2), click the **Expression** field and change the parameter value to **(Bore+OutsideDiameter)/4.**
12. Repeat steps 4.9-4.12 to create a **Rotation radius constraint** on the torus faces of the internal ring. Enter expressions of**(Bore+OutsideDiameter)/4**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/3
13. Change**Bore**,**OutsideDiameter**,**OverallWidth,** and **BallDiameter**parameters to make sure they work.

Figure 3.76

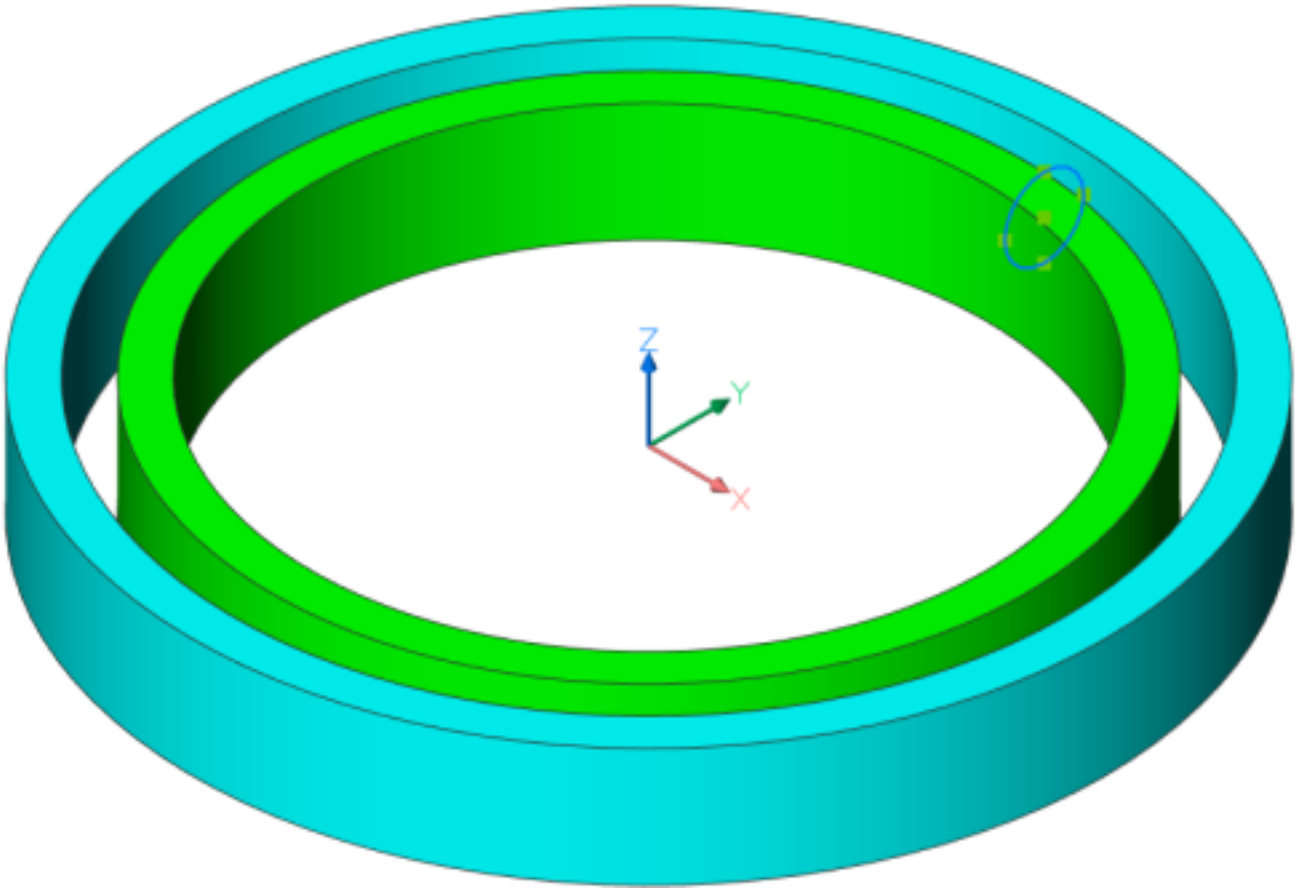


Figure 3.77

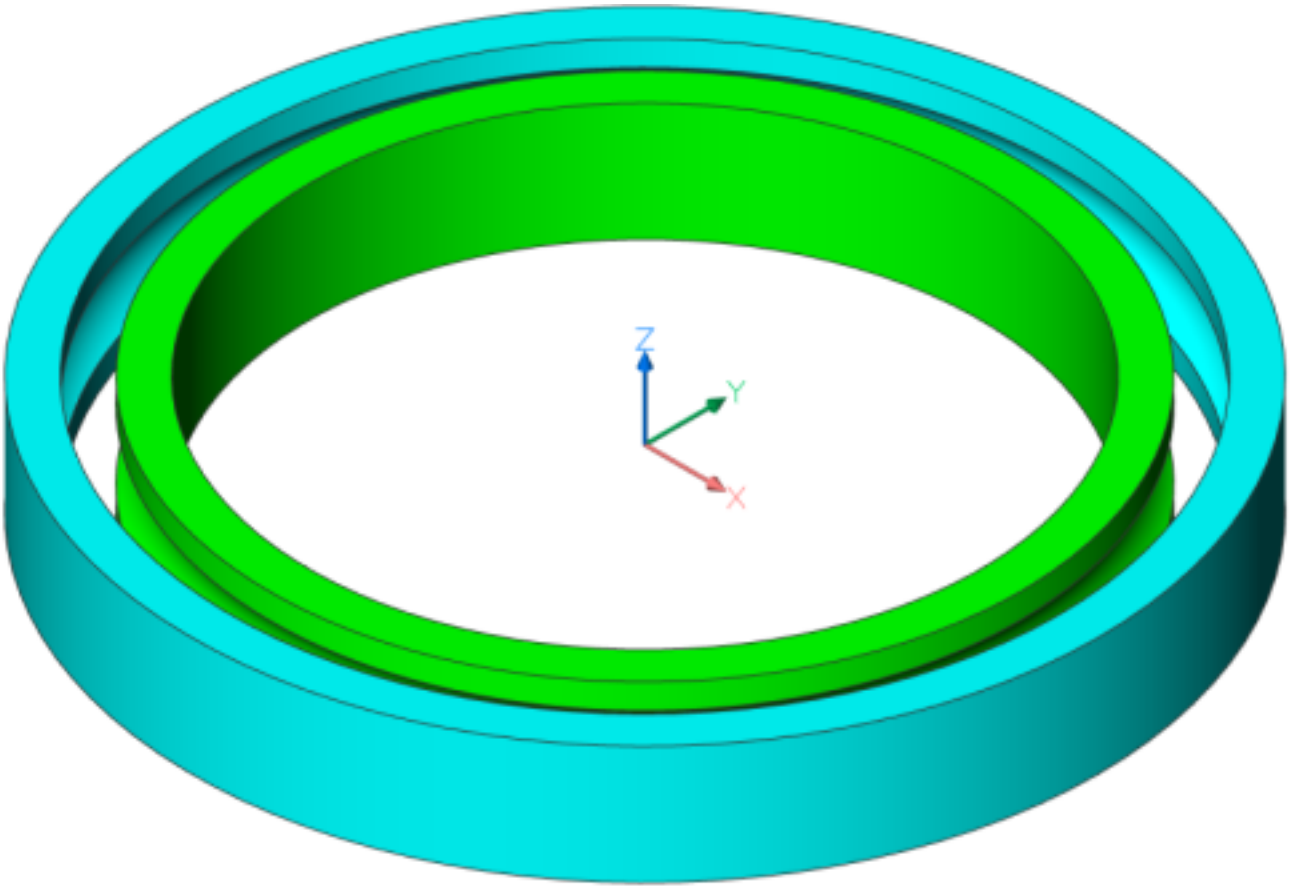
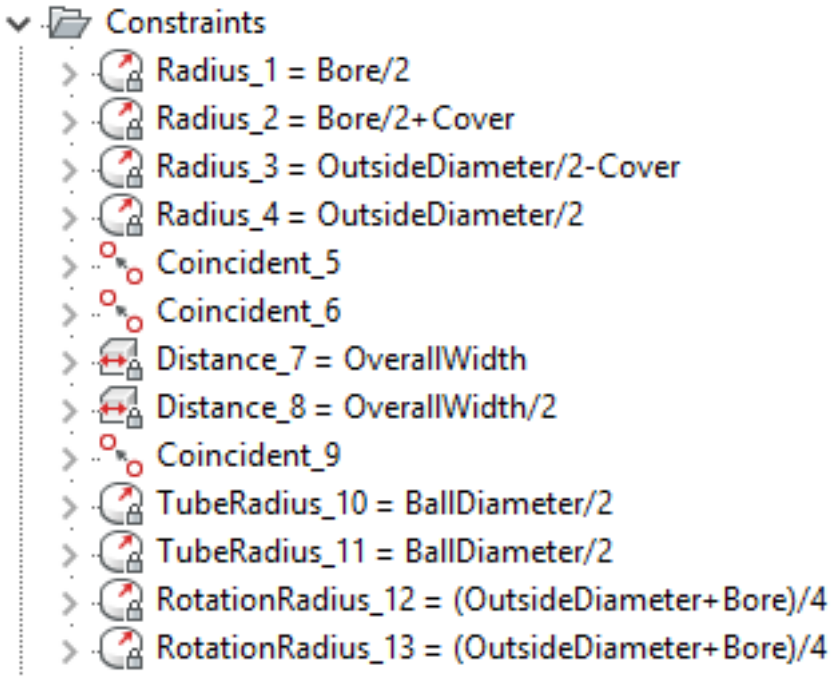


Figure 3.78



3.1.14 Step 5. Add balls

1. Click the **Sphere (6.27.64)** ( ) button in the **Solid** | **Create** panel of the Ribbon (1.13.6). Set the center point to **100,0,0**,the radius to **10** and hit **Enter**.
2. Select the sphere you just created.
3. In the **Properties Panel** click the **Color** field and change the value to**Red. (\vref {fig:C:/Dev/Task-2448/images/helpbricsy**
4. Select**Radius constraint (6.12.21)** ( )from the **Constrain** section of the Quad (2.2.3). Enter an expression of**10.**
5. Select the sphere.
6. Click the **Polar Array (6.4.23)** ( ) button tool in the **Solid** | **Edit** panel of the Ribbon (1.13.6). Enter **0,0,0** as the center point.
7. Select **the radius constraint, inside the array,** in the Mechanical Browser (3.2.2)and enter an expression of**BallDiameter/2**.
8. Repeat steps 3.1-3.4 to create a new parameter:**NumberOfBalls = 6.**
9. Select array (**Polar_***) in the Mechanical Browser (3.2.2)**.** On the bottom part of the Mechanical Browser (3.2.2), click the **Radius** field and change the parameter value to **(OutsideDiameter+Bore)/4, Items=NumberOfBalls** and**Angle=360/NumberOfBalls.(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039**
10. Change**Bore,OutsideDiameter,BallDiameter,** and**NumberOfBalls**parameters to make sure they work. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039318914/mceclip3.png})

Figure 3.79

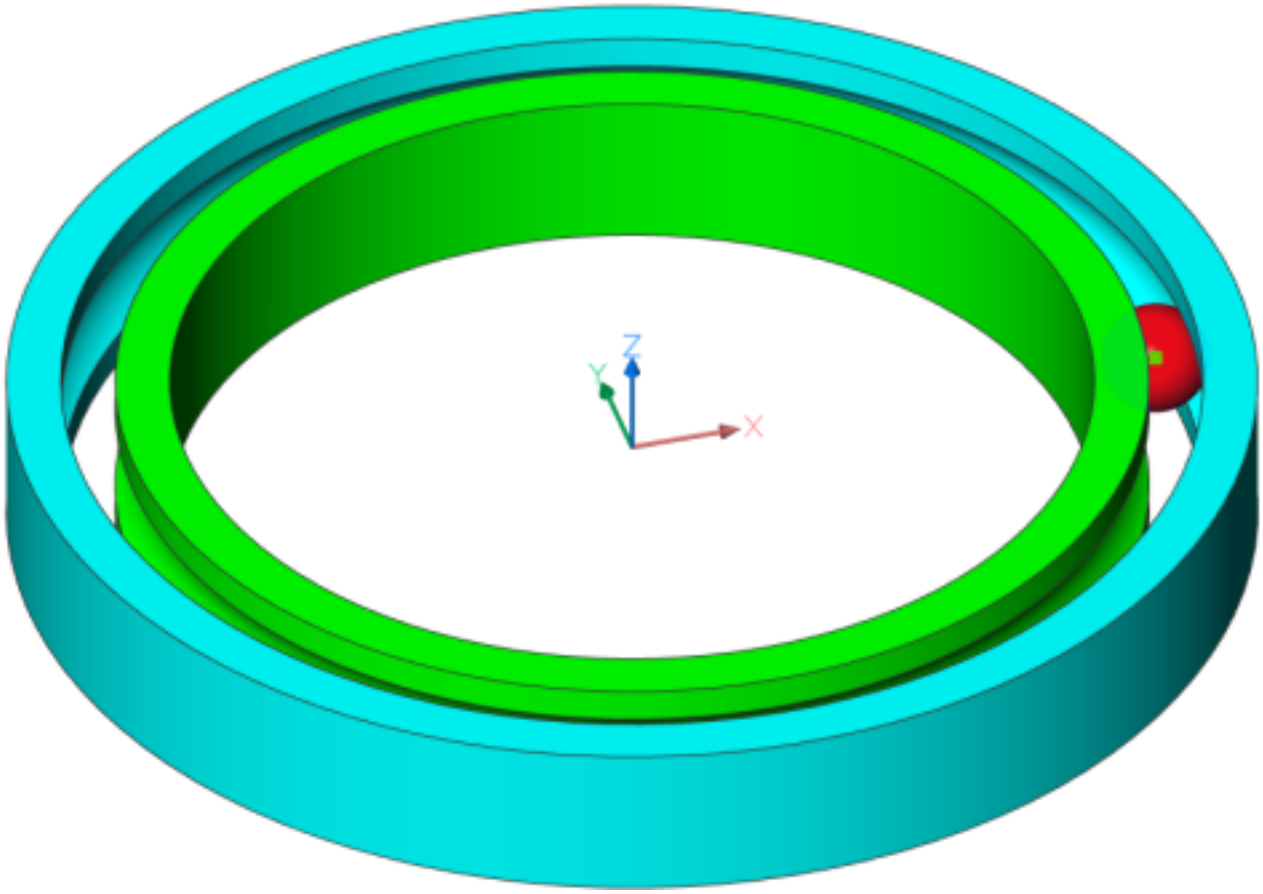
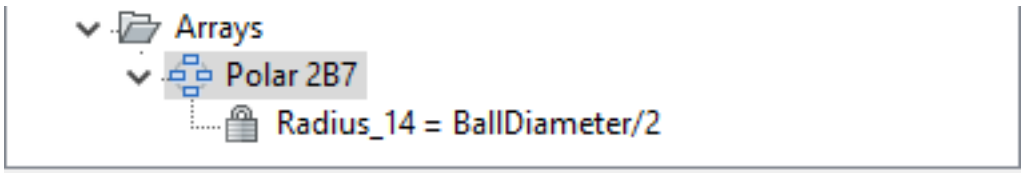
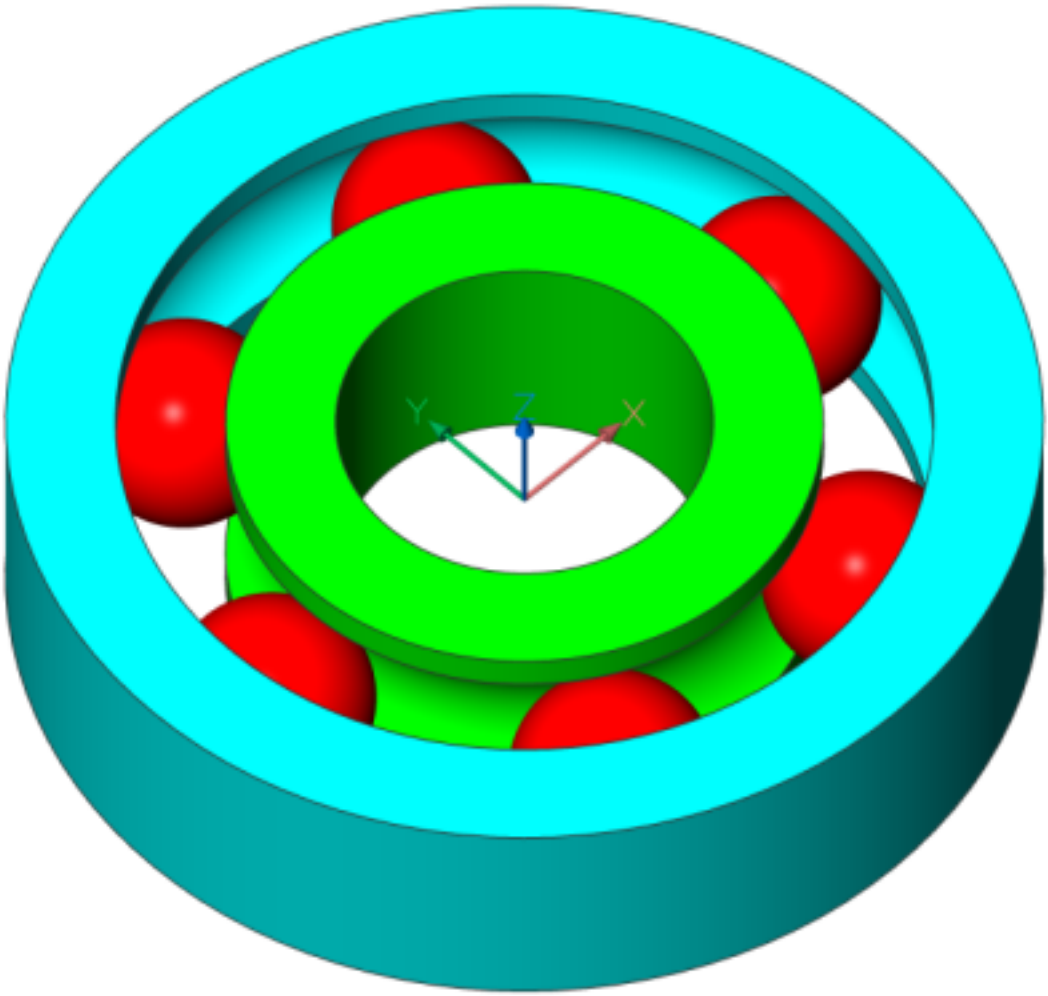


Figure 3.80



Array (Polar)	
Direction	
Radius	= (OutsideDiameter +Bore)/4
Items	= NumberOfBalls
Angle between items	= 360/NumberOfBalls
Fill angle	360°
Rows	1
Row spacing	30 mm
Row elevation increment	0 mm
Levels	1
Level spacing	30 mm
Rotate items	Yes

Figure 3.81



3.1.15 Step 6. Add design table

1. Open a BallBearing.xlsxtable as a native table using a spreadsheet program. Export it to BallBearing.csvfile.
2. Click the **DesignTable (6.10.26)** () button in the **Solid** | **3D constraints** panel of the Ribbon (1.13.6)and select the BallBearing.csvfile.
3. Select the**Type**parameter in the Mechanical Browser (3.2.2)and select an arbitrary configuration from the drop-down list. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040159313/mceclip1.
4. Select**Bore**and**OutsideDimeter**parameters in Mechanical Browser (3.2.2)and set **Exposed** properties to **On** for each of them. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040159233/mceclip0.p
5. Save as **ball_bearing.dwg**file.

Figure 3.82

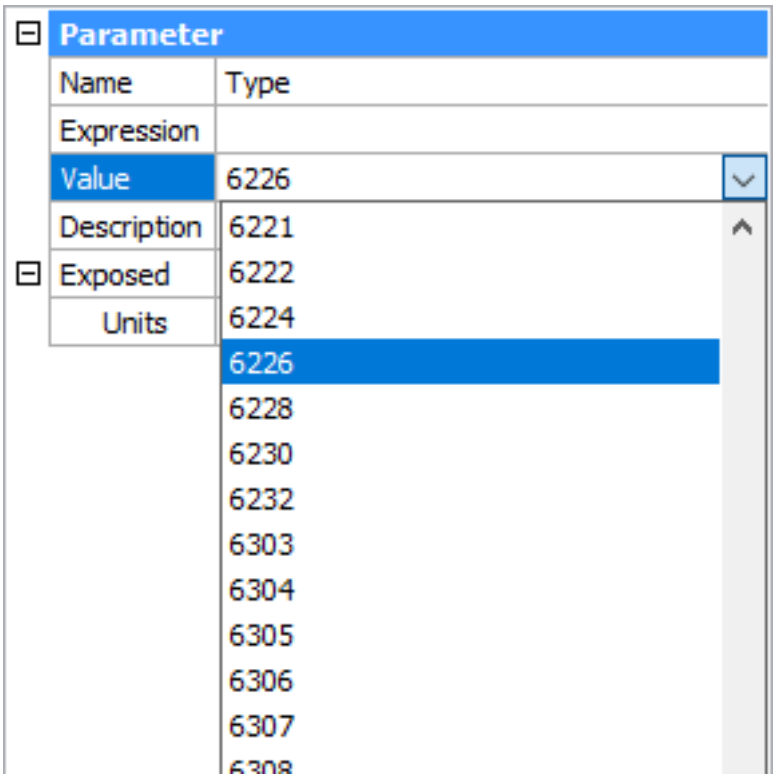
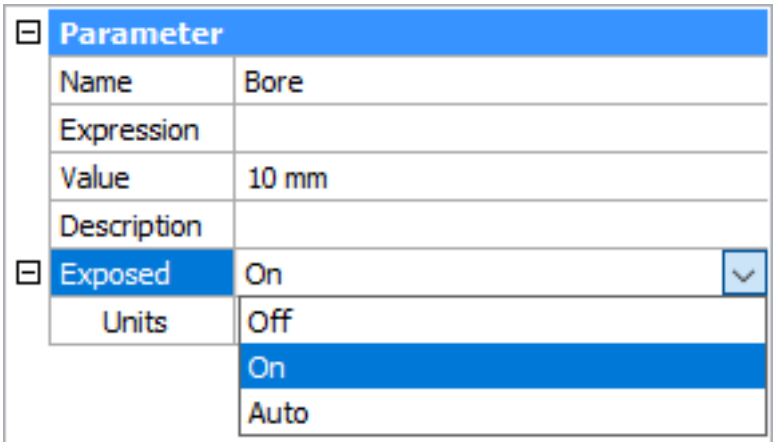


Figure 3.83



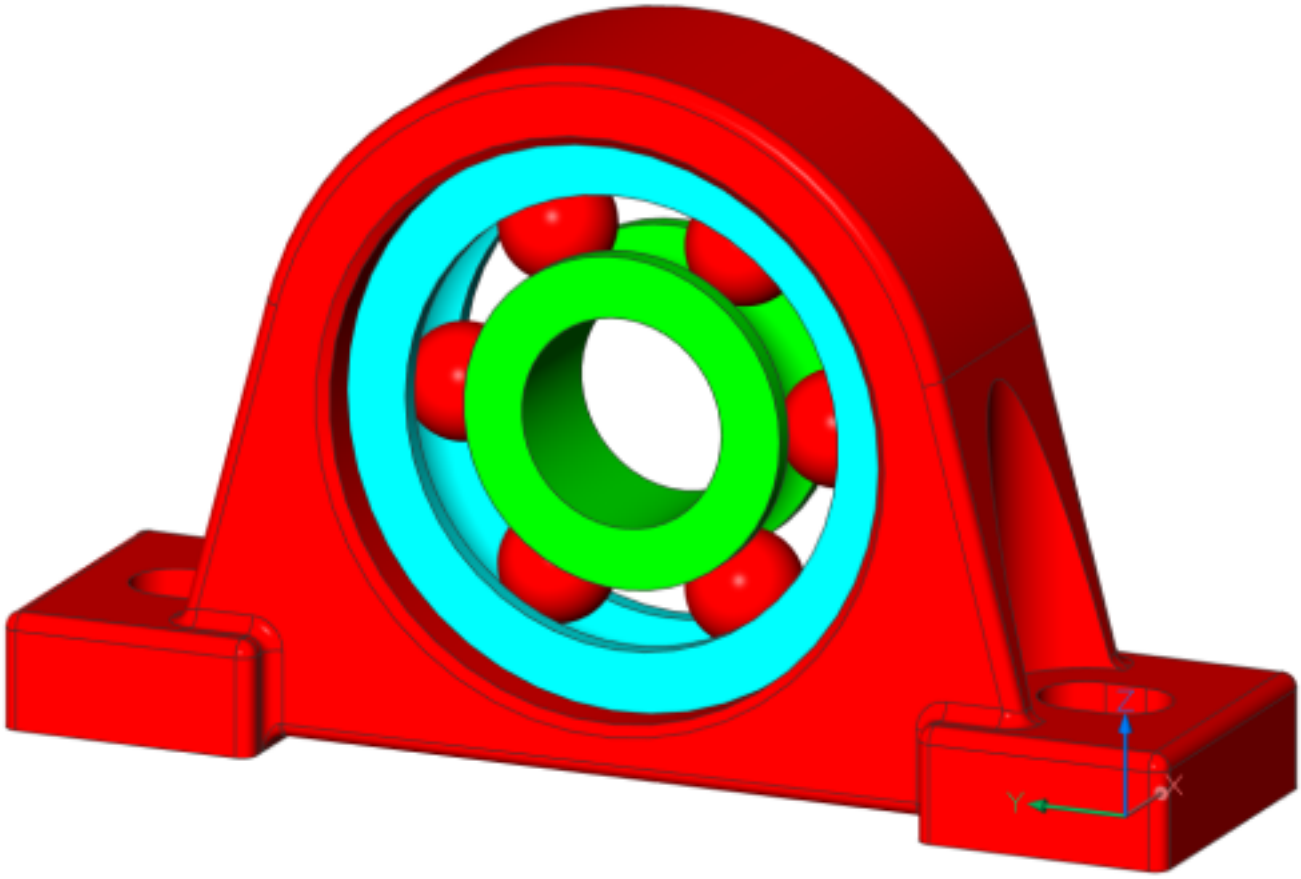
3.1.16 Step 7. Insert the bearing in an assembly

1. Open the Bearing_support_110.dwgfile.
2. Click the **Insert Component (6.8.12)** ( ) button tool on the **Assembly** | **Modify** panel of the Ribbon (1.13.6)and select the ball_bearing.dwgfile.
3. Repeat step 3.10 to set the bearing of the axle the correct position using the**Coincident constraint (6.12.5)**( ).
4. Select the **ball bearing** in the Mechanical Browser (3.2.2). Use the drop-down list for the parameters **Bore** and**OutsideDiameter.** Set an appropriate value for one of them and then repeat for the other.

NOTE	The bearing type is changed automatically.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040159933/mceclip2.png})

Figure 3.84



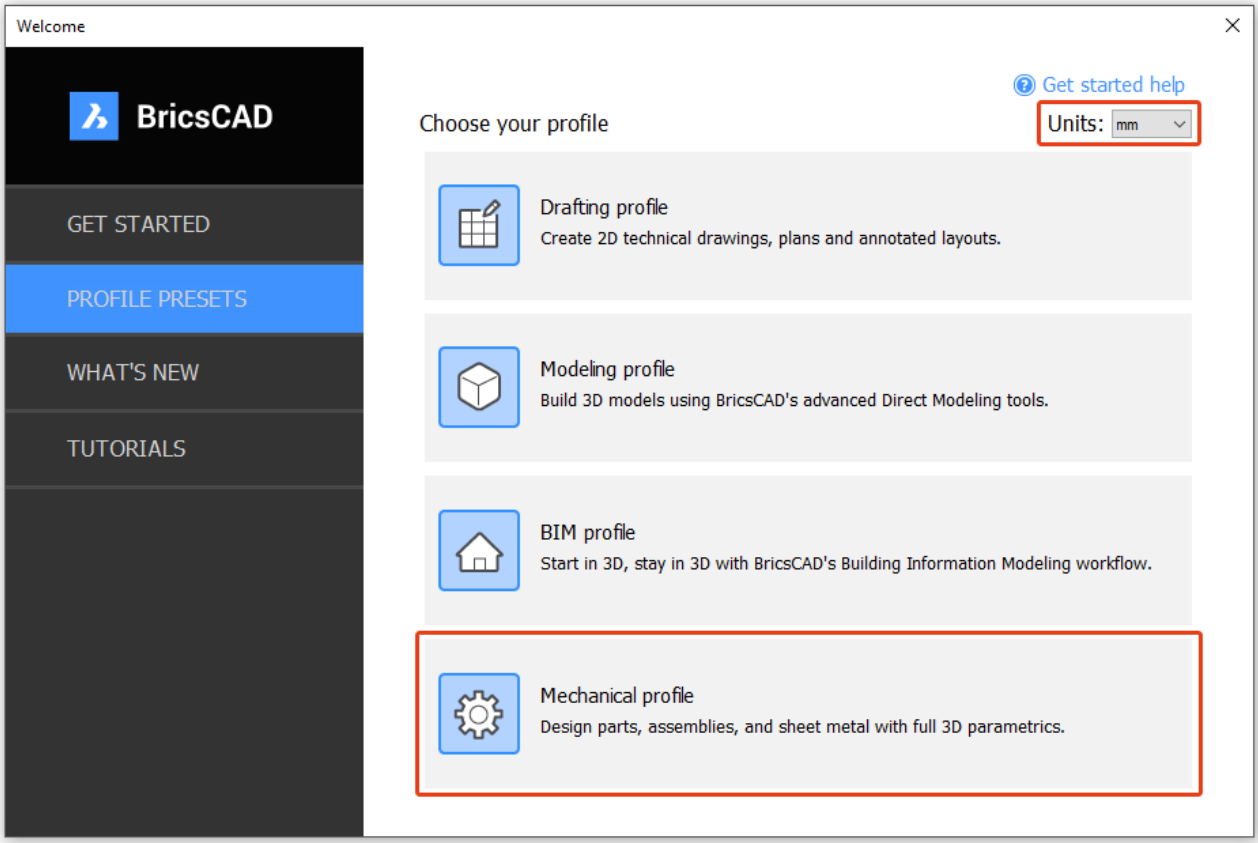
3.2 Getting Started

3.2.1 Mechanical Profile and Workspace

Get Started

To start a mechanical design project, choose the proper units for your document and selectthe Mechanical profile preset: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/GettingStarted/MechanicalProfil
The Mechanical profile switches the current workspace to Mechanical and selects the appropriate basefile and settings that are appropriate for a mechanical design project.

Figure 3.85

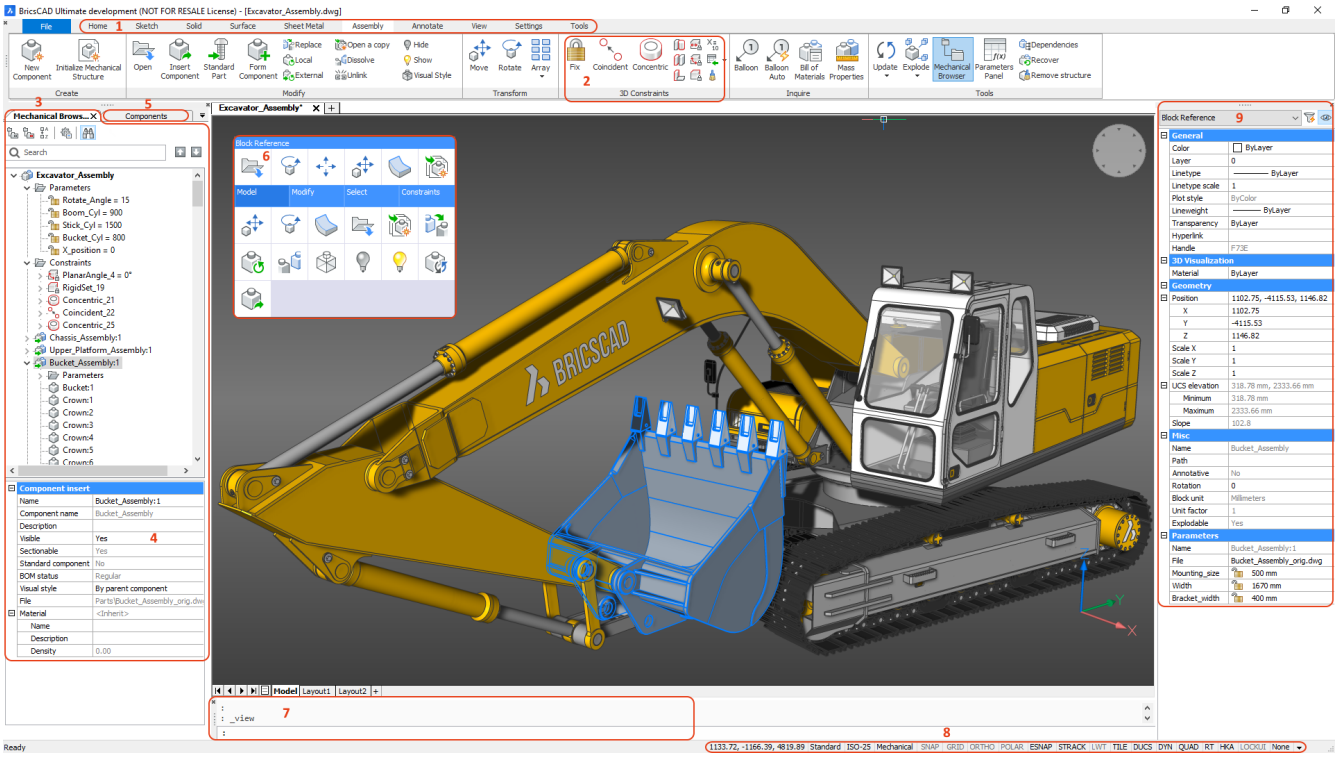


BricsCAD Mechanical Application Window

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/GettingStarted/MechanicalProfileAndWe

1. Ribbon tabs: click to switch to a particular tab.
2. Ribbon panel: every tab contains several panels, which logically group similar tools together.
3. Mechanical Browser: browse through a tree-like representation of your model with the possibility to highlight a particular node in the model area and edit its mechanical properties.
4. Mechanical properties of a particular node: some of are read-only, check the properties of the root node in the tree.
5. Components panel: insert mechanical standard parts and features.
6. Quad menu: apply a particular tool when you hover over an affected entity.
7. Command bar: type in the command.
8. Status bar: status and switch hot options.
9. Properties panel: adjust non-mechanical properties of selected entities; such as color or layer.

Figure 3.86


BricsCAD Ultimate development (NOT FOR RESALE License) - [Excavator_Assembly.dwg]
File
Home
Sketch
Solid
Surface
Sheet Metal
Assembly
Annotate
View
Settings
Tools
Create
Modify
Transform
3D Constraints
Inquire
Tools
Mechanical Brows...
Components
Excavator_Assembly*
Block Reference
Model
Modify
Select
Constraints
Component insert
General
3D Visualization
Geometry
Misc
Parameters
Model
Layout1
Layout2
_view
Ready
1
2
3
4
5
6
7
8
9

3.2.2 Mechanical Browser

Commands: MECHANICALBROWSEROPEN (6.8.29), MECHANICALBROWSERCLOSE. (6.8.28) The Mechanical Browser panel displays the hierarchy of components (3.6.3), features (3.8.3), arrays (3.6.5), parameters (1.19.3) and constraints (1.19.5) for the current drawing. They are displayed in a tree-like form: you can expand and collapse every node to see its child nodes (for example, nested components). The Mechanical Browser allows you to edit properties of a particular node, displayed at the bottom of the panel and calls different tools depending on what type is available in the context menu by right click on a particular node. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
With the Mechanical Browser you can:

- Click an element to highlight related geometric entities in the drawing window.
- Click an element to change its name.
- Right click an element to run a command for it.
- Edit properties of a selected element.

You can also group 3D constraints and features by entity, group elements by type, and sort them alphabetically using the buttons in the upper part of the Mechanical Browser. Use the gear icon you can control the way expressions and parameters are visualized in the Mechanical Browser. You can also search thru the whole tree by using the search tool. To open the Mechanical Browser do one of the following:

- Choose **Mechanical Browser** in the **Assembly** menu.
- Place the cursor on a toolbar, then right click and choose **Mechanical Browser** in the context menu.
- Call **MECHANICALBROWSEROPEN** command.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/GettingStarted/MechanicalBrowser2.pn

Figure 3.87

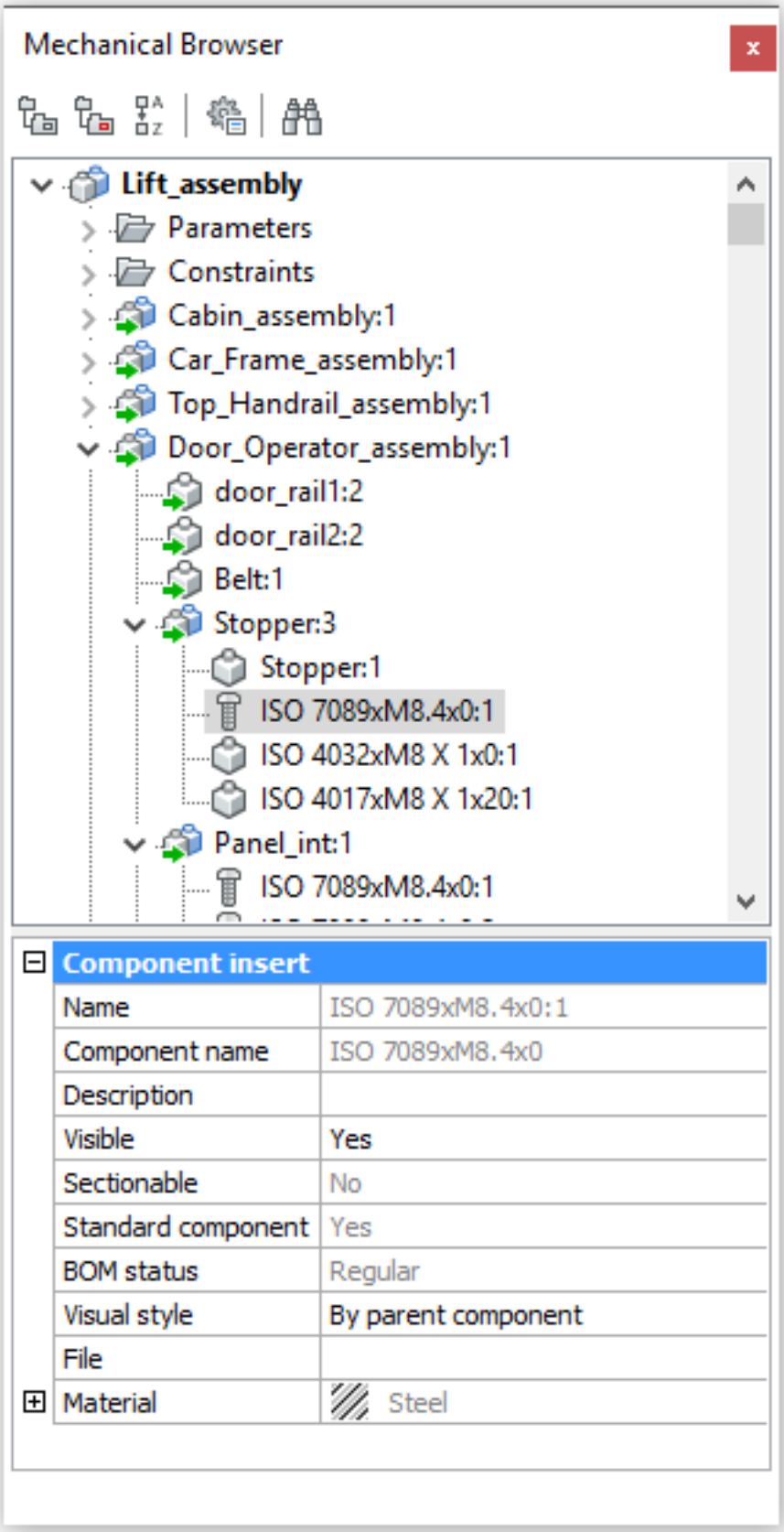
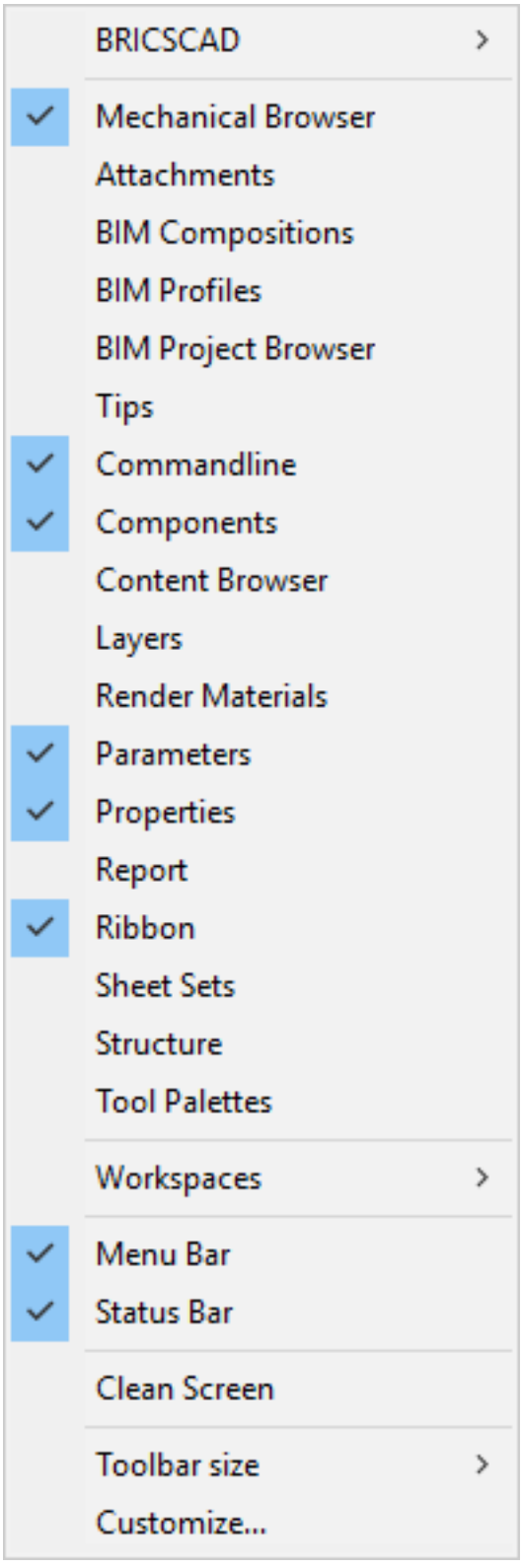

Mechanical Browser
Lift_assembly
Parameters
Constraints
Cabin_assembly:1
Car_Frame_assembly:1
Top_Handrail_assembly:1
Door_Operator_assembly:1
door_rail1:2
door_rail2:2
Belt:1
Stopper:3
Stopper:1
ISO 7089xM8.4x0:1
ISO 4032xM8 X 1x0:1
ISO 4017xM8 X 1x20:1
Panel_int:1
ISO 7089xM8.4x0:1
Component insert
Name
ISO 7089xM8.4x0:1
Component name
ISO 7089xM8.4x0
Description
Visible
Yes
Sectionable
No
Standard component
Yes
BOM status
Regular
Visual style
By parent component
File
Material
Steel

Figure 3.88


BRICSCAD
Mechanical Browser
Attachments
BIM Compositions
BIM Profiles
BIM Project Browser
Tips
Commandline
Components
Content Browser
Layers
Render Materials
Parameters
Properties
Report
Ribbon
Sheet Sets
Structure
Tool Palettes
Workspaces
Menu Bar
Status Bar
Clean Screen
Toolbar size
Customize...

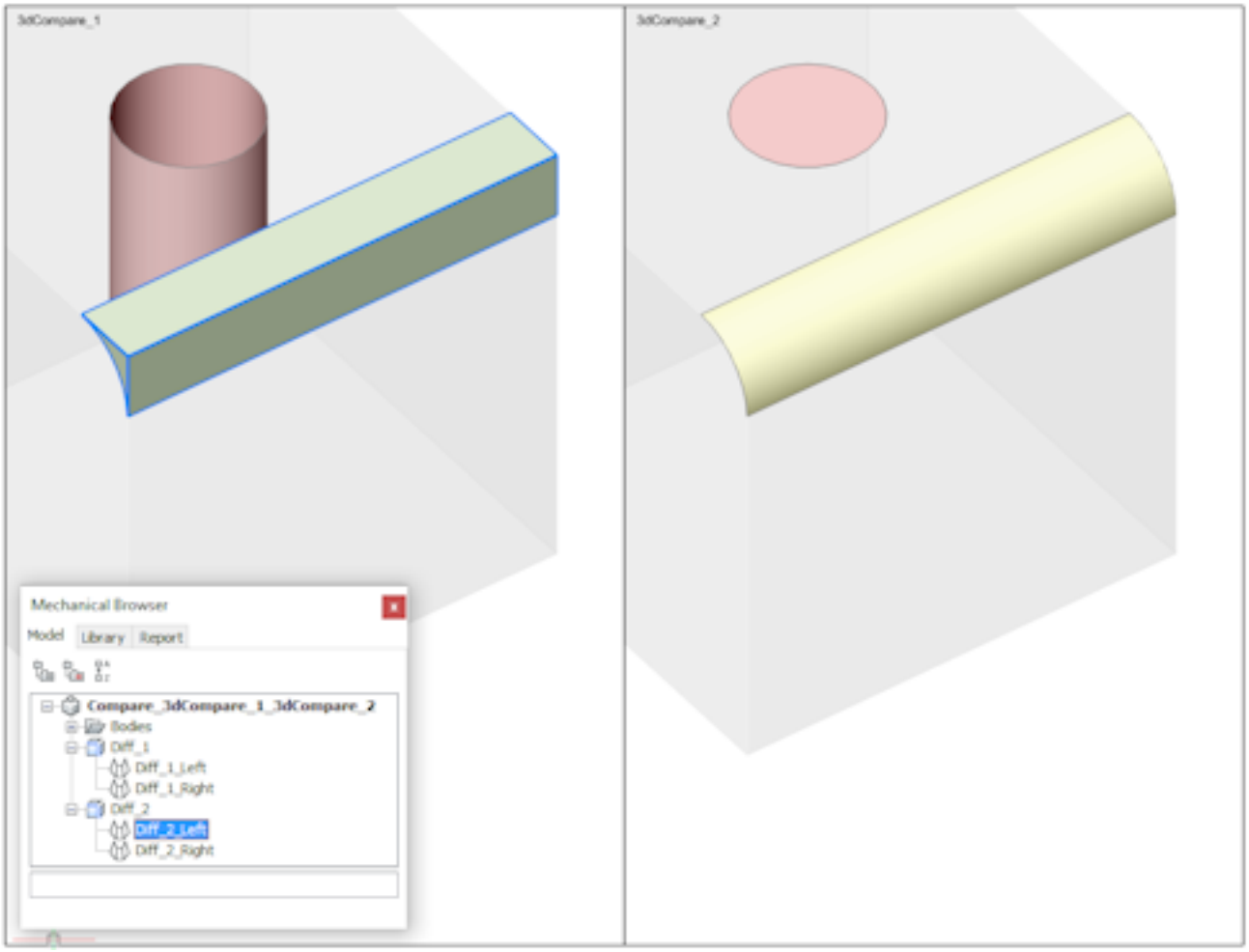
3.2.3 3D Comparison

Command: 3DCOMPARE (6.3.3) You can compare two 3D models, i.e. 3D solids and surfaces stored in two .dwg files, including ones imported in BricsCAD from other file formats. The tool efficiently processes large models and reports differences in the Mechanical Browser (3.2.2). Several visualization modes are supported.

Definitions

A 3D model is a .dwg file with one or more 3D solids or surfaces. When comparing two 3D models, BricsCAD first tries to automatically align them in 3D space to obtain the maximal similarity. Then all faces of these 3D solids and surfaces are compared geometrically with a given tolerance. If for every face in the first file an equal face in the second file exists and vice versa, such models are considered as equal. However, if for a particular face in one file there is no equal face found in another file, or if only part of this face is equal to a part of a face in the other file, then a difference is detected. Therefore a difference is a full face or a part of a face of a 3D solid or a surface, for which there are no equal parts of full faces found in the other file. Two or more differences are grouped together if they are geometrically adjacent to each other. A difference node is created in Mechanical Browser (3.2.2) for every group of adjacent differences. If you expand this node, you can see the particular differences it consists of. When you select the difference in the Mechanical Browser (3.2.2) the corresponding feature highlights in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/GettingStarted/3dCompareHighlight.png})

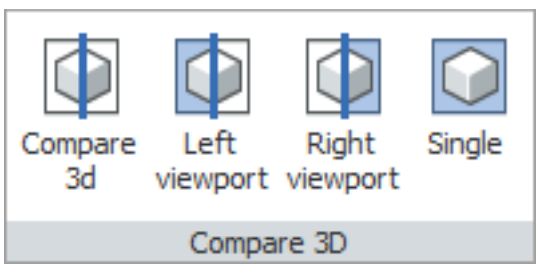
Figure 3.89


3dCompare_1
3dCompare_2
Mechanical Browser
Model
Library
Report
Compare_3dCompare_1_3dCompare_2
Bodies
Diff_1
Diff_1_Left
Diff_1_Right
Diff_2
Diff_2_Left
Diff_2_Right

Toolbars

The 3D Comparison procedure can be accessed using the tools on the **Compare 3D** ribbon panel on the **Tools** ribbon tab (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/GettingStarted/3dCompareTools.png

Figure 3.90



Visualization

The 3DCOMPARE (6.3.3) command creates a new .dwg document named Compare_<File_1>_<File_2>.dwg. The compared models are attached as external references to this drawing, A special layout named **Comparison**in which two viewports exist, one for each of the two models. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/M
The display is controlled through the 3DCOMPAREMODE (1.13.11) system variable. If the value of 3DCOMPARE-MODE (1.13.11) is different from 0 (zero), the corresponding model becomes semi-transparent, while the differences are displayed using different colors (each difference is painted in its individual color). Using the Mechanical Browser (3.2.2) context menu you can zoom in on a particular difference. To display the differences, do one of the following:

- Click the **Left Viewport** ( ) and/or the **Right Viewport** ( ) tool button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
  Double click in one of the viewports, then use the mouse/keyboard shortcuts to manipulate the display (e.g. zoom in/out or rotate). The display in the other viewport is synchronized automatically, allowing to compare the same part in both models in detail.

- Click the **Single** ( ) tool button to see the differences in model space. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys

Figure 3.91

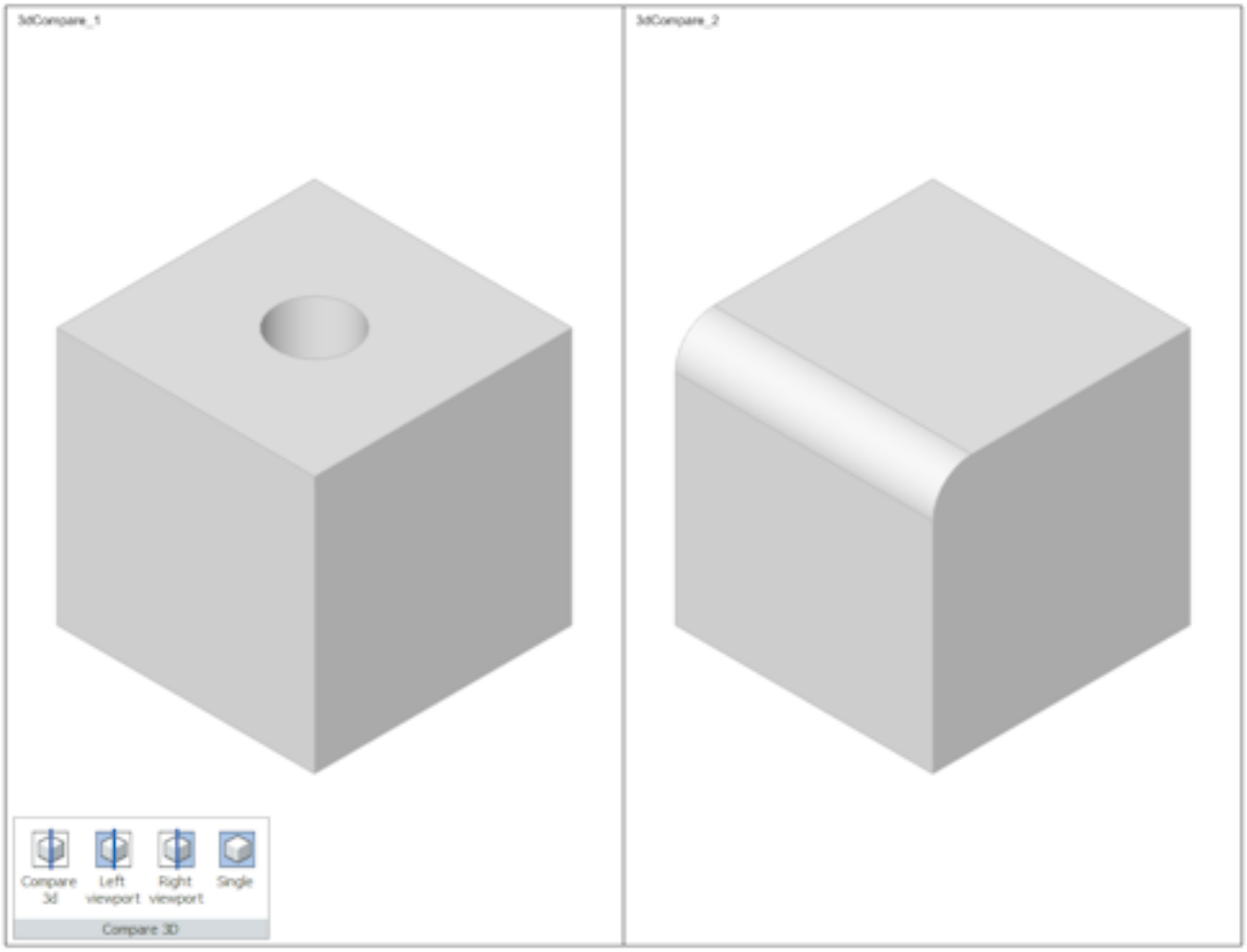

3dCompare_1
3dCompare_2
Compare 3d
Left viewport
Right viewport
Single
Compare 3D

Figure 3.92

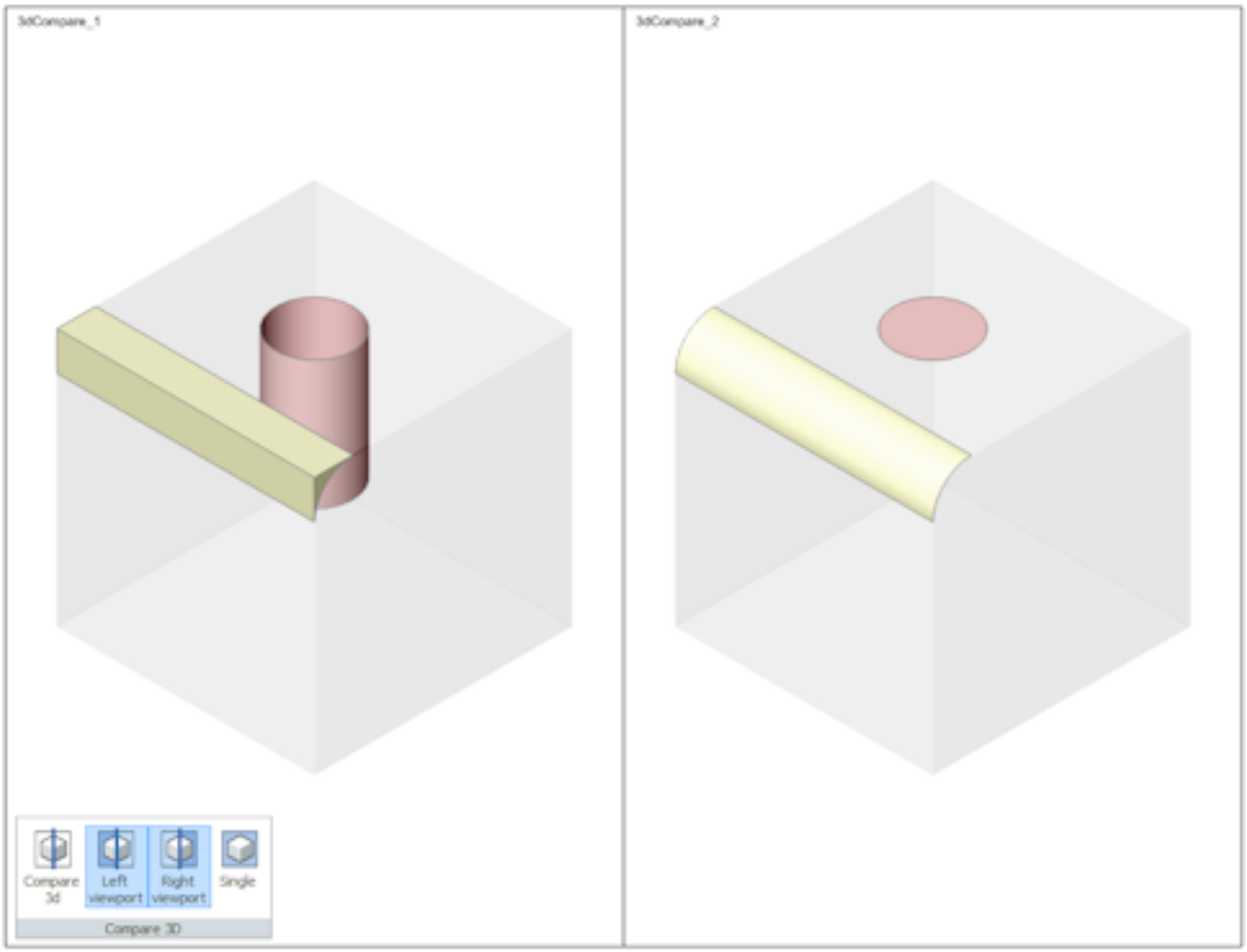
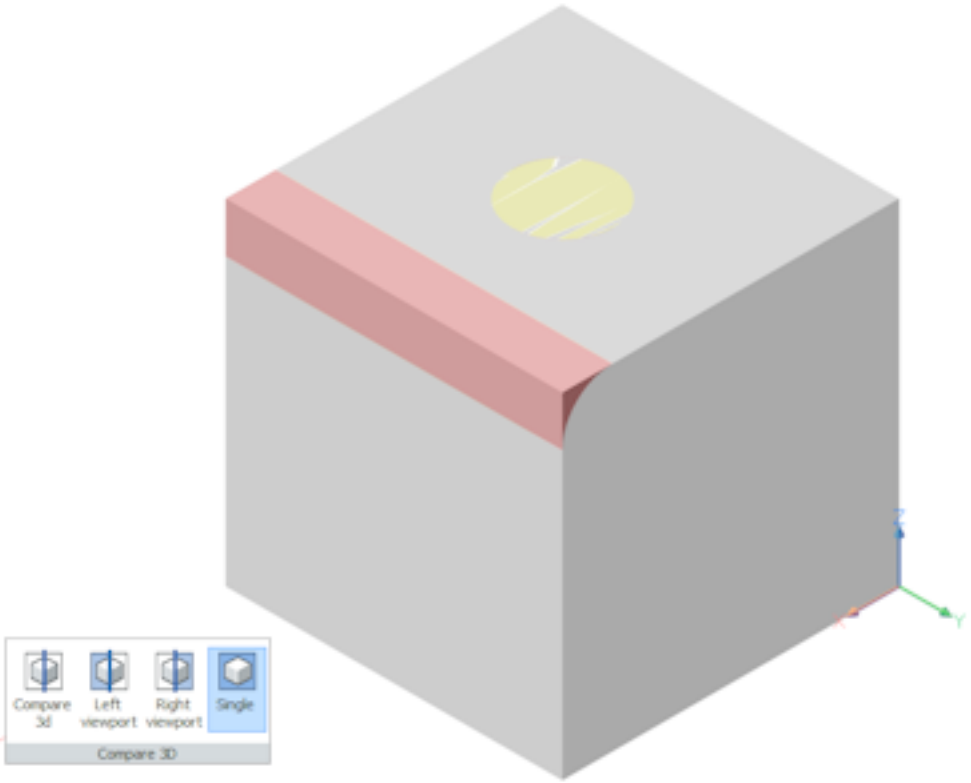


Figure 3.93



3.3 Assembly Design - Workflows

3.3.1 How to work with standard parts

https://www.youtube-nocookie.com/embed/lw4tjfFo6cg

Using standard parts in combination with direct modeling in BricsCAD BricsCAD comes with a rich library of more than 30,000 standard parts. In this short movie we will show you how these parts can be used. To access the library, select the corresponding tab in the Mechanical Browser (3.2.2) window. Here you can search for a specific part. Select the standard part. Then, you can see the preview and set the size and length of the part you want to insert. Insert the part by dragging it to a desired position using snapping and dynamic dimensions.

> You can create complex models very easily and rapidly. Use a rich library of standard parts in combination with direct modeling technology.

Back to Tutorial Overview

3.3.2 Introduction to standard parts

https://www.youtube-nocookie.com/embed/CptV_DLk2y8

If you are looking for an affordable 3D mechanical design software with a rich set of standard parts included, then BricsCAD is your right choice.

To access the library of standard parts, open the Mechanical Browser (3.2.2) panel. Then click on the “Library” tab. You can choose between 13 different standards, including ANSI, DIN, and ISO. All theparts are grouped into categories. Expand a category to see its content. To insert the selected standard part into your model, just drag it from the library and drop it in the drawing area. You can use dynamic dimensions and snapping to position the part precisely. For many parts you can also specify the length. If you are looking for an affordable 3D mechanical design software with a rich set of standard parts included, then BricsCAD is your right choice. Back to Tutorial Overview

3.3.3 Solve inverse kinematic problems in real time

https://www.youtube-nocookie.com/embed/zWj3mZ3bkvI

Kinematic Analysis

If you move the car door, you see how the robot follows it's movement. What you are seeing here is called inverse kinematics of a mechanism. This is a typical inverse problem: we know the result – how our door should be moved, but we don't know how to obtain this result – how the robot should move its platform, shoulder, arm, hand and head to reach the desired position for the door. BricsCAD is able to solve this inverse kinematic problem in real time. Our robot has five degrees of freedom and all of them are used to find the position for the door.

The constraints force other components to rotate correspondingly.

(\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageMain-1024x768.jpg}) Back to Tutorial Overview

Figure 3.94



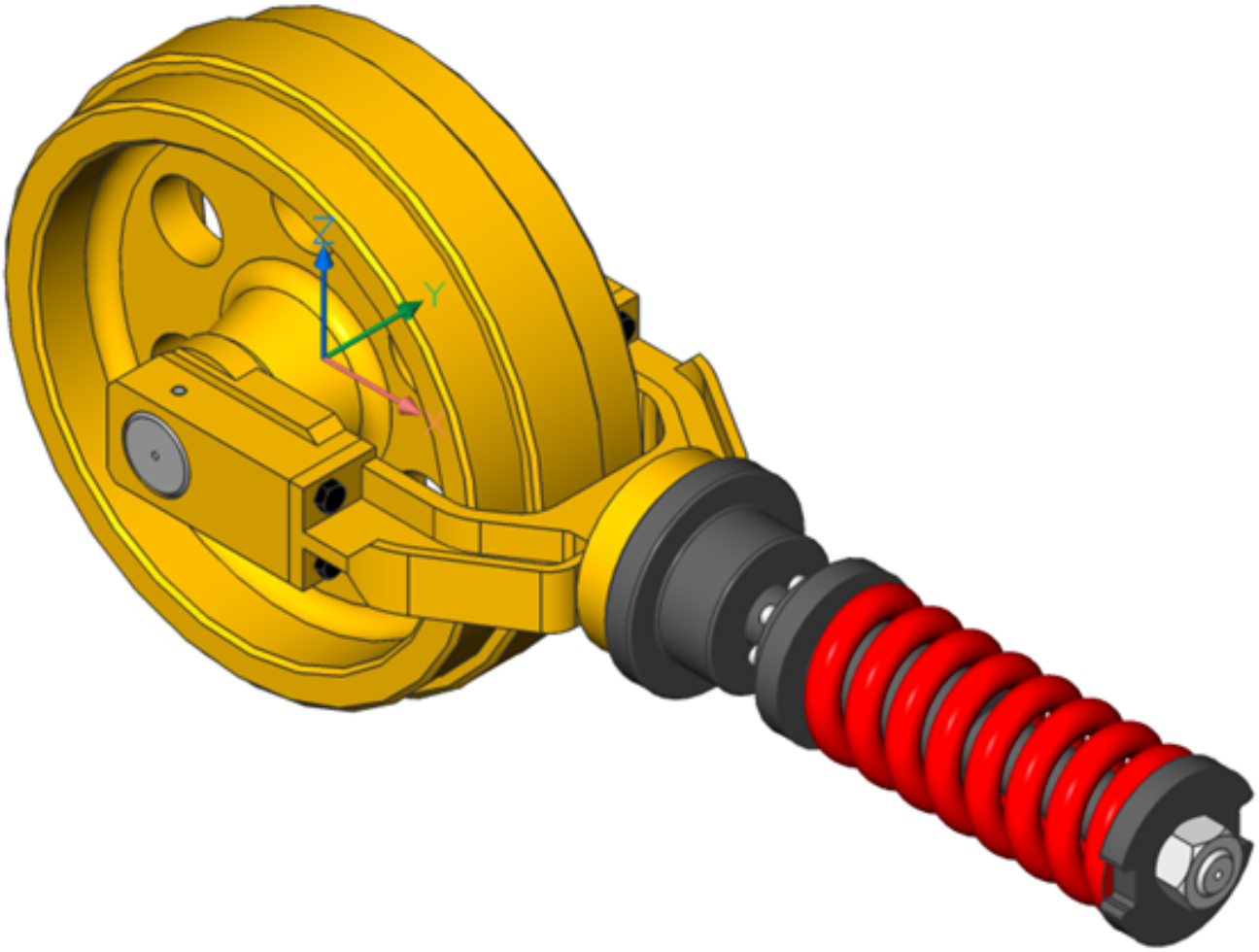
3.3.4 Using mass properties in BricsCAD

https://www.youtube-nocookie.com/embed/2n7tlzIbMMw With BricsCAD you can create assemblies of parts with different densities. In this video we'll show you how to design a self-balancing wine bottle holder by using assembly mass properties calculations and 3D constraints. Alternatively, we can simply use Mass Properties to output information on our selection to the text screen. Here you see the combined mass and volume of the selected objects, along with the center of mass, and moments of inertia. You are even given the option to align your UCS with the equilibrium point. To find the equilibrium point for the self-balancing wine holder, simply draw a line from the center of the holder to the center of mass of the assembly. This line represents the normal direction of force caused by gravity relative to the top of our table. To orientate our wine holder to its balance point, we simply have to apply a perpendicular constraint.

A self-balancing wine bottle holder created with BricsCAD Platinum, using assembly mass properties calculations and 3D constraints.

Back to Tutorial Overview

Figure 3.95



3.3.5 Exploded view of assembly

1. You need a**BricsCAD**Platinumand **Mechanical**licenses for this tutorial.
2. Open the Front_Idler_Assembly.dwgfile. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/3

3. Click the **Explode (6.8.6)** ( ) tool button on the **Tools** panel of the **Assembly** tab on the Ribbon (1.13.6).
4. Select the **Settings, Bottom** option to create an exploded view to the bottom level of all components.
5. Click the **Linear** options on the **BmExplode** Prompt menu to create an exploded representation of assemblies in a given direction.
6. Select **X-axis** on **BmExplode** Prompt menu as thedirection.
7. Enter **3000** in dynamic dimension as the distance. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattach

8. To edit, click the **Edit** option on the **BmExplode** Prompt menu to open the exploded representation.
9. Select the 4 **DIN 933** bolts. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039320054

10. Select**Move (6.12.17)** ( )from the **Model** section of theQuad (2.2.3),move the bolts **400** on the **X-axis**.
11. Select the **Linear_Bottom_1** node in theMechanical Browser (3.2.2).
12. Click **Add new step** from the context menu.
13. Select the 2 pins. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040161213/mceclip

14. Select**Move (6.12.17)** ( )from the **Model** section of theQuad (2.2.3). Move all Pins**400** on the **Z-axis**.
15. Select the **Linear_Bottom_1** node in the Mechanical Browser (3.2.2).
16. Click **Add new step** from the context menu.
17. Select the **wheel** and related parts with Green Crossing Window Selection (1.23.6). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039320114/mceclip4.png})

Figure 3.96

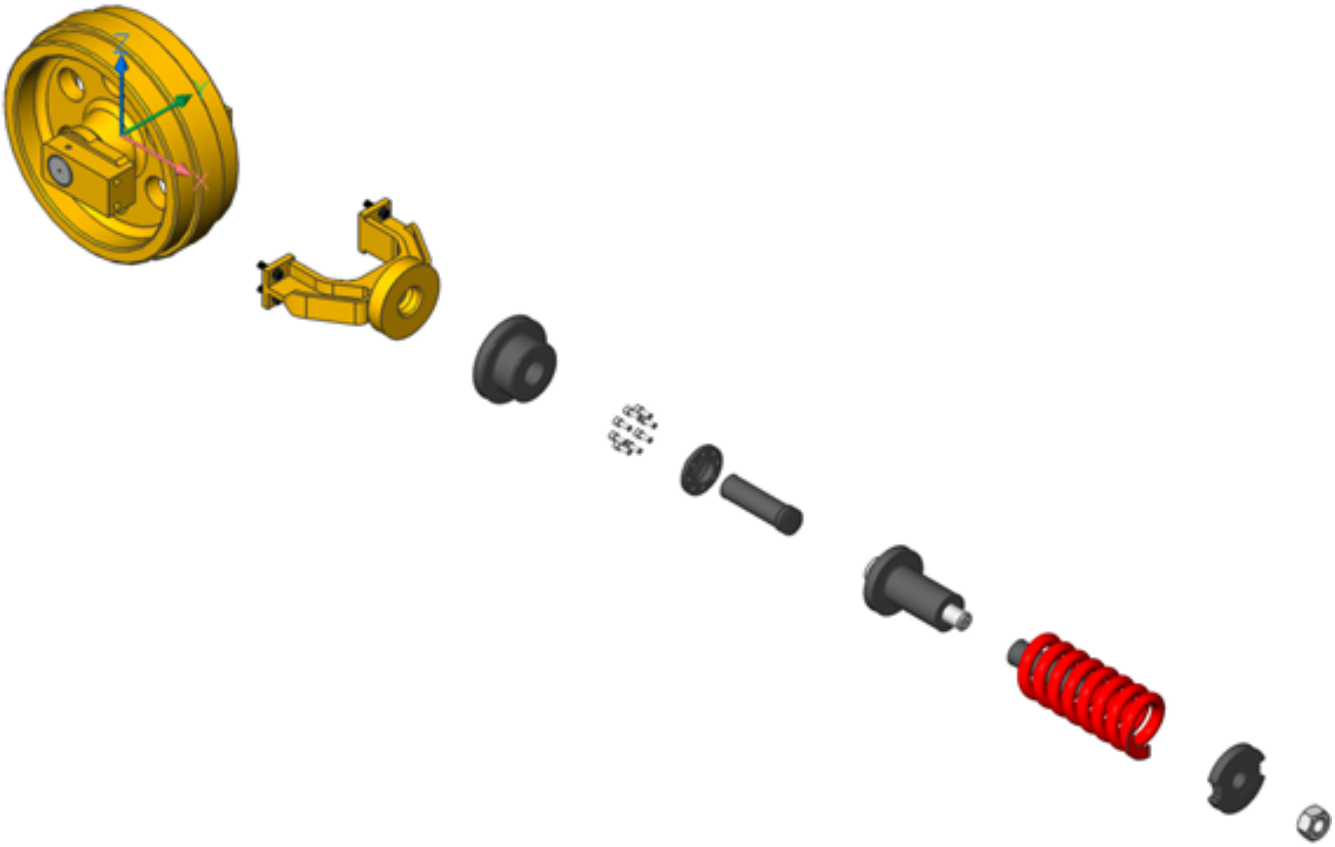


Figure 3.97

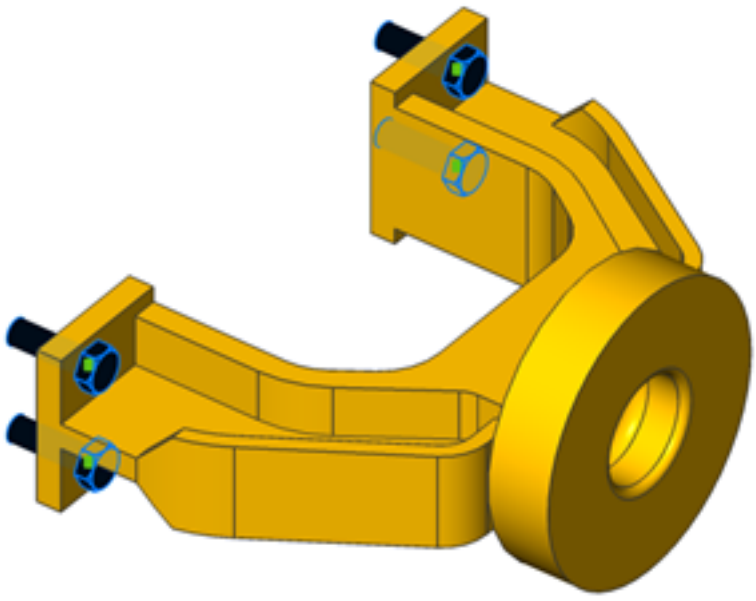


Figure 3.98

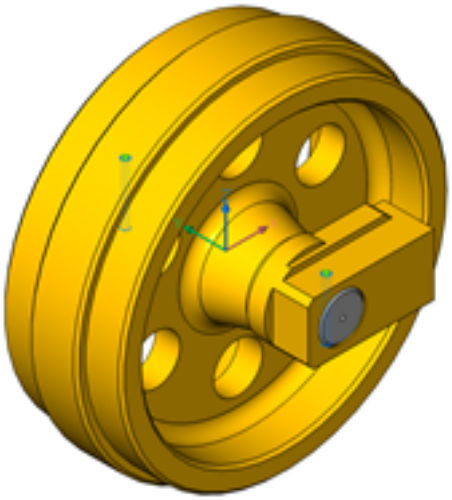


Figure 3.99

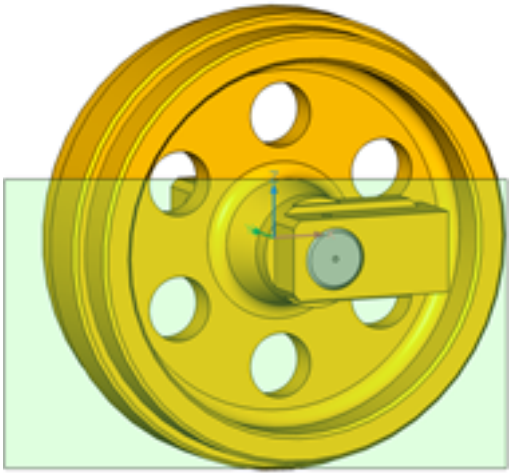


Figure 3.100

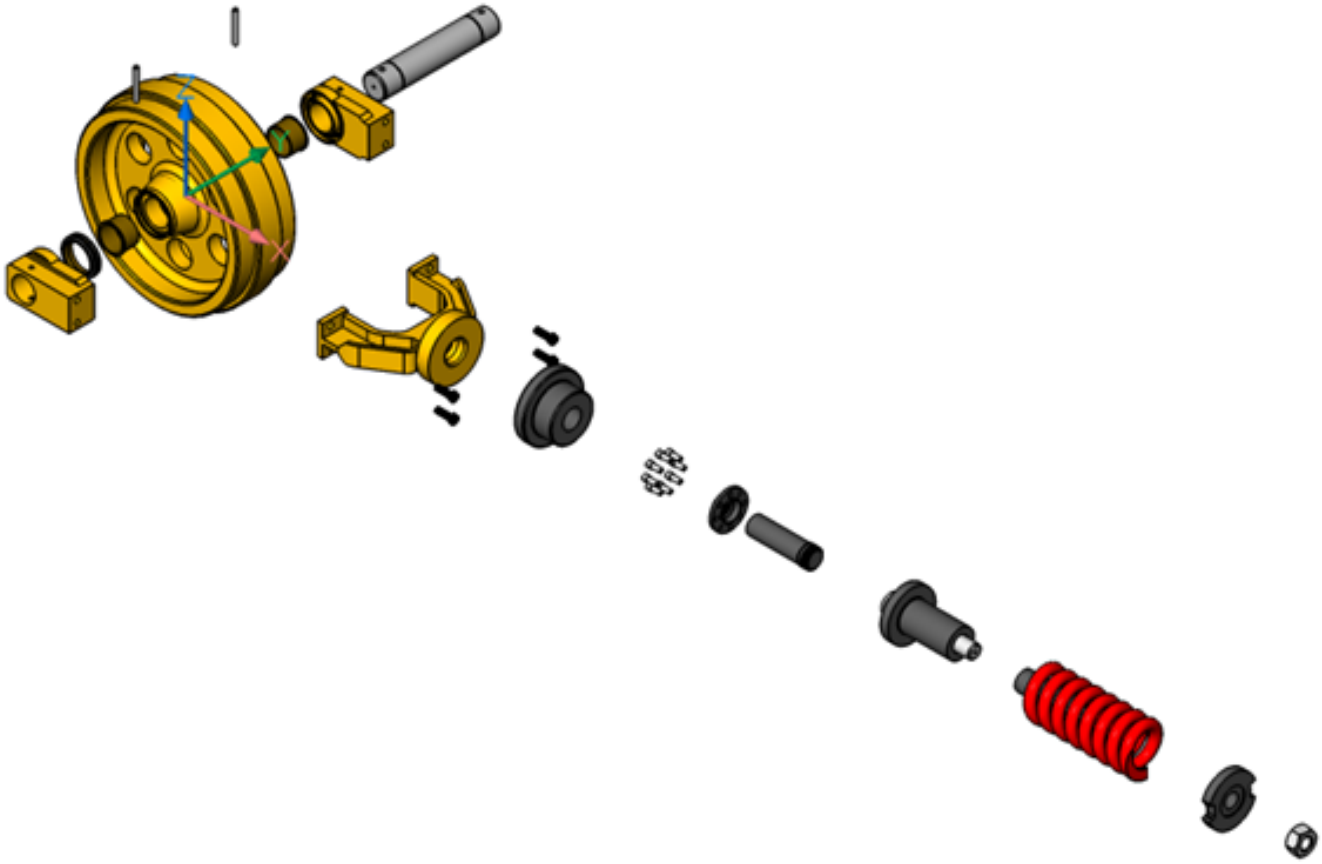


Figure 3.101

Exploded view step	
Name	Step 20
Duration, ms	1000
Duration type	Absolute
Duration value	500

18. Click the **Explode Move (6.8.7)** ( ) tool button on the **Tools** panel of the **Assembly** tab on the Ribbon (1.13.6).
19. Select the **wheel** as the base part.
20. Select **Y-axis** on **BmExplodeMove** Prompt menu as thedirection.
21. Enter **1000** with dynamic dimension as a distance. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattac

22. Select **Steps 15-20** of the **Linear_Bottom_1** node on the Mechanical Browser (3.2.2).
23. Click **Merge with previous** from the context menu.
24. Select **Step 20** of the **Linear_Bottom_1** node on the Mechanical Browser (3.2.2).
25. Change **Duration type** to **Absolute** and **Duration value** to **500** on the bottom part of the Mechanical Browser (3.2.2). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040161533/mceclip6.png})

26. Select the **Linear_Bottom_1** node in the Mechanical Browser (3.2.2).
27. Click **Animate** from the context menu. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360

28. Click the **BClose and Save (6.6.4)** ( ) tool button on the **Block Edit** toolbar.

Figure 3.102

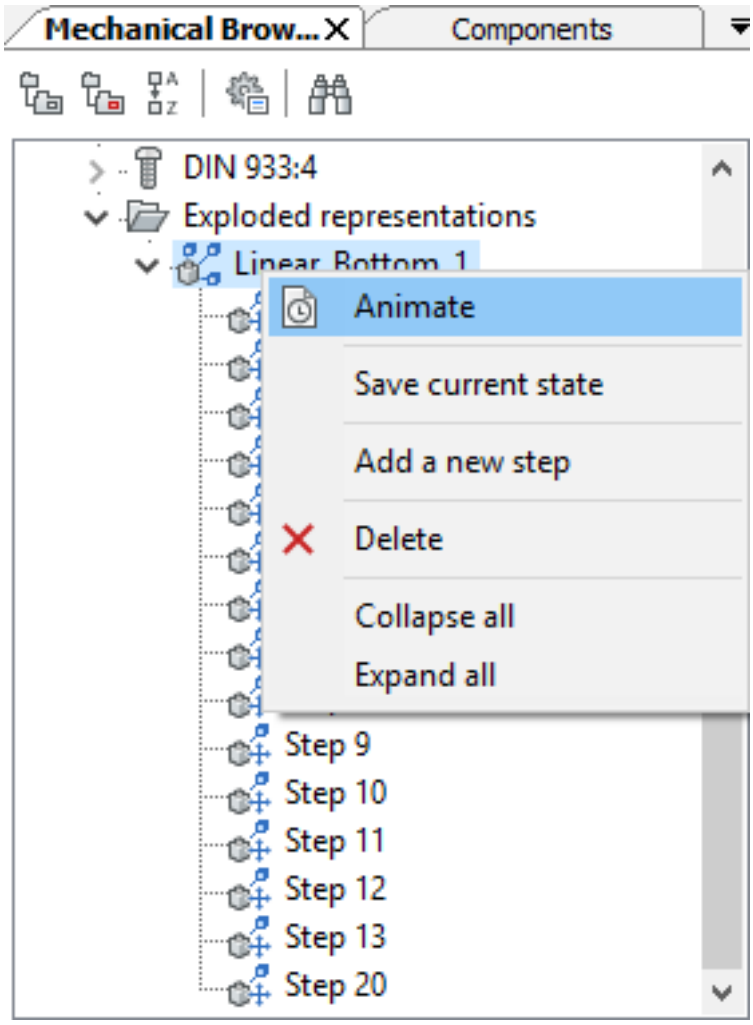
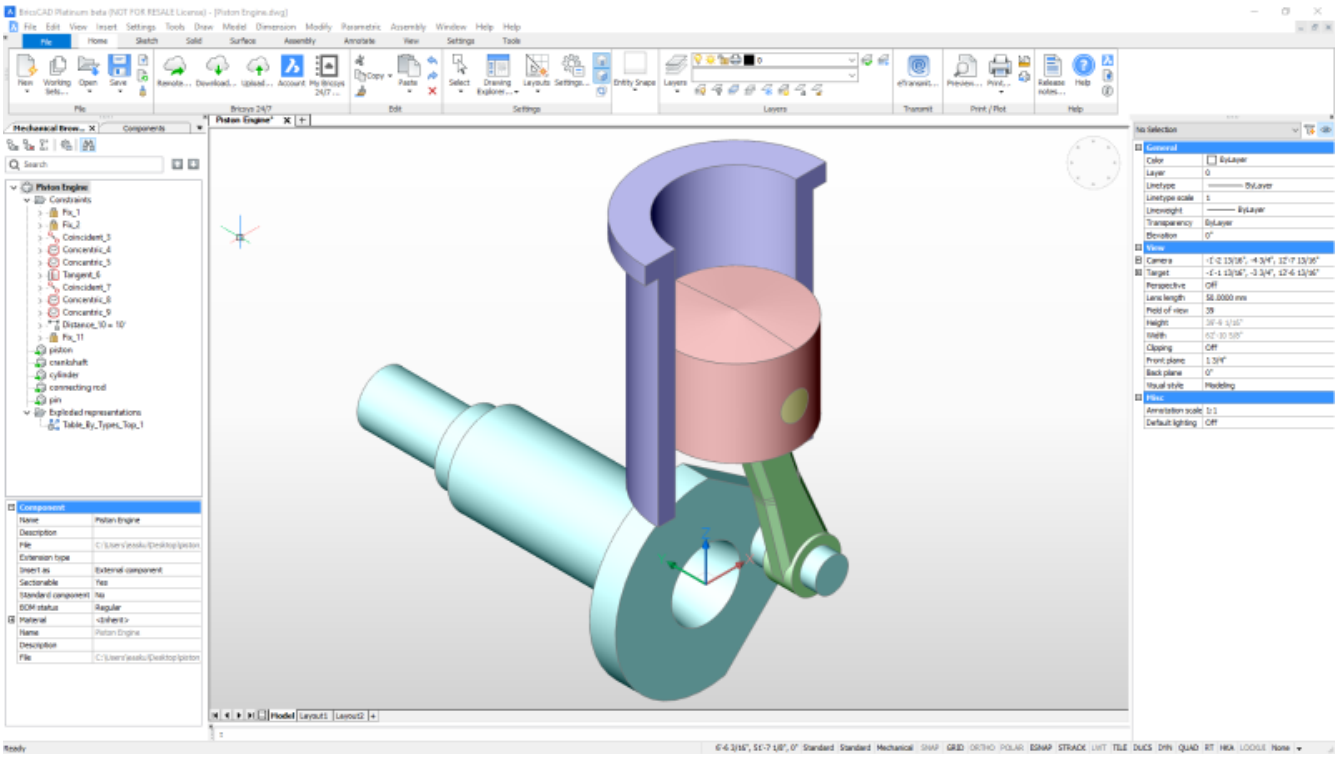

Mechanical Brow...
Components
DIN 933:4
Exploded representations
Animate
Save current state
Add a new step
Delete
Collapse all
Expand all
Step 9
Step 10
Step 11
Step 12
Step 13
Step 20

Figure 3.103



3.3.6 Assembly design and kinematic analysis tutorial

The source files for this tutorial are installed in the **[BricsCAD Installation folder]\Samples\Mechanical\piston** folder. Assembly design is a typical application where constraints are commonly used. BricsCAD Platinum allows users to assemble mechanisms and run a kinematic analysis. This tutorial shows you how to start working with assemblies in BricsCAD. You will learn how to assemble the different parts in a 3D model, which lets you control the final result. There are two approaches in MCAD (Mechanical Computer-Aided Design) to build an assembly: top-down and bottom-up. In the top-down approach, you start with an empty assembly and the geometry of each component is then created, one by one, in the assembly. In the bottom-up approach, each component is first created as a single entity. All components are then inserted into the assembly. The position of each component is controlled through 3D constraints. We will now assemble a simplified - in comparison with a real-world engine - a model of a piston engine. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017981113/mceclip39.png})

Step 1: Preparing for the exercise

1. You need a **BricsCAD Platinum** license for this tutorial.
2. Make sure the **Assembly** and **3D Constraints** toolbars are open. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc
(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017184874/mceclip18.png})
To open a toolbar: move the cursor over an open toolbar, then right click and select the required toolbars from the toolbar list. Toolbars that are already open are marked in the list.
3. Make sure that the Quad (2.2.3) cursor menu is active. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/h
Click the **Quad** field in the Status bar (1.12.2) if necessary.
4. Make sure that the **Exclude entities in Xrefs** option of the PREVIEWFILTER (1.13.11) system variable is switched off. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10Drawin

5. Open the Mechanical Browser (3.2.2) dialog box. Do one of the following:

Figure 3.104



Figure 3.105



Figure 3.106

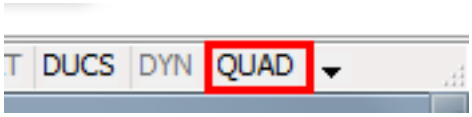


Figure 3.107

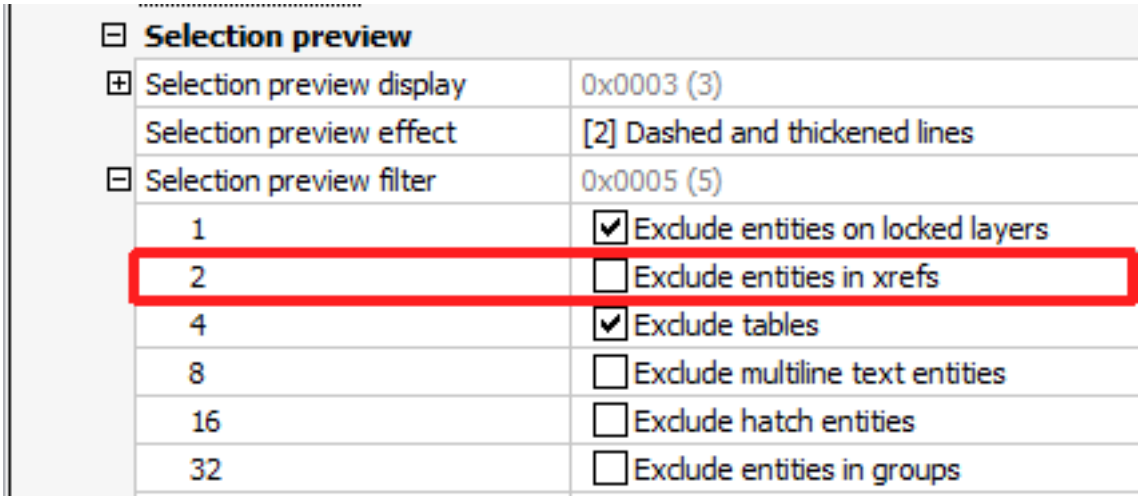


Figure 3.108

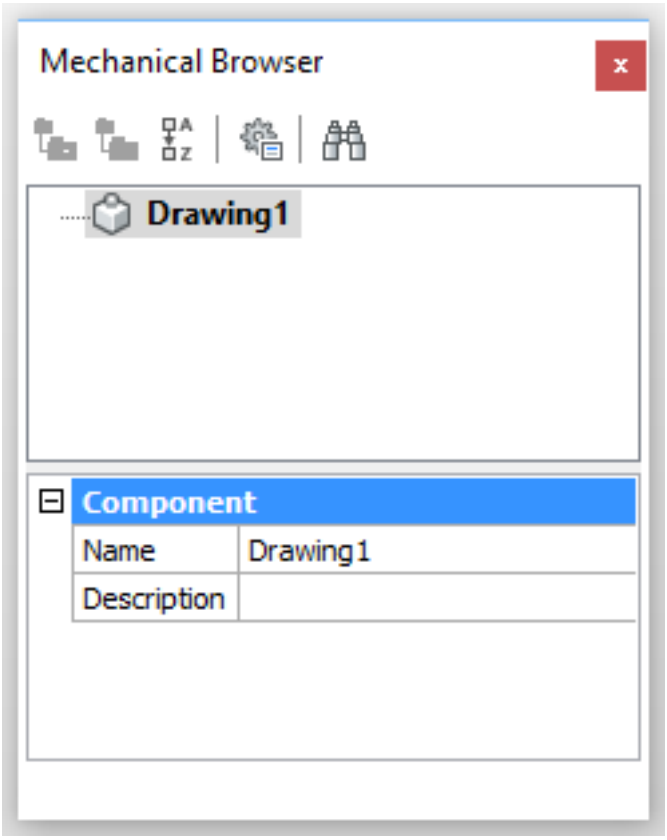


Figure 3.109



- Click the **Mechanical Browser** tool button ( ) on the **Mechanical | Tools** panel in the ribbon.
- Click the **Mechanical Browser** tool button ( ) on the **Mechanical** toolbar.
- Choose **Mechanical Browser** in the **Mechanical** menu.
- Type MechanicalBrowserOpen (6.8.29) in the command bar.

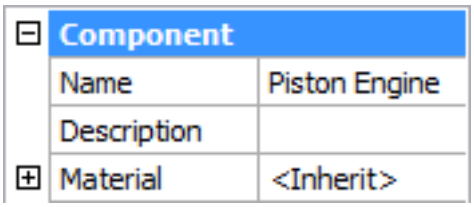
(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017185134/mceclip21.png})

6. Make sure **Selection Modes** = 3 Do one of the following:
   - Type **selectionmodes** in the command bar, then type **3**.
   - Set the **Selection Modes** toolbar as indicated below. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricsca

Step 2: Creating the assembly drawing

1. Click the **New Component** tool button ( ) on the **Mechanical | Create** panel in the ribbon. A new drawing of the name **Component_1.dwg** is created and **Component_1** is added in the Mechanical Browser (3.2.2).
2. In the bottom part of the Mechanical Browser (3.2.2), click the **Name** field and rename the main component: **Piston Engine**. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10Draw The drawing name automatically matches the main component name.
3. Save the drawing.

Figure 3.110

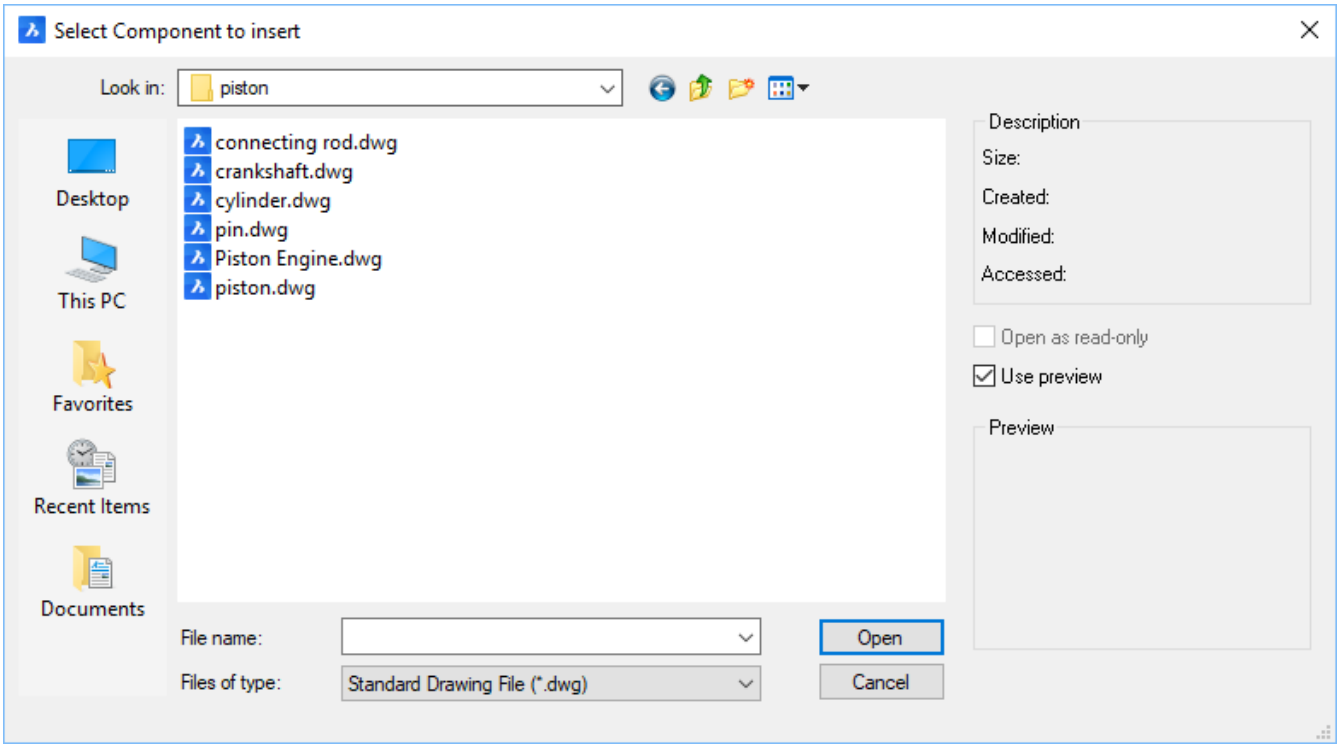


Step 3: Adding the components

1. Click the **Insert Component** tool button ( ) on the **Mechanical | Modify** panel in the ribbon. The **Insert Mechanical Component** dialog displays: Open the **[BricsCAD Installation folder]\Samples\Mechanical\piston** folder. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017186494/mceclip25.png})

2. Select the crankshaft.dwg, then double click or click the **Open** button. The component is attached to the cursor with its origin point. Dynamic dimensions display distance from the origin of the current coordinate system.
3. We will consider the crankshaft as the anchor component of the assembly and therefore insert it at the origin (0,0,0). Please make sure the insertion point is exactly at 0,0,0, otherwise the kinematic analysis in **step 5** might fail. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017258674/mceclip30.png

Figure 3.111



4. Use the Look From (1.16.8) widget to adjust the view orientation.
5. Repeat the previous steps to insert the other components: cylinder (1), piston (2), pin (3) and connecting rod (4). Avoid visual intersection of the components. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattach

Figure 3.112

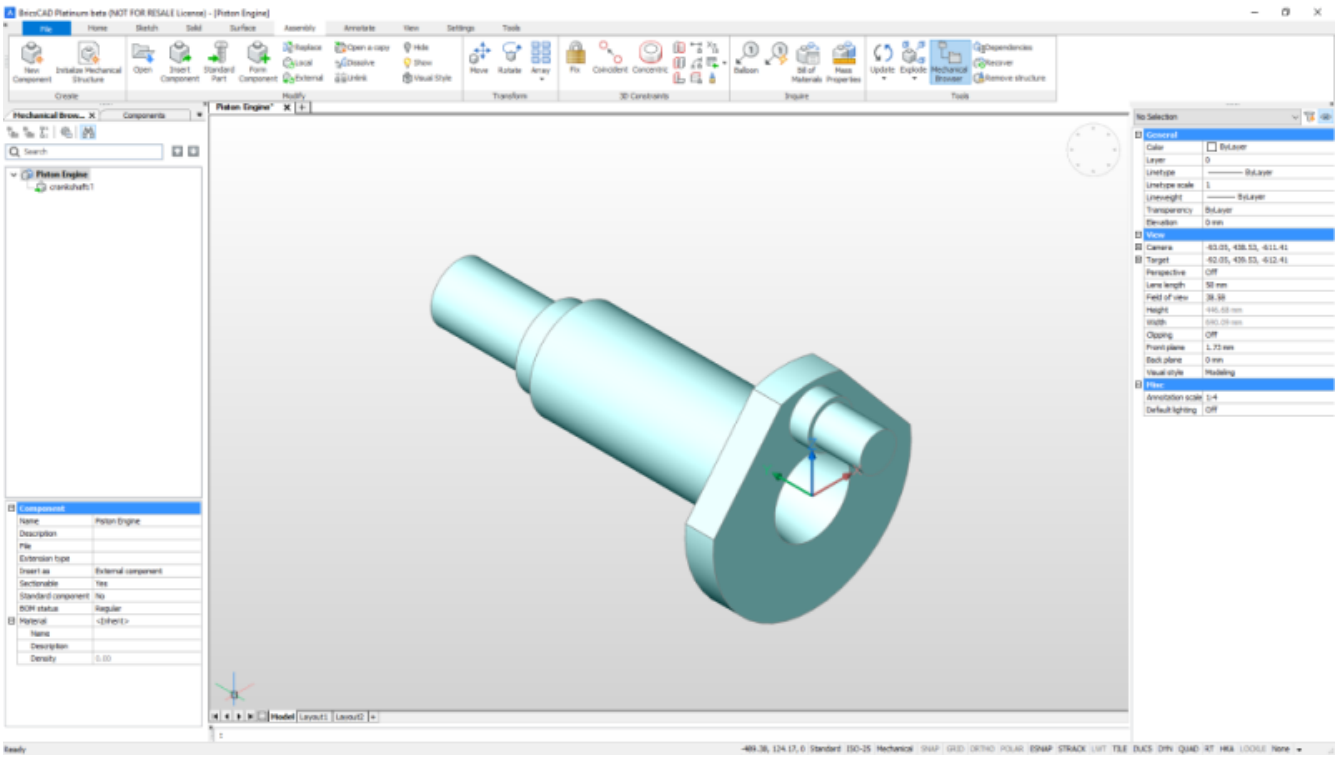
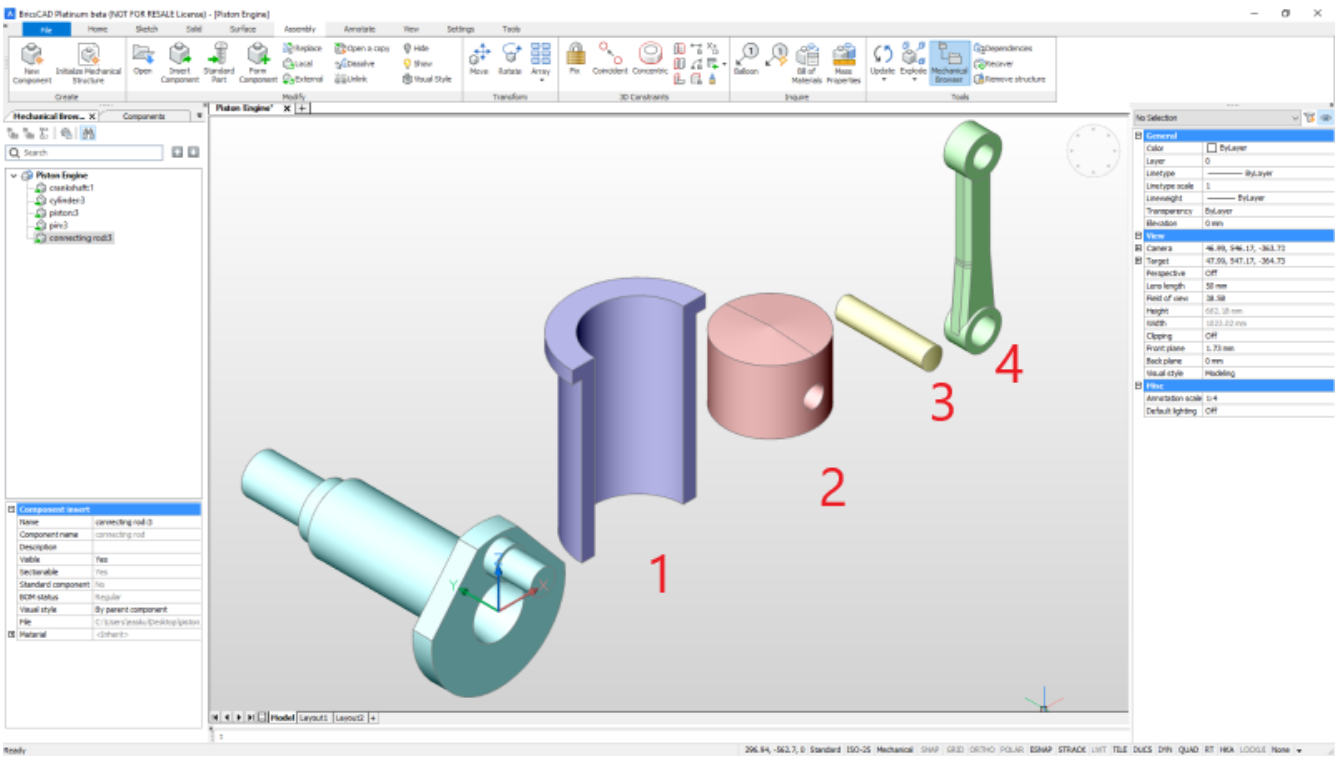


Figure 3.113



Step 4: Positioning the components

1. Click the **Fix** tool button ( ) on the **Solid | 3D Constraints** panel in the ribbon. You are prompted: **Select an edge, face or 3D solid:**
2. Move the cursor over the largest cylindrical face of the crankshaft. Click when the face highlights.
3. Repeat the DmFix3d (6.12.15) command. Move the cursor over the currently obscured circular face of the crankshaft. Hit the TAB key until the face highlights and click when the face highlights. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017258894/mceclip33.png}) The position of the selected faces of the crankshaft is now locked, as indicated in the **Constraints** tree in the Mechanical Browser (3.2.2). (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEnti
Clicking a constraint highlights the solid, face or edge it applies to.
4. Place the connecting rod near the crankshaft.
5. Click the **Coincident** tool button ( ) on the **Solid | 3D Constraints** panel in the ribbon. You are prompted:**Select a pair of subentities or [Autoconstrain] <Autoconstrain>:**
6. Move the cursor over two planar faces of the crankshaft (1) and the connecting rod (2) as indicated in the image below. Click when the correct face highlights. Hit the TAB key to select the obscured face of the connection rod and click when the face highlights. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments
7. Click the **Concentric** tool button ( ) on the **Solid | 3D Constraints** panel in the ribbon. You are prompted:**Select the first entity:**
8. Move the cursor over two cylindrical faces of the crankshaft (1) and the connecting rod (2) as indicated in the image below. Click when the first face highlights, then select the second face. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017260914/mceclip43.png}) The connecting rod is automatically moved to the right position according to our design intent. (\vref {fig:C:/Dev/Task-2448/images/helpbrics
9. Click the **Concentric** tool button ( ) on the **Solid | 3D Constraints** panel in the ribbon. You are prompted: **Select the first entity:**
10. Move the cursor over two cylindrical faces of the piston (1) and the pin (2) as indicated in the image below. Click when the first face highlights, then select the second face. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc
(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017975533/mceclip12.png})
11. Click the **Tangent** tool button ( ) on the **Solid | 3D Constraints** panel in the ribbon. You are prompted: **Select the first entity:**
12. Move the cursor over two cylindrical faces of the piston (1) and the pin (2) as indicated in the image below. Click when the first face highlights, then select the second face. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc
(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017254794/mceclip7.png})

Figure 3.114

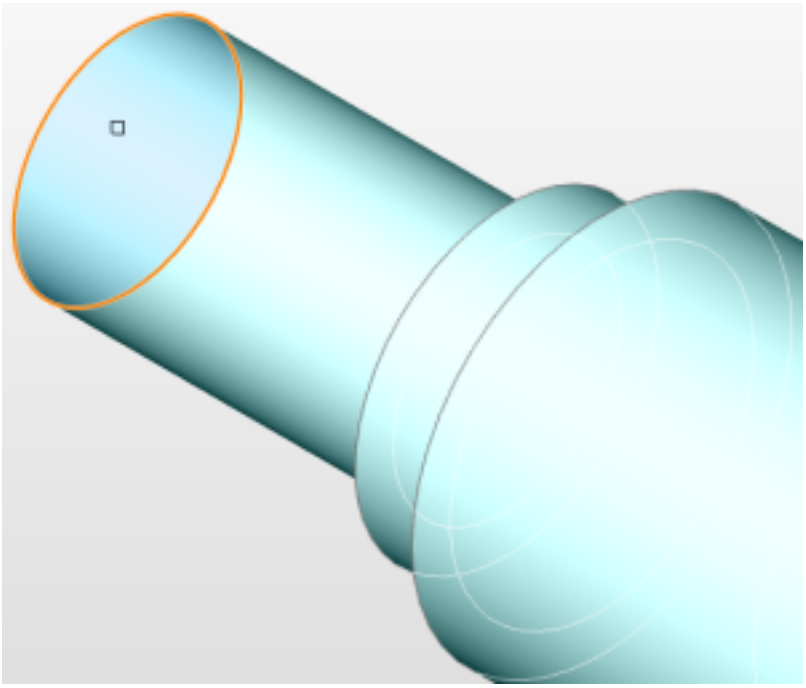


Figure 3.115

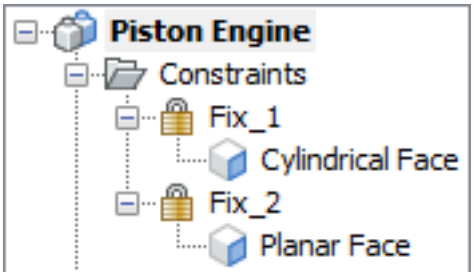


Figure 3.116

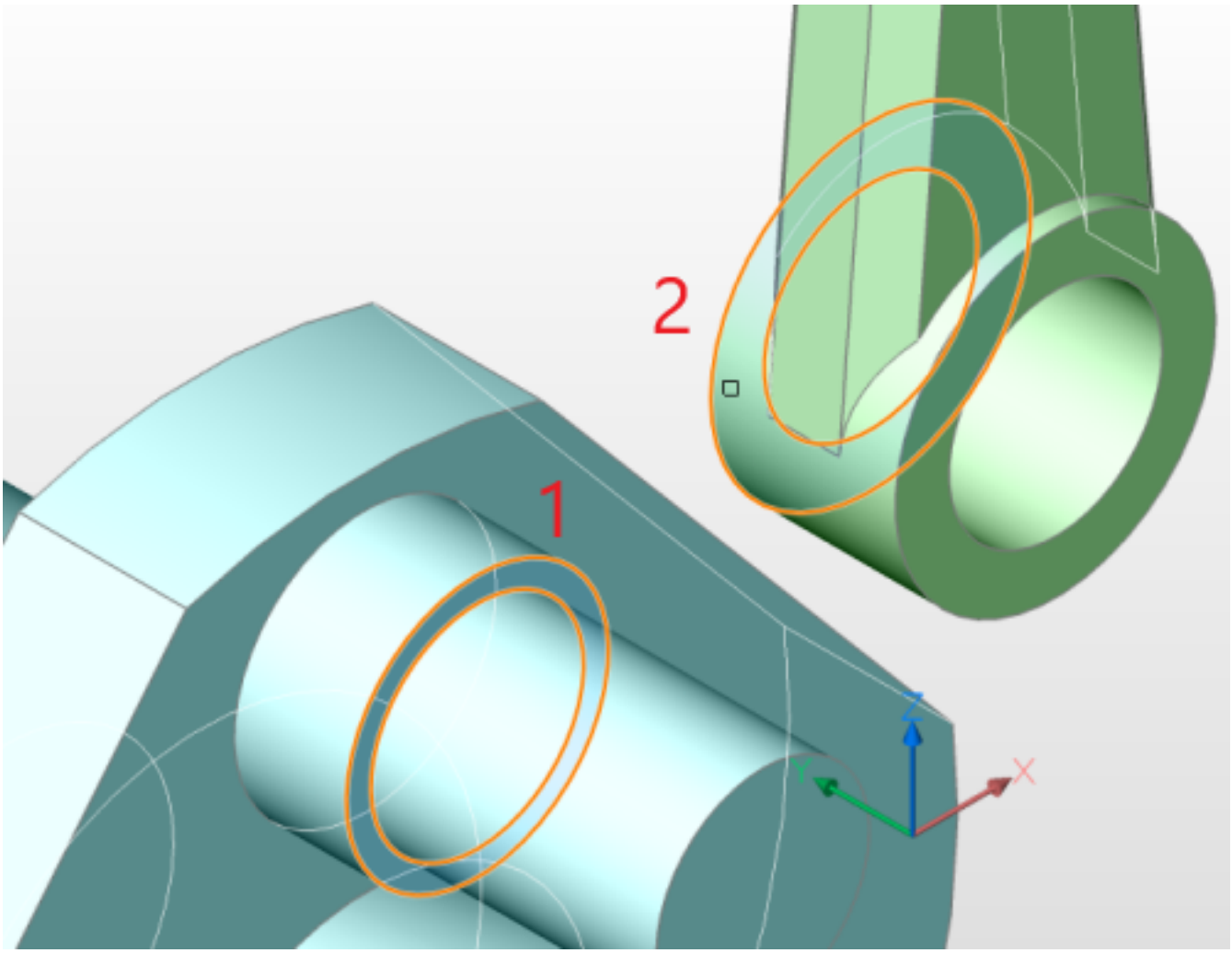


Figure 3.117

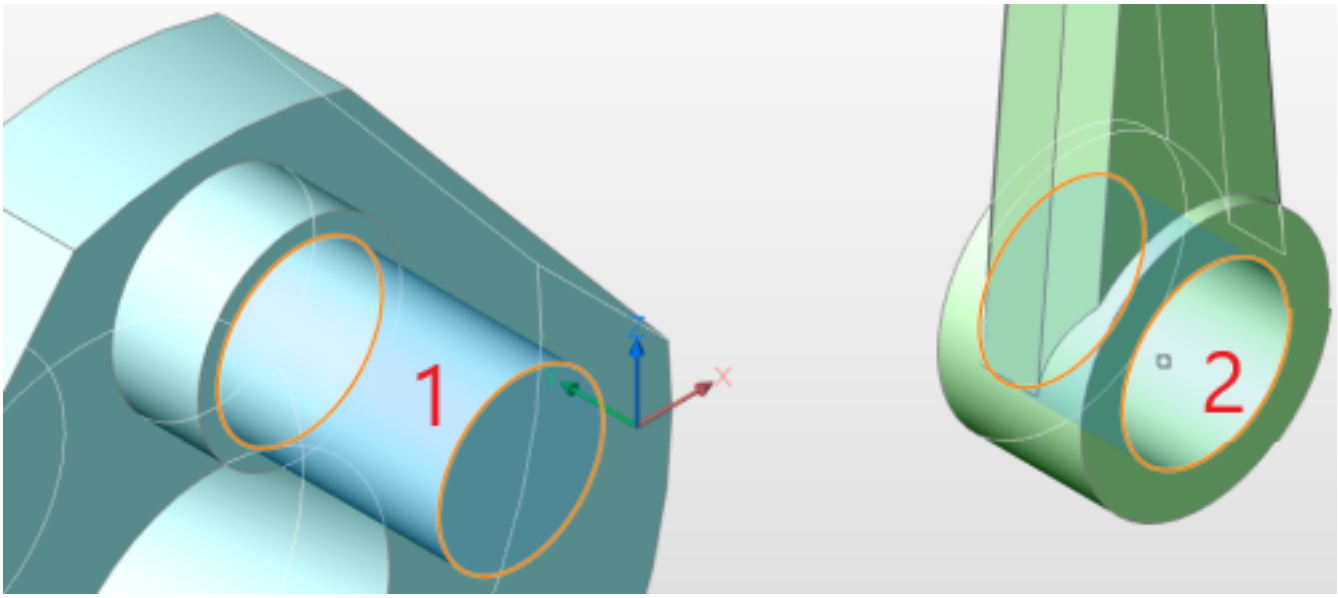


Figure 3.118

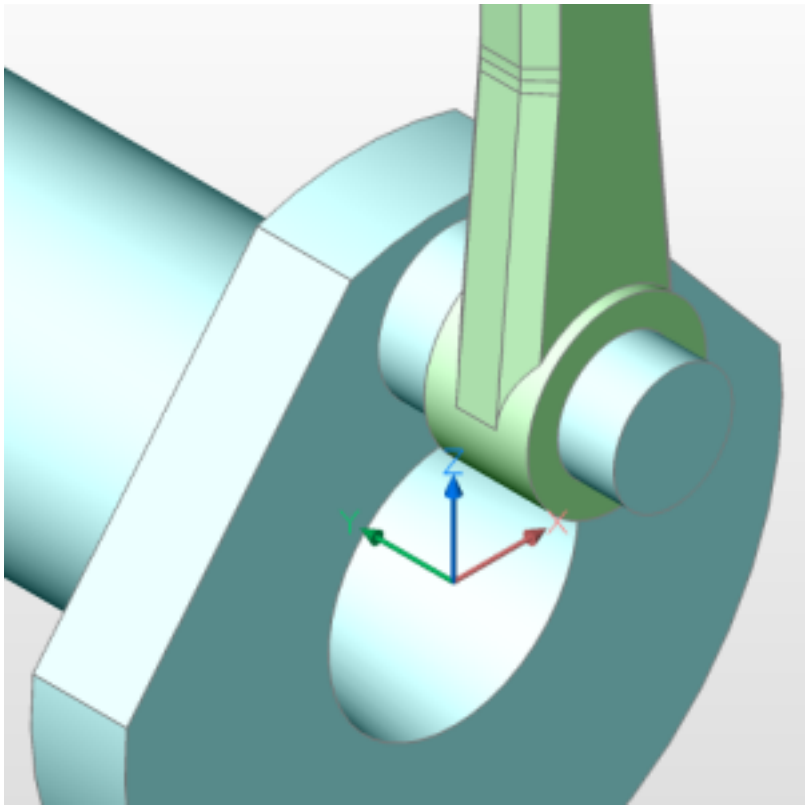


Figure 3.119

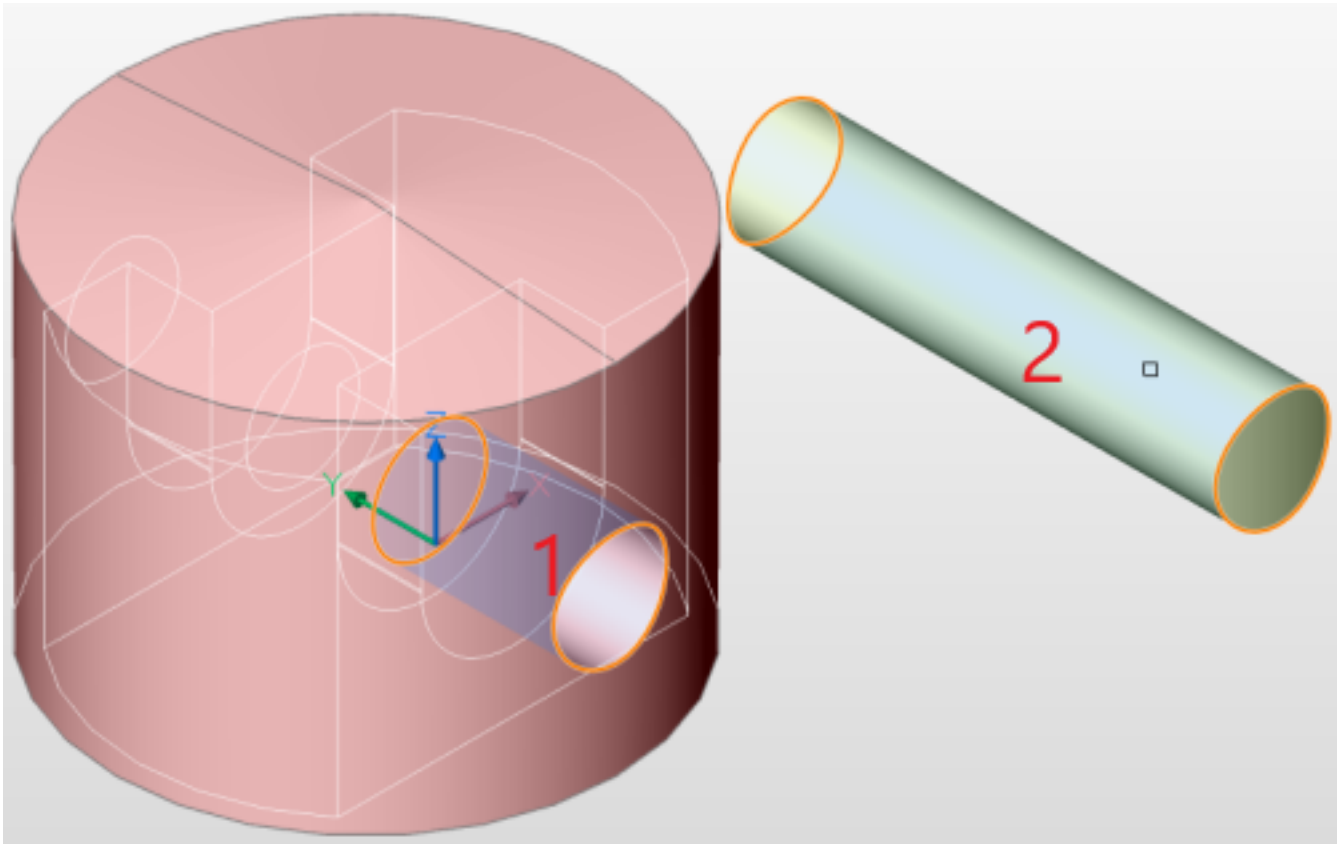


Figure 3.120

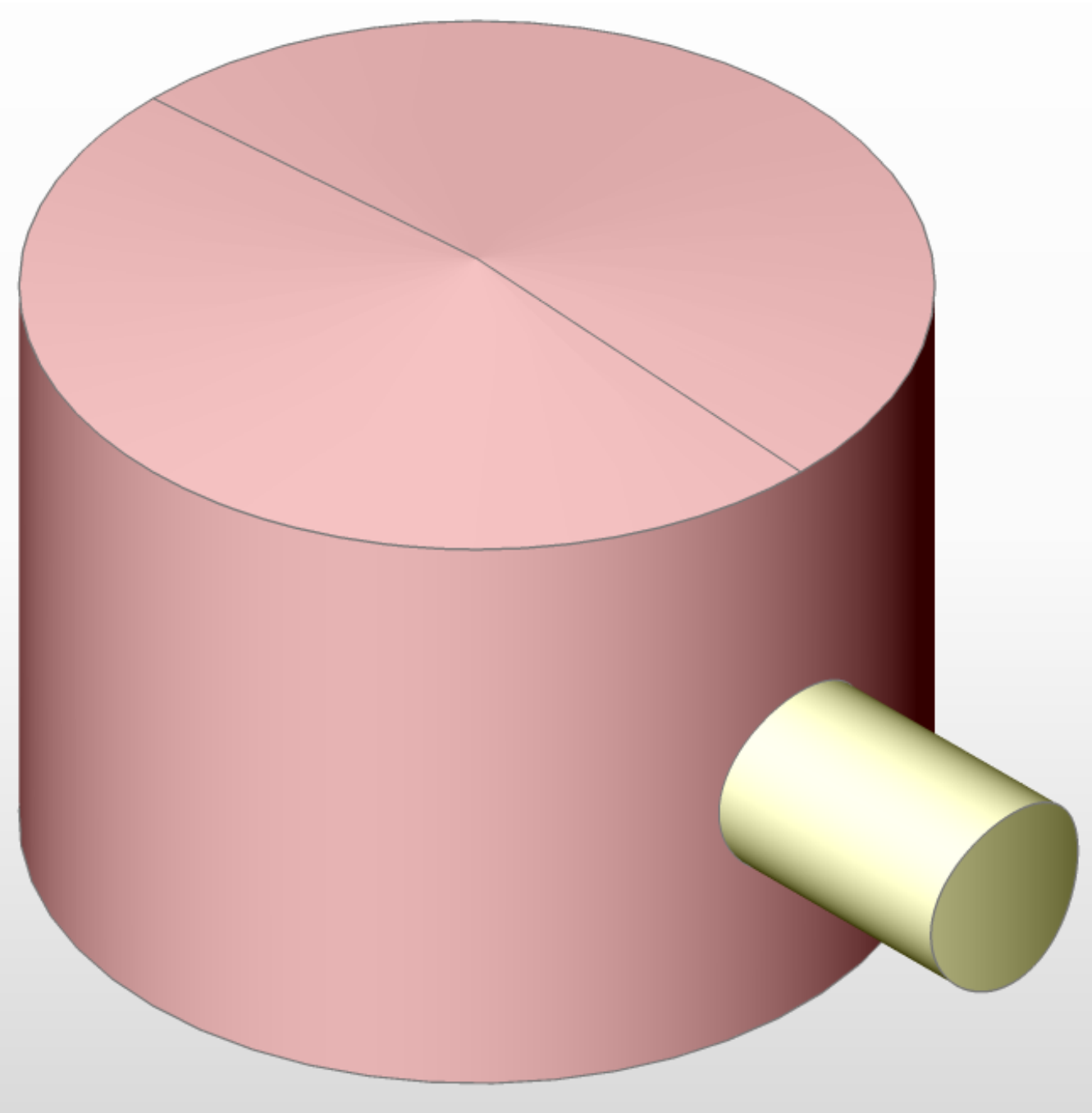
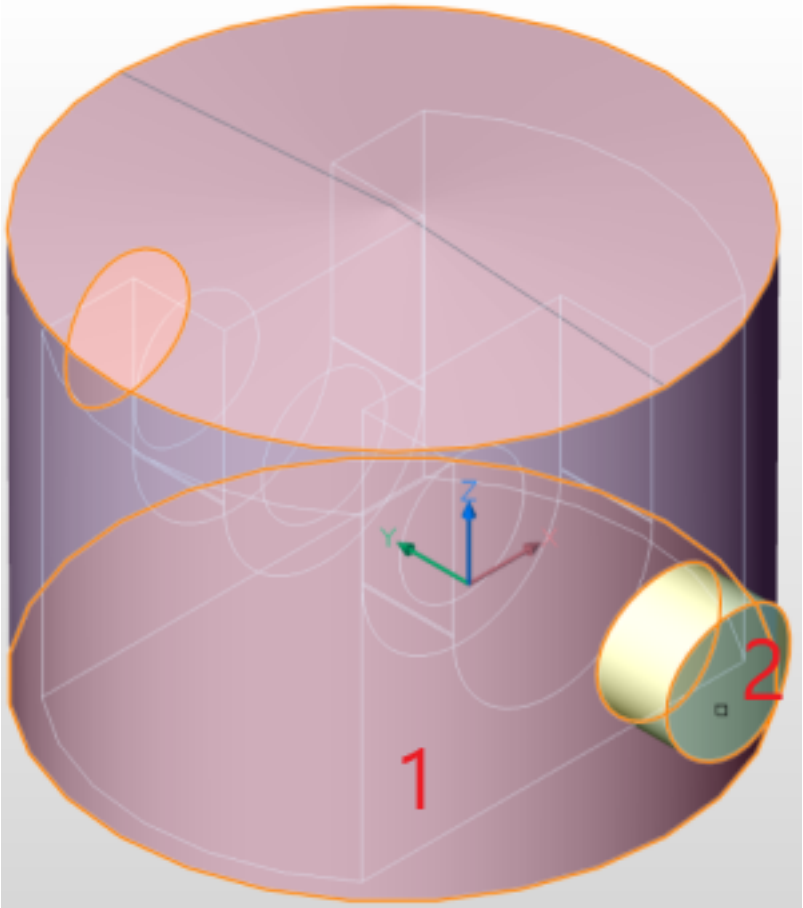


Figure 3.121



13. Click the **Coincident** tool button ( ) on the **Solid | 3D Constraints** panel in the ribbon. You are prompted:**Select a pair of subentities or [Autoconstrain] <Autoconstrain>:**
14. Move the cursor over two planar faces of the connecting rod (1) and the piston (2) as indicated in the image below. Click when the first face highlights, then select the second face. Hit the TAB key to select the interior face of the piston. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017259454/mceclip36.p
15. Click the **Concentric** tool button ( ) on the **Solid | 3D Constraints** panel in the ribbon. You are prompted:**Select the first entity:**
16. Move the cursor over two cylindrical faces of the connecting rod (1) and the pin (2) as indicated in the image below. Click when the first face highlights, then select the second face. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017259534/mceclip37.png}) (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017255494/mceclip17.png})
17. Click the **Concentric** tool button ( ) on the **Solid | 3D Constraints** panel in the ribbon. You are prompted: **Select the first entity:**
18. Move the cursor over two cylindrical faces of the piston (1) and the cylinder (2) as indicated in the image below. Click when the first face highlights, then select the second face. (\vref {fig:C:/Dev/Task-2448/images/helpbricsysco (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017975813/mceclip19.png})
19. Click the **Distance** tool button ( ) on the **Solid | 3D Constraints** panel in the ribbon. You are prompted:**Select the first entity or specify [measurement Mode]:**
20. Move the cursor over the cylindrical faces of the crankshaft (1) and the planar bottom face of the cylinder (2) as indicated in the image below. Click when the first face highlights, then select the second face. Hit the TAB key to select the bottom face of the cylinder. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachment Type 120 in the dynamic distance field.
21. (optional) Edit the **Expression** field of the distance constraint in the Mechanical Browser (3.2.2) panel to adjust the vertical position of the cylinder.

Figure 3.122

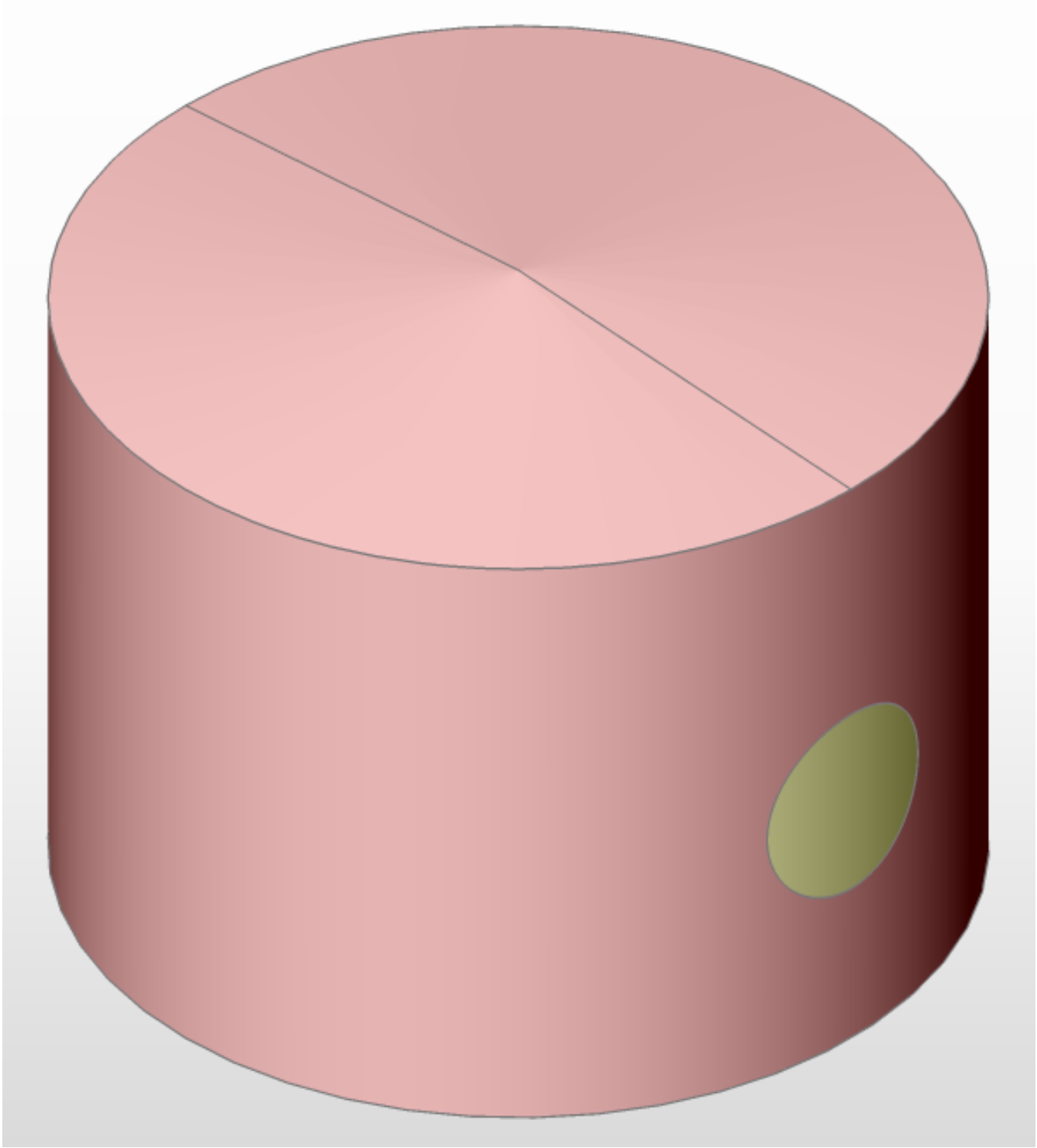


Figure 3.123

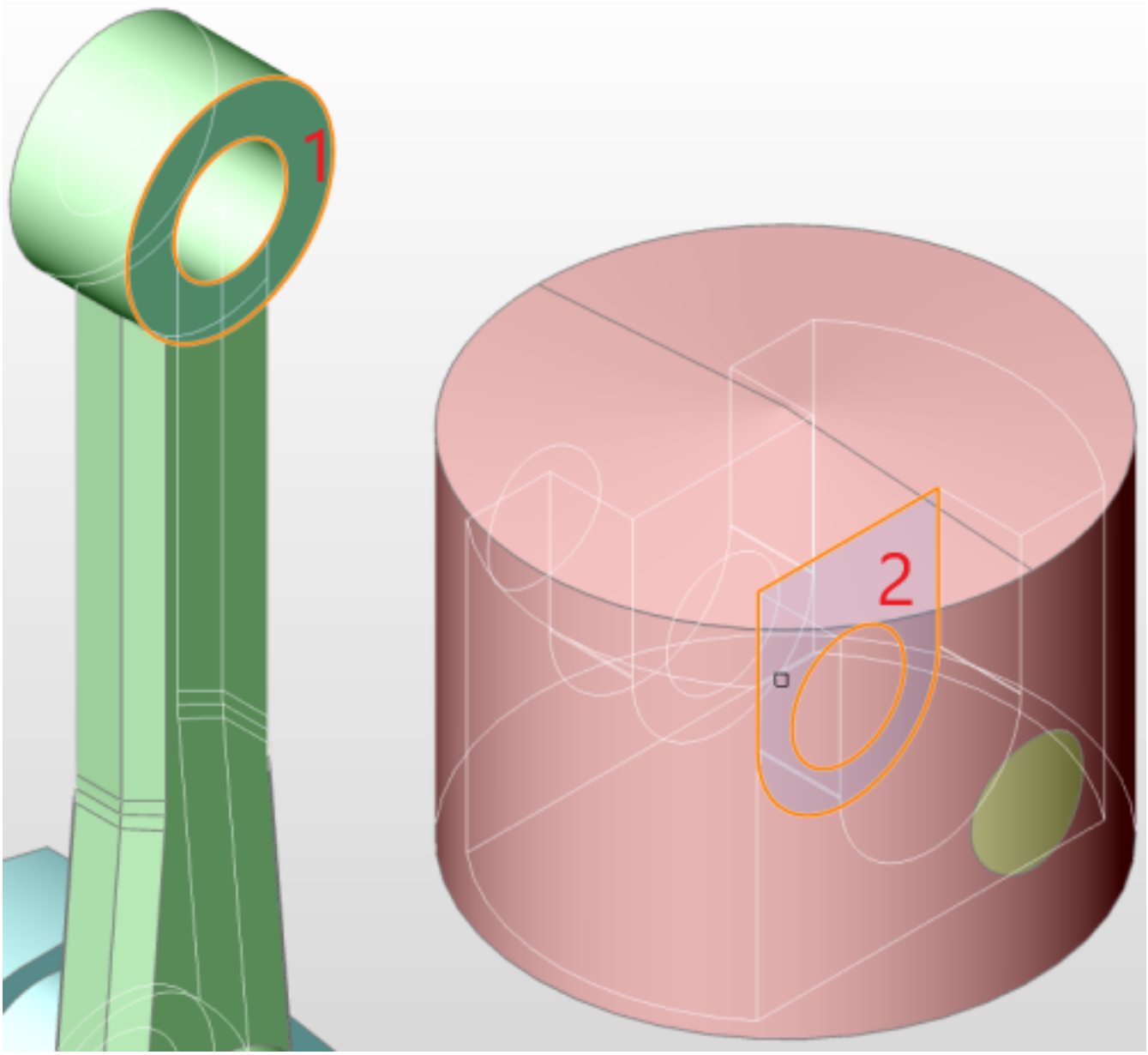


Figure 3.124

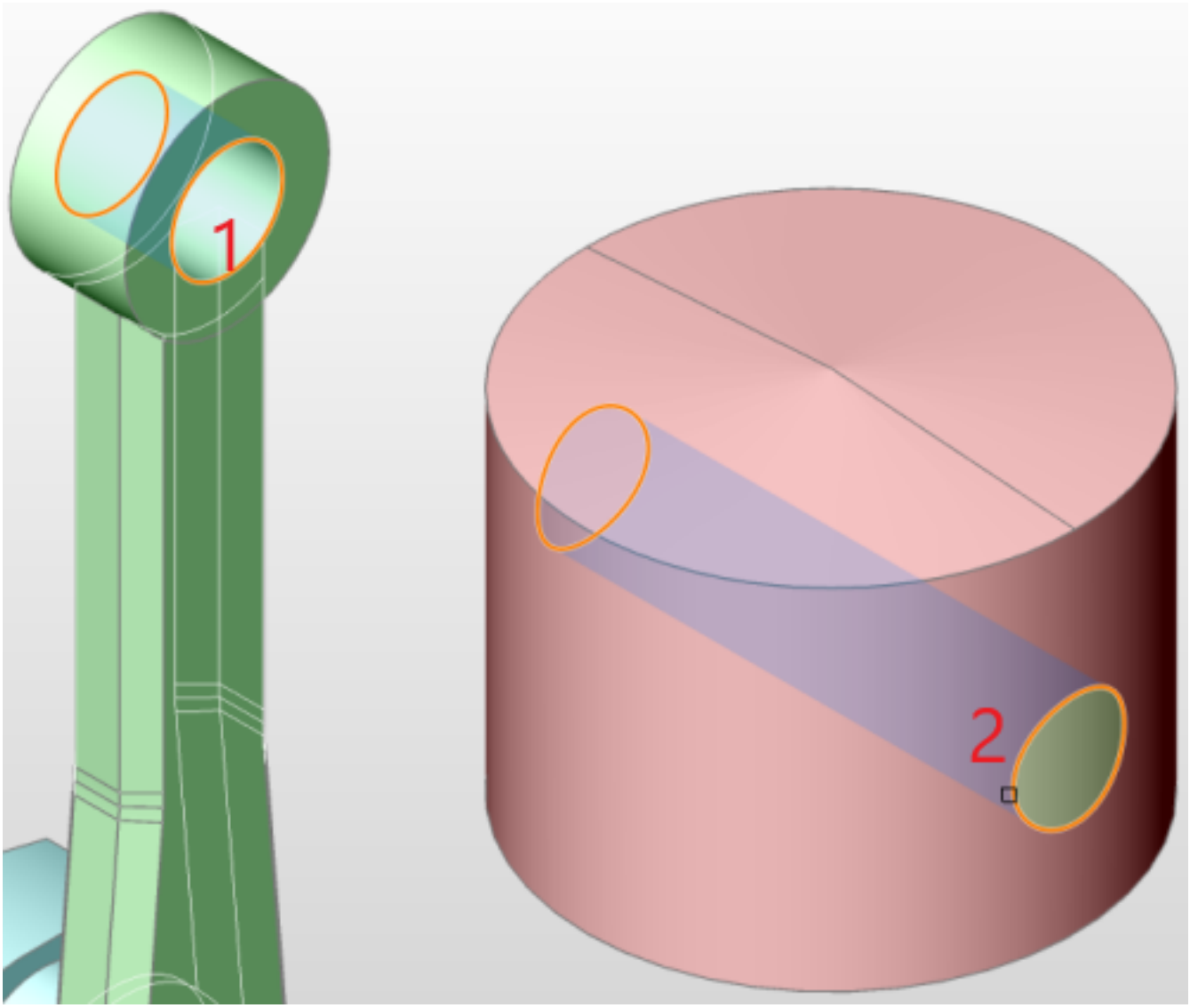
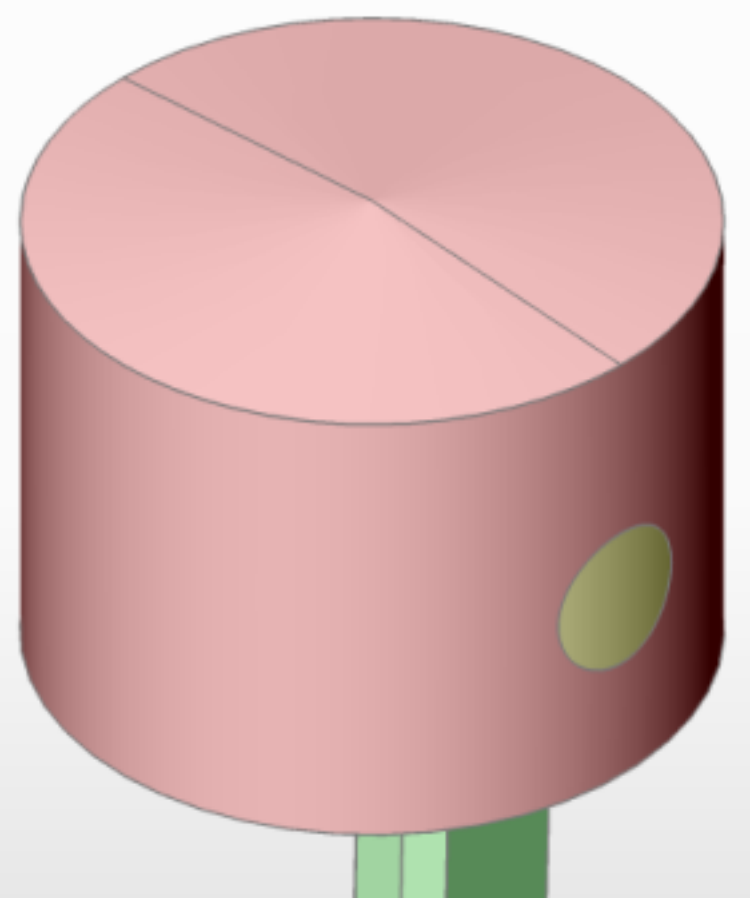


Figure 3.125



22. Click the **Fix** tool button ( ) on the **Solid | 3D Constraints** panel in the ribbon. You are prompted: Select an edge, face or 3D solid:
23. Click the outside cylindrical face of the cylinder.

Figure 3.126

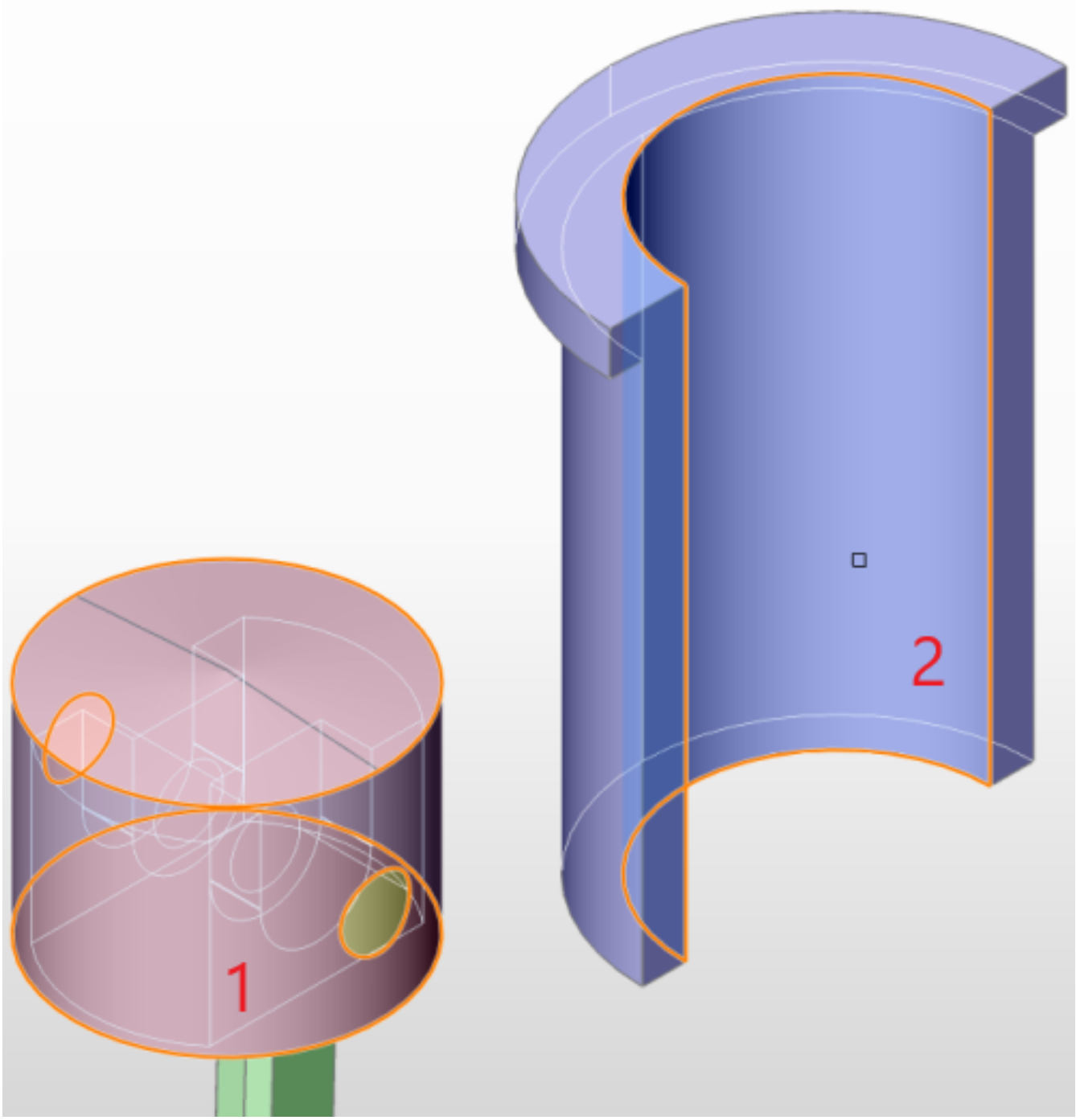


Figure 3.127

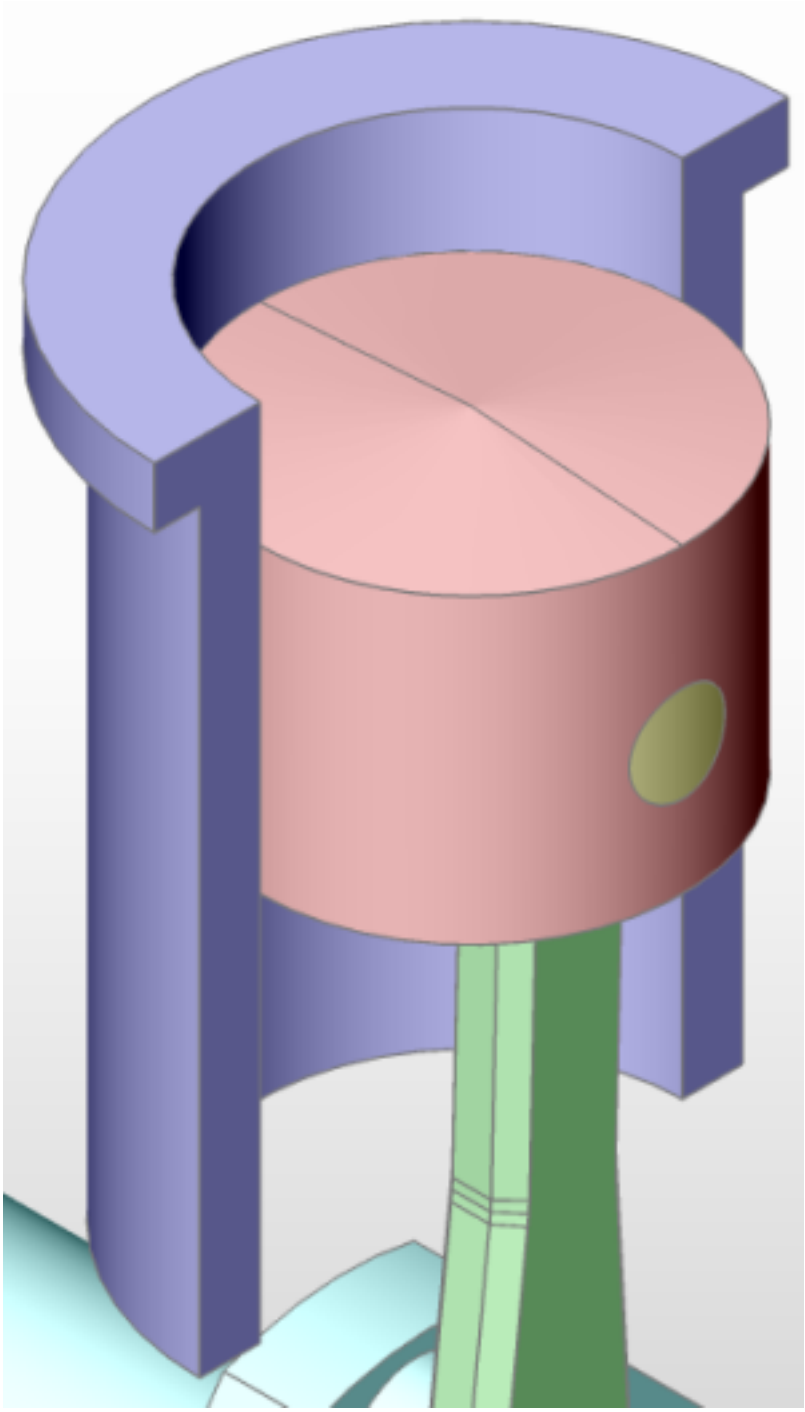
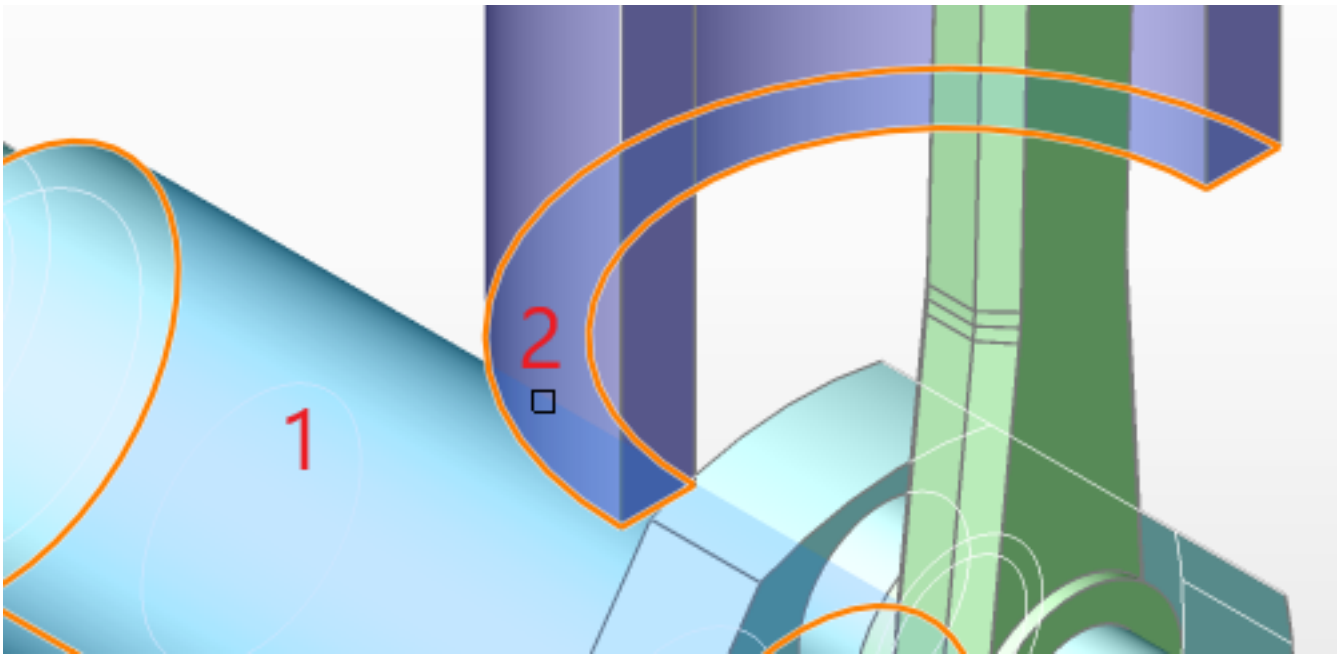


Figure 3.128



Step 5: Kinematic Analysis

In this assembly the crankshaft is the driving component. To run the kinematic analysis we will rotate the crankshaft around its axis, which in this case is the Y-axis of the WCS.

1. Move the cursor over the crankshaft, then choose **Rotate** ( ) in the **Model** command group in the Quad (2.2.3) cursor menu. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017979373/mce
   You are prompted: **Select axial entity or define axis by [2Points/Xaxis/Yaxis/Zaxis] <2Points>:**
2. Do one of the following:
   - Type **Y**, then press Enter.
   - Choose **Y axis** in the prompt menu.

   You are prompted: **Pick start point in the rotation plane:**
3. Pick a point in the drawing, then move the mouse to define the rotation angle. The piston and the connecting rod move dynamically along with the rotation of the crankshaft, while the cylinder stays at its fixed position. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017255994/mceclip23.png})

Figure 3.129

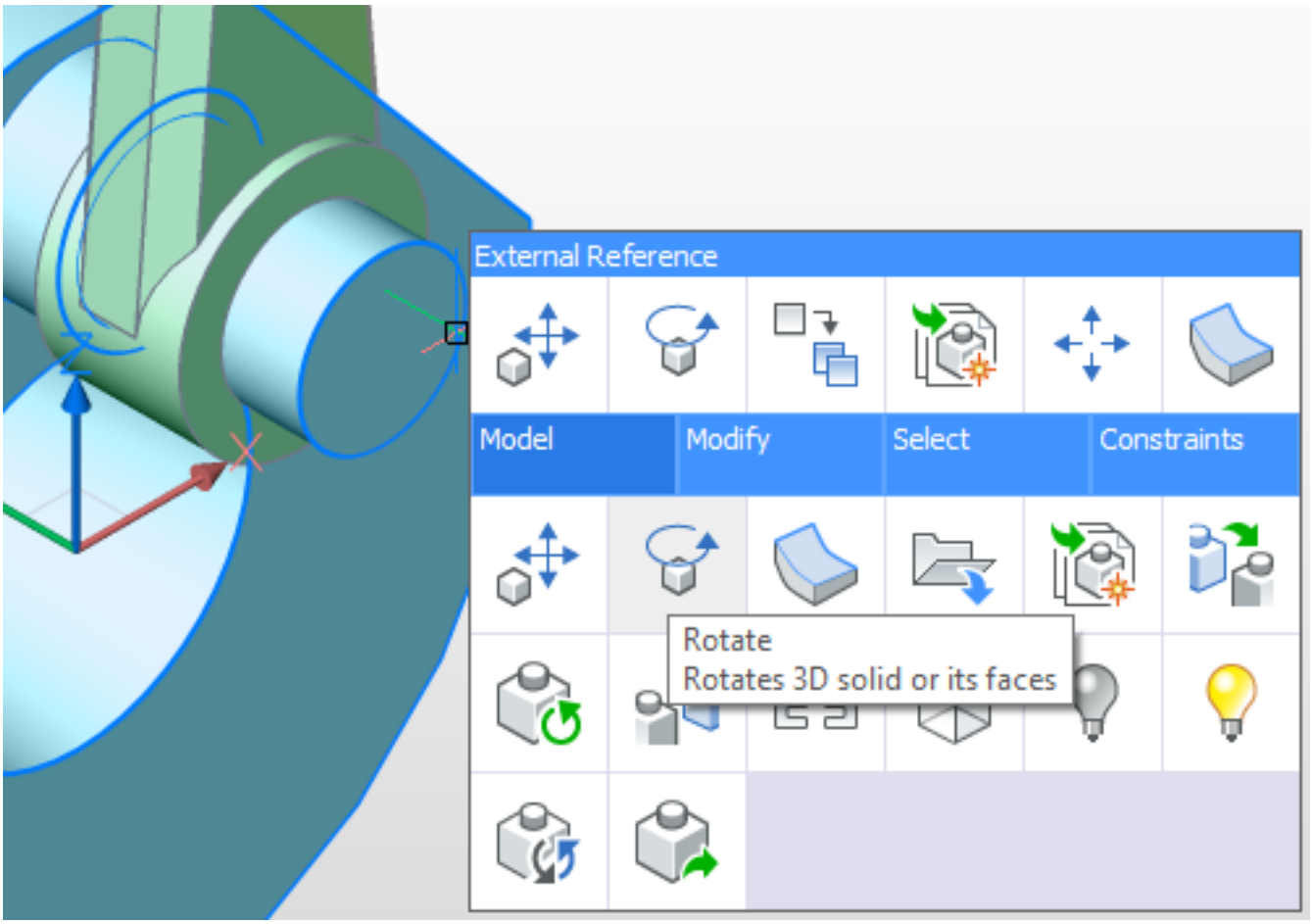
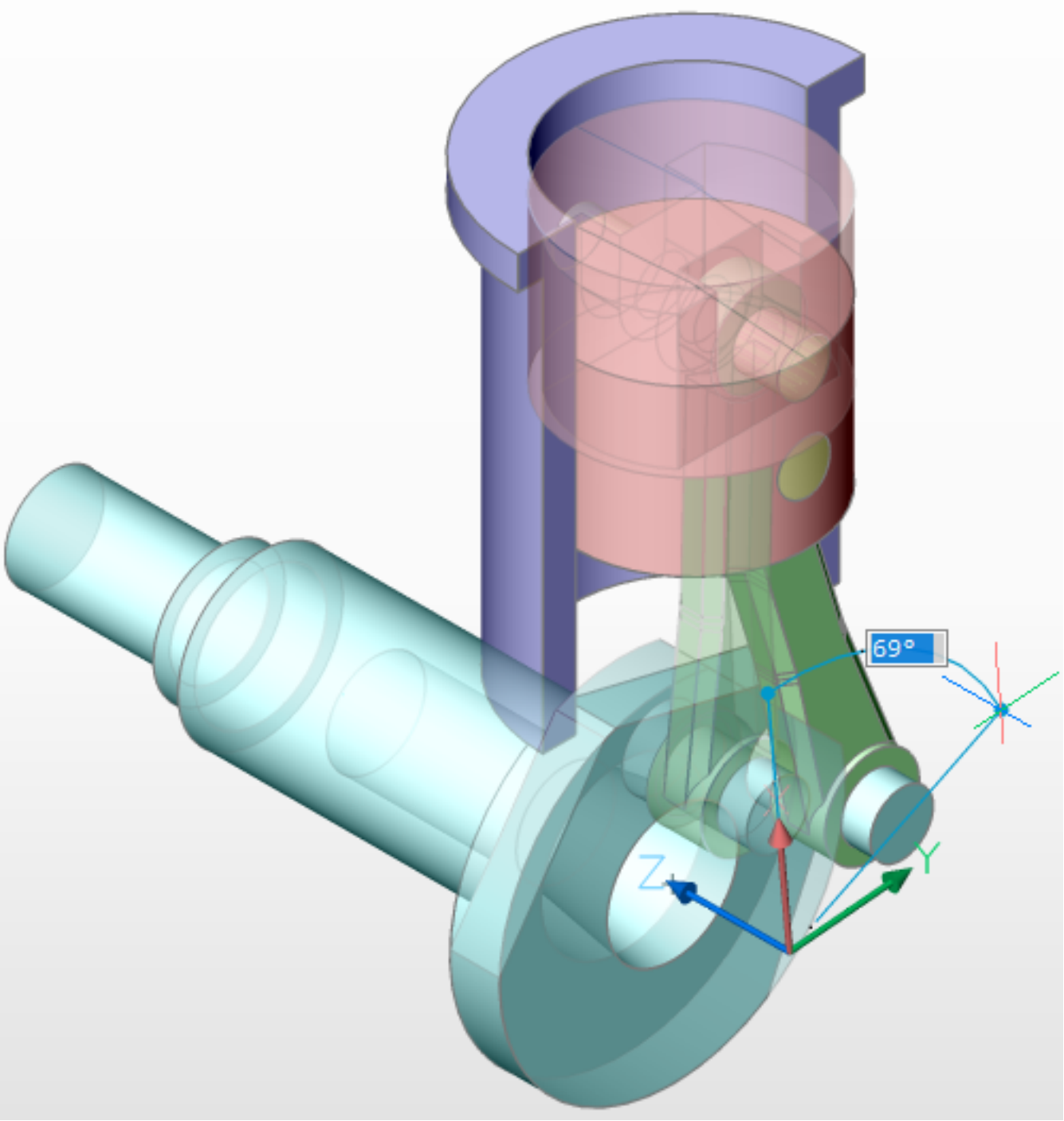

External Reference
Model
Modify
Select
Constraints
Rotate
Rotates 3D solid or its faces

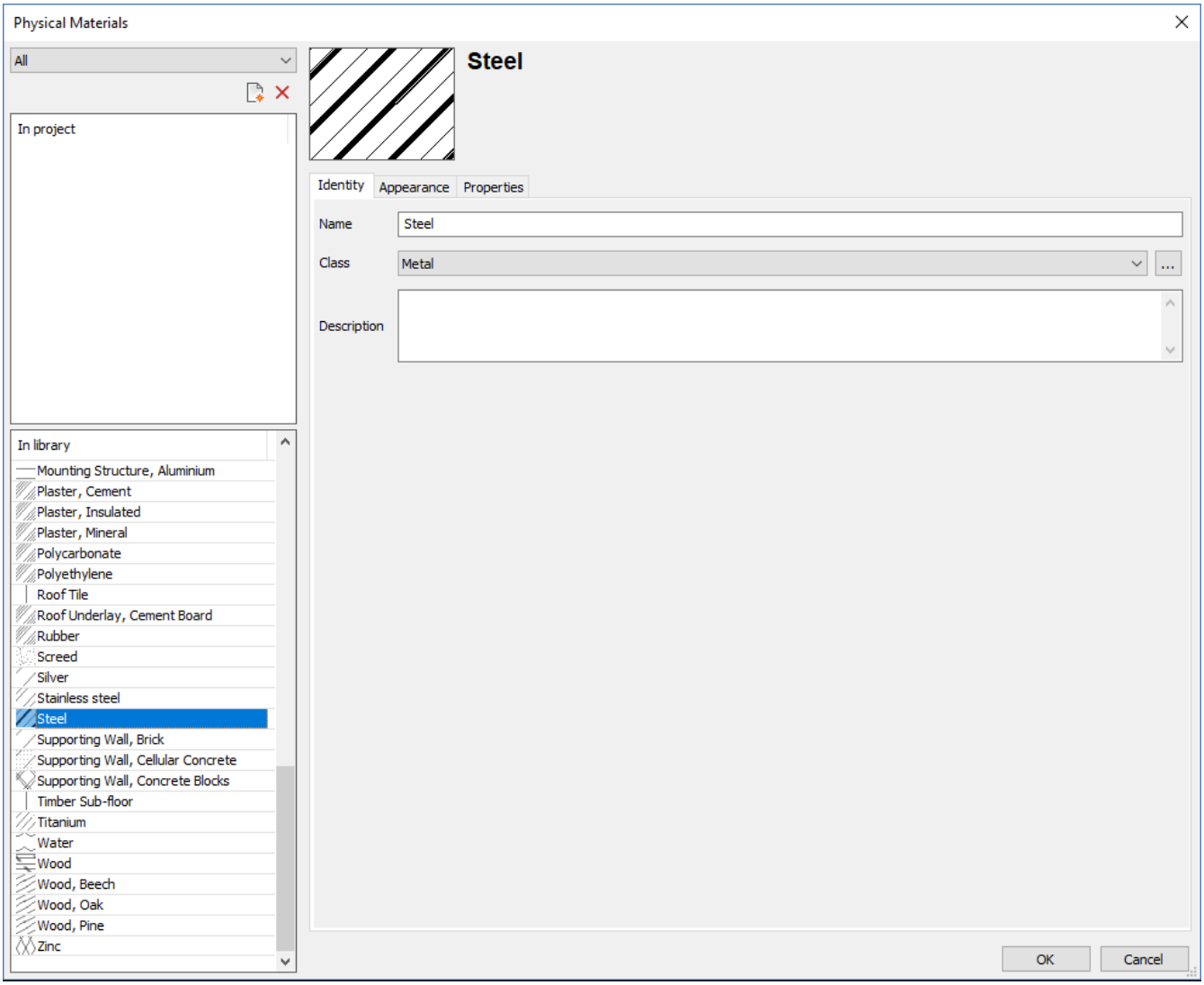
Figure 3.130



Step 6: Material assignment

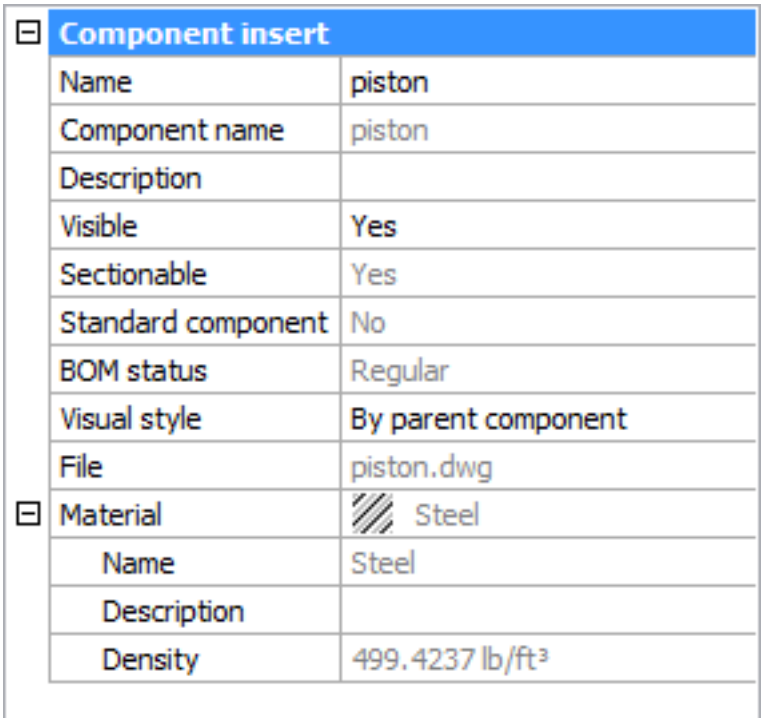
1. Select the**piston** onthe Mechanical Browser (6.8.29).
2. Click **Open**( ) on the right click menu.
3. In Property of the Mechanical Browser select the**Material**field, then click buttonand specify the **Steel** material in the **Physical Materials** dialog box. Click the **OK**button. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/
4. Save **piston.dwg** file.
5. Return to **Piston Engine.dwg** file.
6. Click **Update** button ( ) on the right click menu for the piston. The value of the **Material** setting has changed. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017256194/mceclip25.png})

Figure 3.131


Physical Materials
All
In project
In library
Mounting Structure, Aluminium
Plaster, Cement
Plaster, Insulated
Plaster, Mineral
Polycarbonate
Polyethylene
Roof Tile
Roof Underlay, Cement Board
Rubber
Screed
Silver
Stainless steel
Steel
Supporting Wall, Brick
Supporting Wall, Cellular Concrete
Supporting Wall, Concrete Blocks
Timber Sub-floor
Titanium
Water
Wood
Wood, Beech
Wood, Oak
Wood, Pine
Zinc
Steel
Identity
Appearance
Properties
Name
Steel
Class
Metal
Description
OK
Cancel

7. Repeat the 2-8 steps for the other components:**crankshaft, cylinder,pin,**and**connecting rod.**

Figure 3.132



Component insert	
Name	piston
Component name	piston
Description	
Visible	Yes
Sectionable	Yes
Standard component	No
BOM status	Regular
Visual style	By parent component
File	piston.dwg
Material	Steel
Name	Steel
Description	
Density	499.4237 lb/ft³

Step 7: Mass properties

1. Open theSettings (1.13.11)dialog box.
2. Expand the**Property units**settings under**Drawing | Drafting | Drawing**units. (\vref {fig:C:/Dev/Task-2448/images/helpbric
3. Check the**Format mass properties**checkbox to enable formatting of mass values with appropriate units.
4. Select the**Mass units**field, then click the**Browse**button ( ) and specify the desired mass units in the **MASSUNITS** dialog box.
5. Check the**Slugs**checkbox. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017904433
6. Specify the 0.00 precision in the**Linear units precision** and **Mass precision**fields.
7. Close theSettings (1.13.11)dialog box.

Figure 3.133

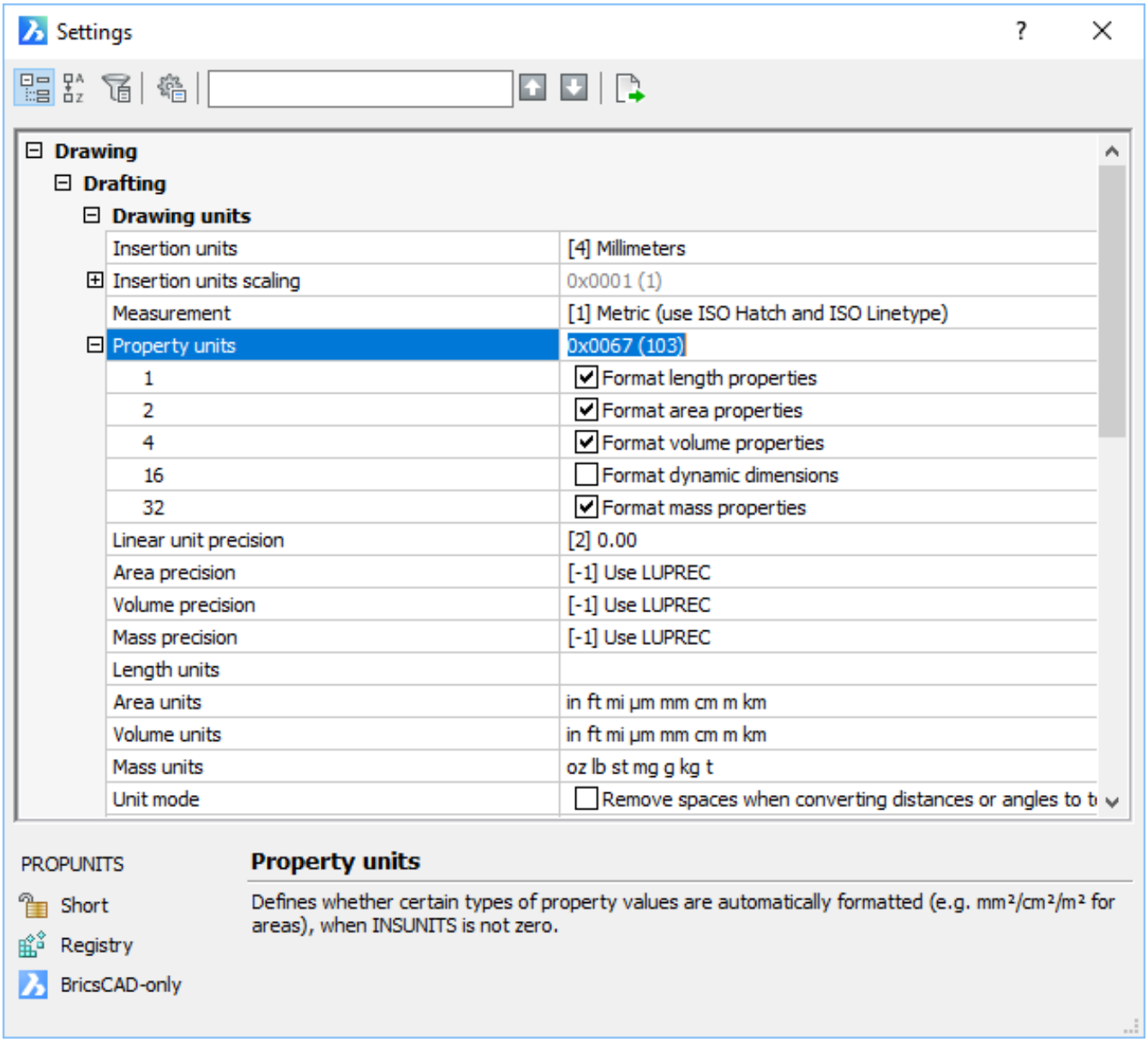
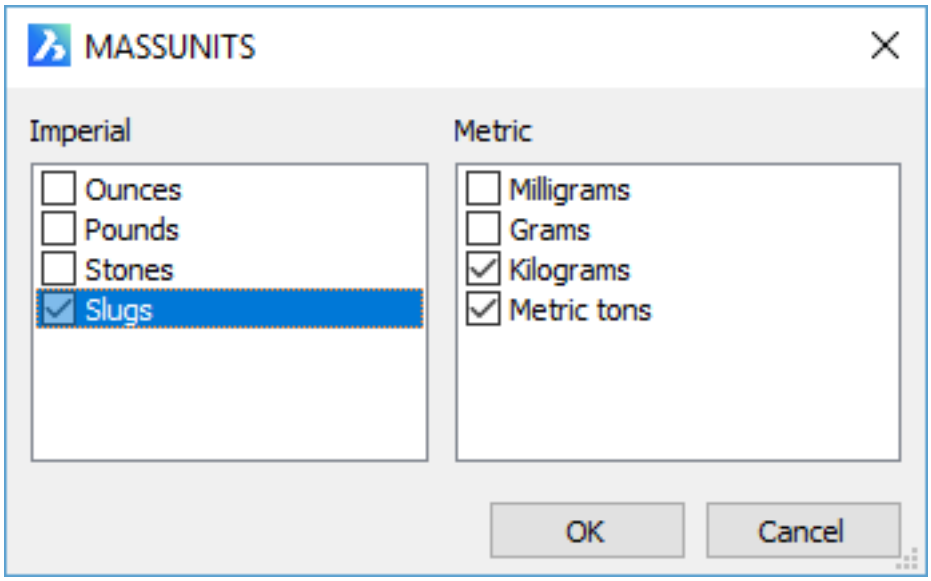


Figure 3.134



Step 8: Drawing generation

1. Select all components.
2. Click the**Base Views**tool button () on the**Annotate | Drawing Views**panel in the ribbon.
3. Enter new or existing layout name or click **Enter** to take **<Layout1>**.
4. On the **VIEWBASE** menu select the **Projection type | First angle.**
5. On the **VIEWBASE** menu select the **Orientation | Front.**
6. On the **VIEWBASE** menu select the **Scale | Custom scale.**
7. Enter scale**: 0.005.**
8. Select positions for base views.

Step 9: Exploded view

1. Switch to model space.
2. Click the**Explode**tool button () on the**Assembly | Tools**panel in the ribbon.
3. In the **BMEXPLODE** dialog box select **Table by Level.**
4. In the **BMEXPLODE** dialog box select **Generate drawingviews.**
5. Enter new or existing layout name or clickEnterto take**<Layout1>**.
6. Select position of the exploded representation.
7. Freeze **"Drafting Viewports"** layerin the drop-down **Layers control**menu on the **Home| Layers**panel in the ribbon.

Step 10: Bill of Materials

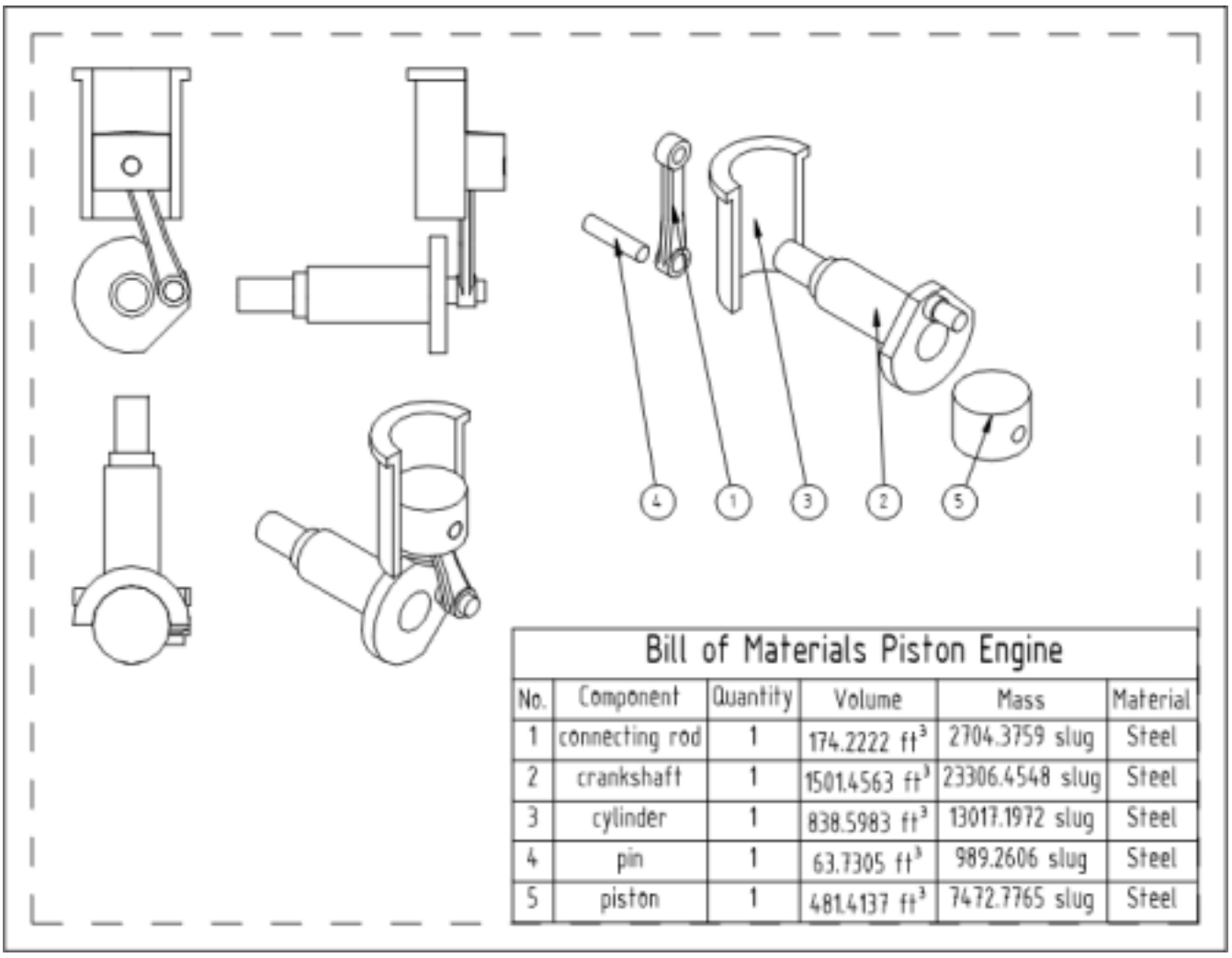
1. Click**Bill of Materials**tool button () on the**Annotate | Assembly**panel in the ribbon.
2. On the **BMBOM** menu select the **Configure| Volume, Mass, Materials.**
3. Select the insertion point.
4. Enter EXPLORER (6.13.14) command in the Command line.
5. On**Drawing Explorer** menu select **Text styles.**
6. Change **Standard** style:
   - Font: **Isocpeur;**
   - Width Factor: **0.8.**
7. Close **Drawing Explorer** menu.
8. Select **Bill of Materials Piston Engine** table. Use the knobs to align the width of the table to the width of the layout.

Step 11: Balloons

1. Click the**Balloon**tool button () on the**Annotate | Assembly**panel in the ribbon.
2. Select a first component on the View ports with Exploded view. Pick a point to place balloon.
3. Repeat the previous step for the other components**.**
4. Select balloons the 2-5 numbers. In **Property** panel select **Leader arrow | Dot small. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017978693/mceclip26.png})**

Back to Tutorial Overview

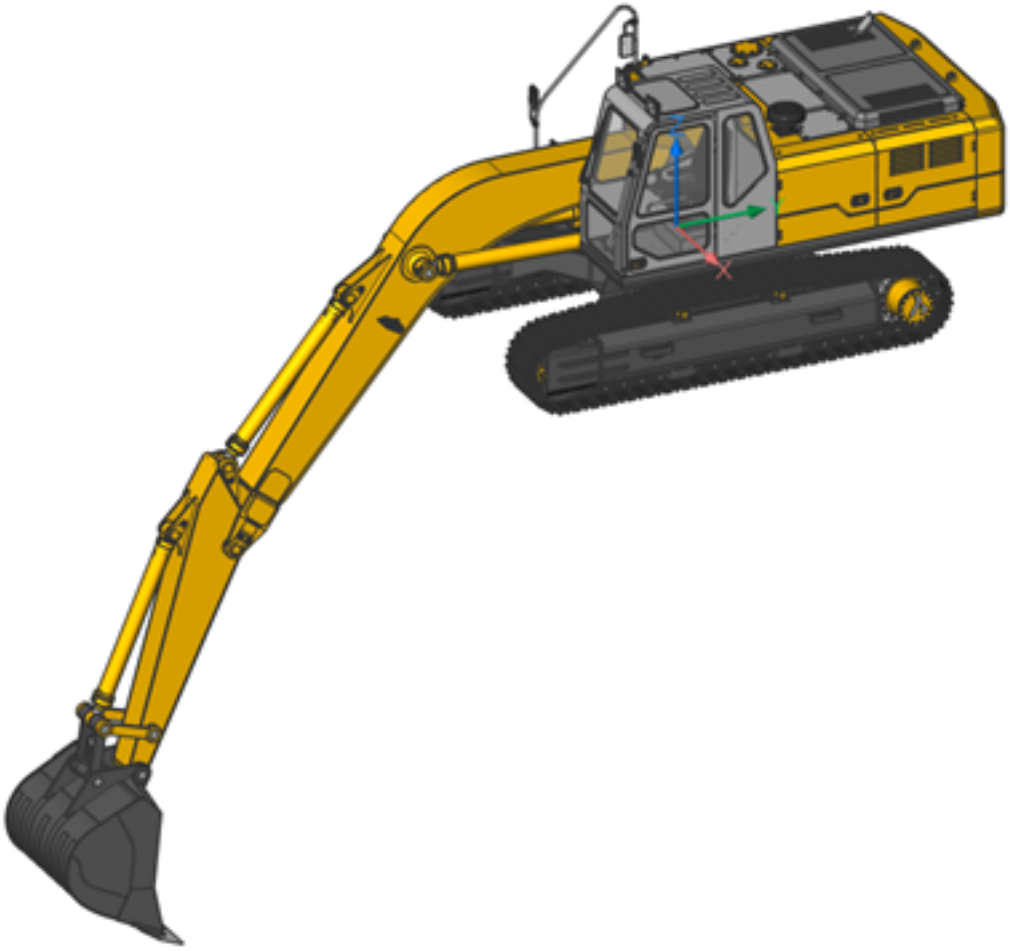
Figure 3.135



Bill of Materials Piston Engine

No.	Component	Quantity	Volume	Mass	Material
1	connecting rod	1	174.2222 ft^3	2704.3759 slug	Steel
2	crankshaft	1	1501.4563 ft^3	23306.4548 slug	Steel
3	cylinder	1	838.5983 ft^3	13017.1972 slug	Steel
4	pin	1	63.7305 ft^3	989.2606 slug	Steel
5	piston	1	481.4137 ft^3	7472.7765 slug	Steel

Figure 3.136



3.3.7 Inverse and direct analysis of kinematics

You need a**BricsCAD**Platinumand **Mechanical**licenses for this tutorial.

1. Open the **Excavator_Assembly.dwg**file: **Kinematic_analysis_inverse_and_direct_kinematics_files.zip**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039324454/mceclip0.png})
2. Enable the **UCSOrtho**option inthe Settings (6.27.23)dialog (the UCSORTHO system variable is 1). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039137253/mceclip0.png})



3. Switch to **Right** view with theLookFrom (6.20.39)widget.
4. Hover over the**Bucket.**
5. Click**Move (6.12.17)** ( ) in the **Model** section of theQuad (2.2.3).
6. Move the bucket-stick-boom and its pins up and left.
7. Hover over the **Upper_Platform_Assembly**. Click **Add Fixation (6.12.15)** ( ) from the **Constraint** section of the Quad (2.2.3).
8. Click the **Concentric constraint (6.12.6)** ( ) tool button on the **Assembly** | **3D constraints** tab on theRibbon (1.13.6).
9. Select the cylindrical faces of the **Boom** and of **Pin:1**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleatt
10. Hover over the **Bucket.**

Figure 3.137

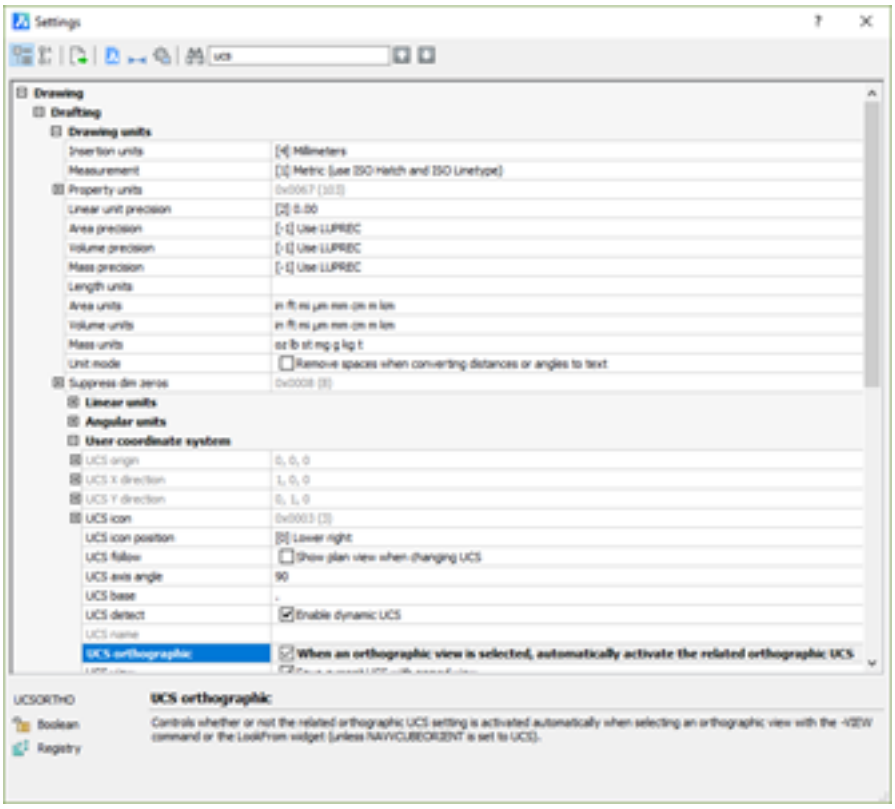


Figure 3.138

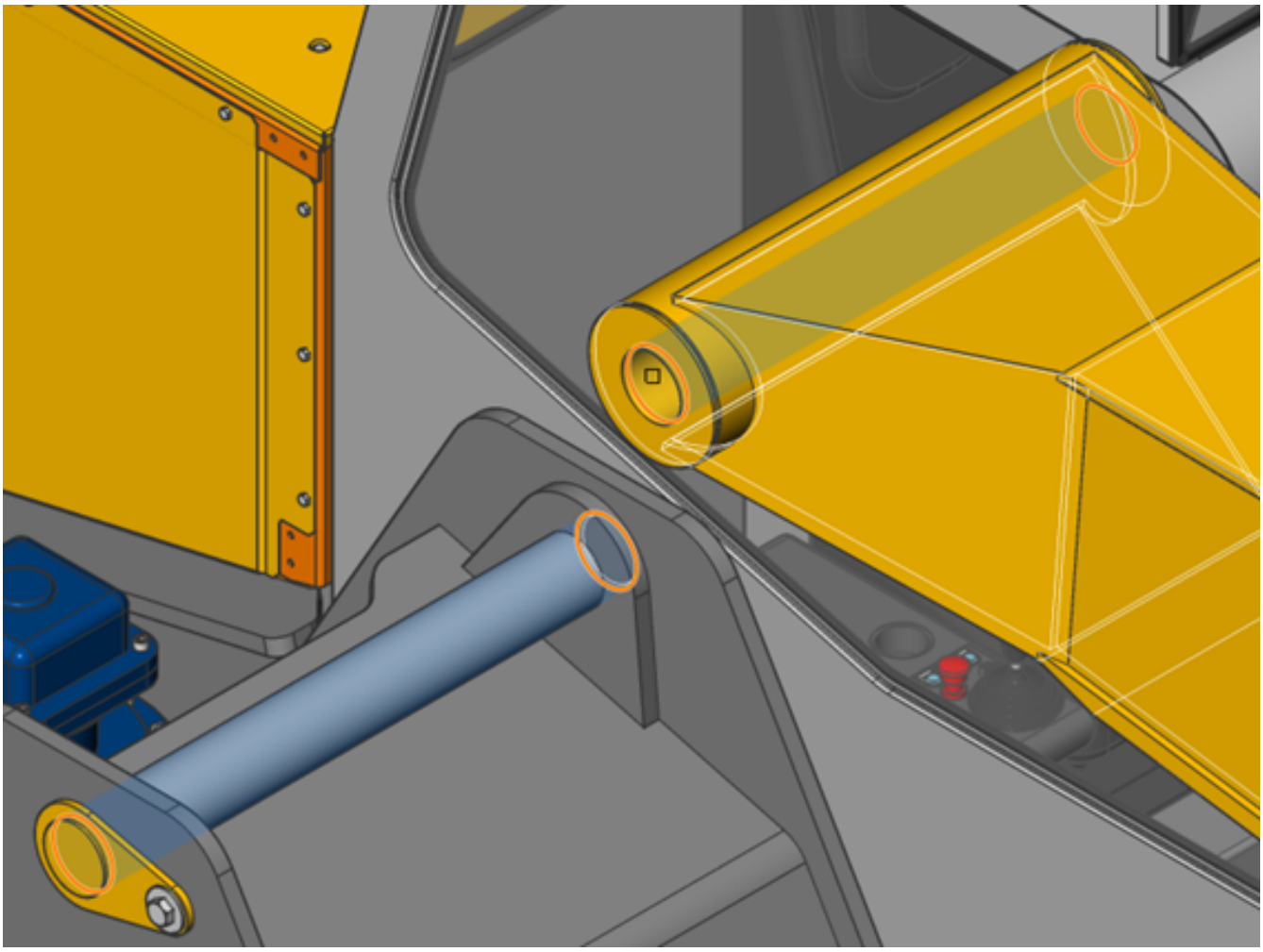
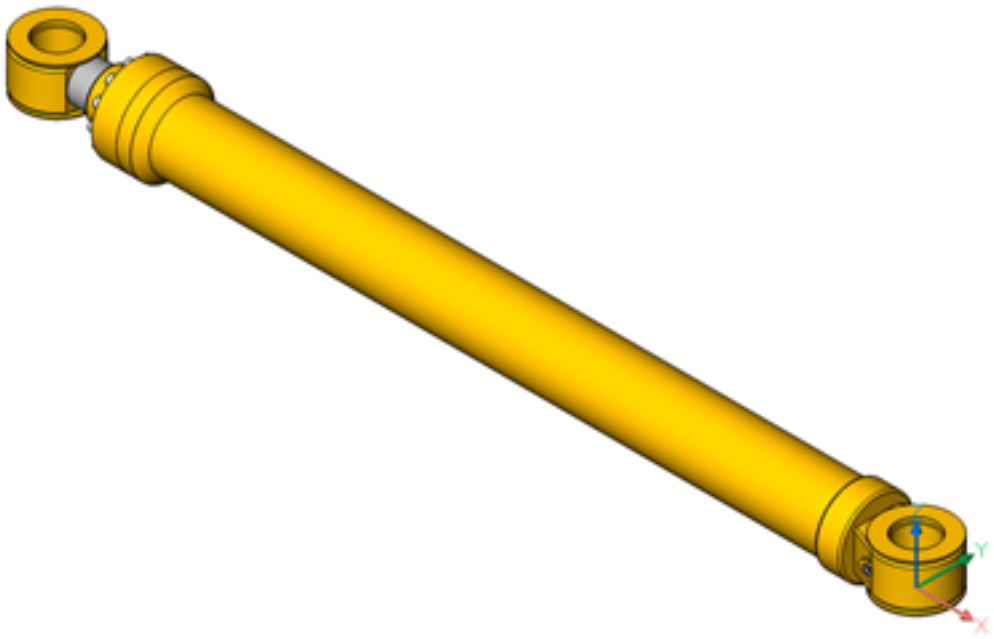


Figure 3.139



11. Click**Move (6.12.17)** ( )from the**Model** section of the Quad (2.2.3)to see how the bucket moves with the new constraints. Hit ESC to cancel moving.
12. Open **Hydrailic_Cylinder_Assembly** with the Mechanical Browser (3.2.2). (\vref {fig:C:/Dev/Task-2448/images/helpbrics
13. Hover over the **Cylinder.** Select **Add Fixation (6.12.15)** ( ) from the **Constraint** section of the Quad (2.2.3).
14. Use a Green Crossing Window Selection (1.23.6) and select the **Stock,** 2 **Washers,** and the **Pipe Plug. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040170653/mceclip12.png})**
15. Click **Add Rigid Set (6.12.23)** ( ) from the **Constraint** section of the Quad (2.2.3).
16. Call the context menu (right-click menu) on **Hydrailic_Cylinder_Assembly** component in the Mechanical Browser (3.2.2).
17. Click the **Add new parameter (6.24.5)** ( ) button.
18. Select the new parameter (**v1**) in the Mechanical Browser (3.2.2).
19. On the bottom part of the Mechanical Browser (3.2.2), click the **Name** field and rename the parameter to **Offset.**
20. In the bottom part of the Mechanical Browser (3.2.2), click the **Expression** field and change the parameter value to **0**.
21. Click the **Distance constraint (6.12.12)** ( ) tool button on the **Assembly** | **3D constraints** panel on the Ribbon (1.13.6).
22. Select the cylindrical faces of the **Stock** and of the **Cylinder**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/art
23. Enter **2200+Offset** as the distance value.
24. Repeat step 20 to change the **Expression** of the **Offset** parameter to **500** on the bottom part of the Mechanical Browser (3.2.2). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039329454/mceclip1
25. Click the **Animate** tool button in the context menu(right-click menu) of the **Offset** parameter in the Mechanical Browser (3.2.2) to make sure it works. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360
26. Repeat step 20 to change the **Expression** of the **Offset** parameter to **0** on the bottom part of the Ribbon (1.13.6).
27. Click the **Save (6.27.1)** ( ) tool button on the **Home** | **File** panel on the Ribbon (1.13.6) to save the**Hydrailic_Cylinder_As** file.
28. Switch to the **Excavator_Assembly.dwg** file.
29. Select **Hydrailic_Cylinder_Assembly** from the Mechanical Browser (3.2.2).

Figure 3.140

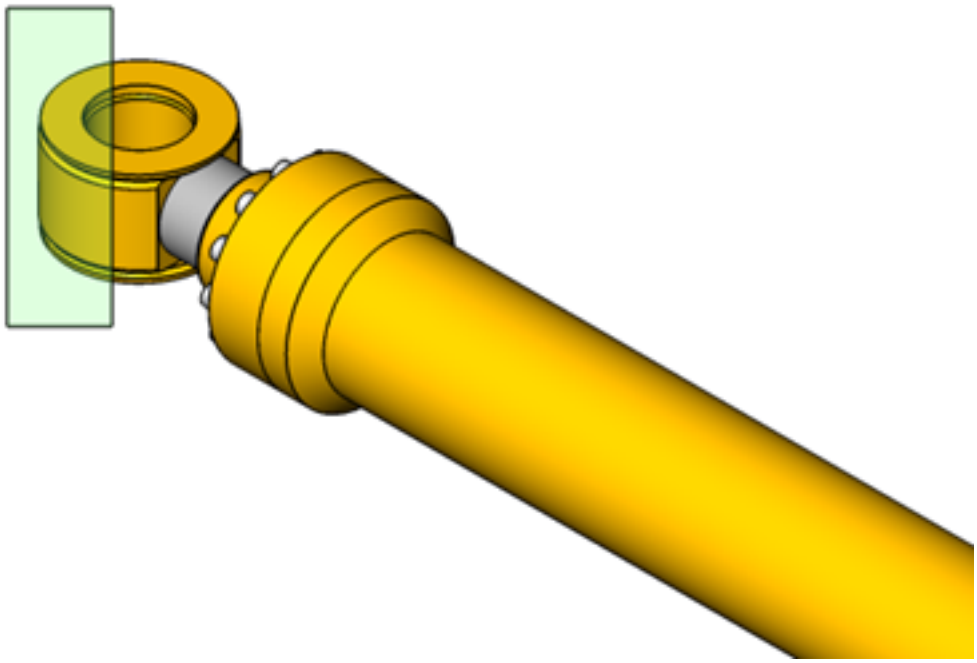


Figure 3.141

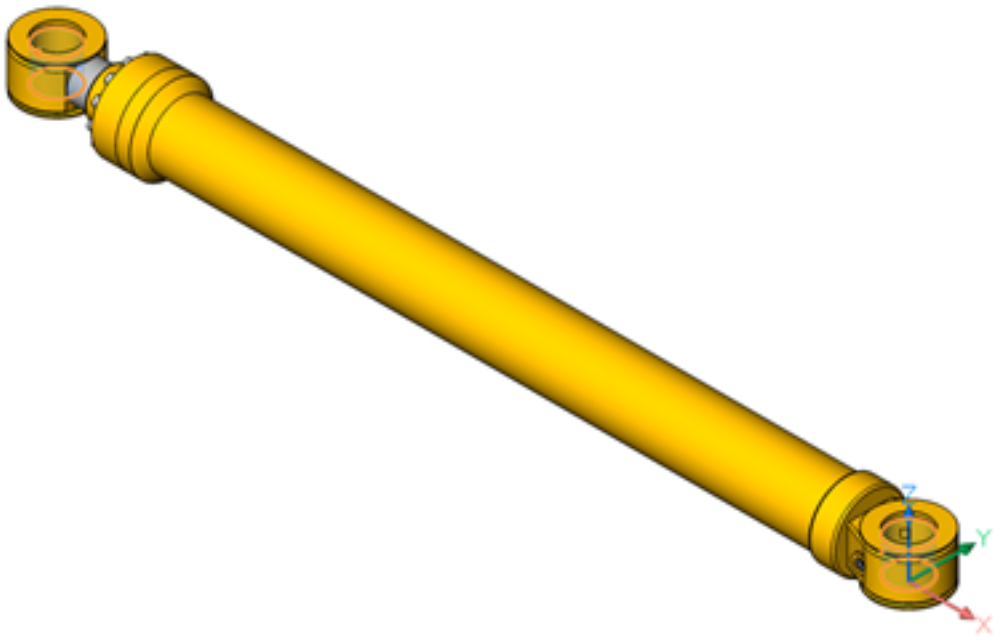
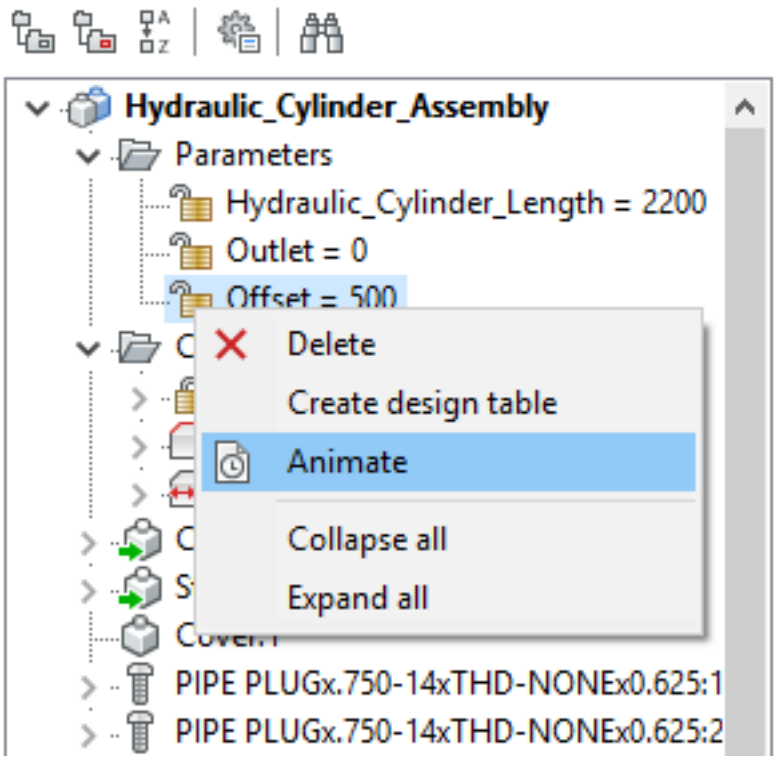


Figure 3.142

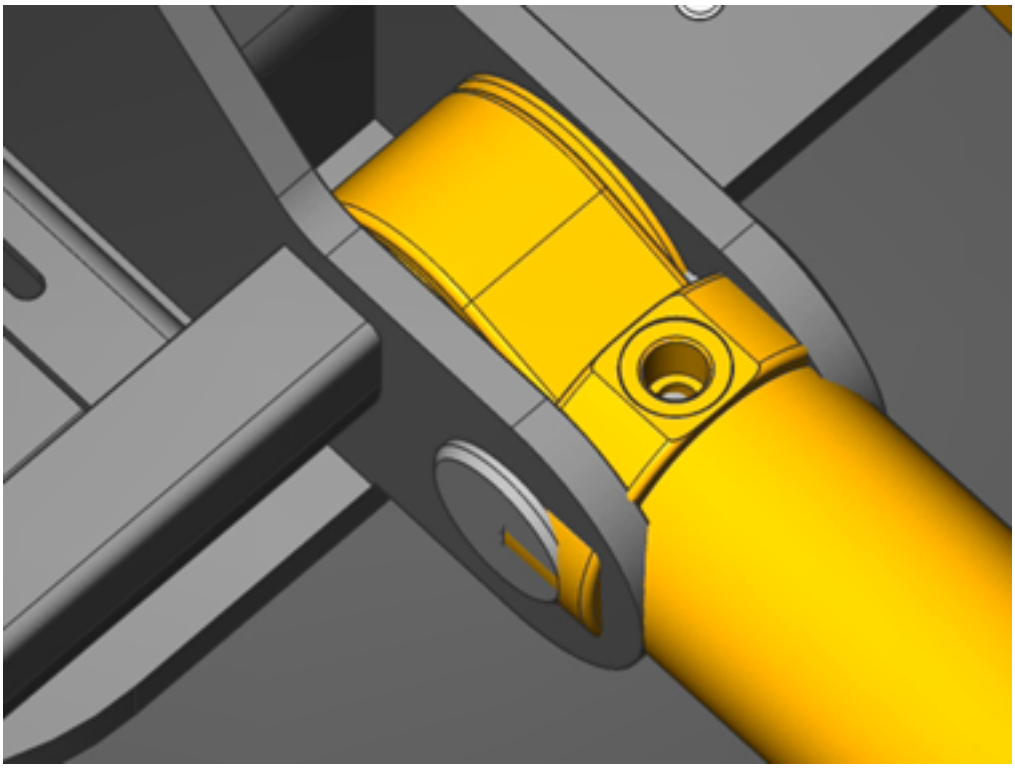
Parameter	
Name	Offset
Expression	500
Value	0
Description	
Exposed	Auto
Units	

Figure 3.143



30. Click **Update (6.8.25)** ( ) on the context menu.
31. Select the **Offset** parameter of the **Hydrailic_Cylinder_Assembly:1** component on the Mechanical Browser (3.2.2).
32. Click **Link to parameter** in the context menu(right-click menu)
33. Repeat previous steps to **Offset** the parameter of the **Hydrailic_Cylinder_Assembly:2** component**.**
34. Change the name of the **Offset** parameter to **Boom_Offset** on the bottom part of the Mechanical Browser (3.2.2).
35. Change the **Expression** of the **Boom_Offset** parameter to **100** on the bottom part of the Mechanical Browser (3.2.2). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039329174/mceclip5.png})
36. Repeat steps 9-10 to create a**Concentric constraint (6.12.6)** ( )between the cylindrical faces of **Hydrailic_Cylinder_Ass and Pin:2.**
37. Repeat steps 9-10 to create a**Concentric constraint (6.12.6)** ( ) between the cylindrical faces of **Hydrailic_Cylinder_As and Pin:3. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040170373/mceclip6.pn**

Figure 3.144



38. Change the expression of the **Boom_Offset** parameter to **500** to make sure it works. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039329274/mceclip7.png})
39. Select the **Offset** parameter of the **Hydrailic_Cylinder_Assembly:3** component in the Mechanical Browser (3.2.2).
40. Click **Link to parameter** on the right-click menu.
41. Rename the **Offset** parameter to **Stick_Offset**.
42. Change the **Expression** of the **Stick_Offset** parameter to **300**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/a

43. Select the **Offset** parameter of the **Hydrailic_Cylinder_Assembly:4** component in the Mechanical Browser (3.2.2).
44. Click **Link to parameter** on the context menu.
45. Change the **Name** of the **Offset** parameter to **Bucket_Offset**.
46. Change the **Expression** of the **Bucket_Offset** parameter to **100**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039329294/mceclip8.png})

Figure 3.145

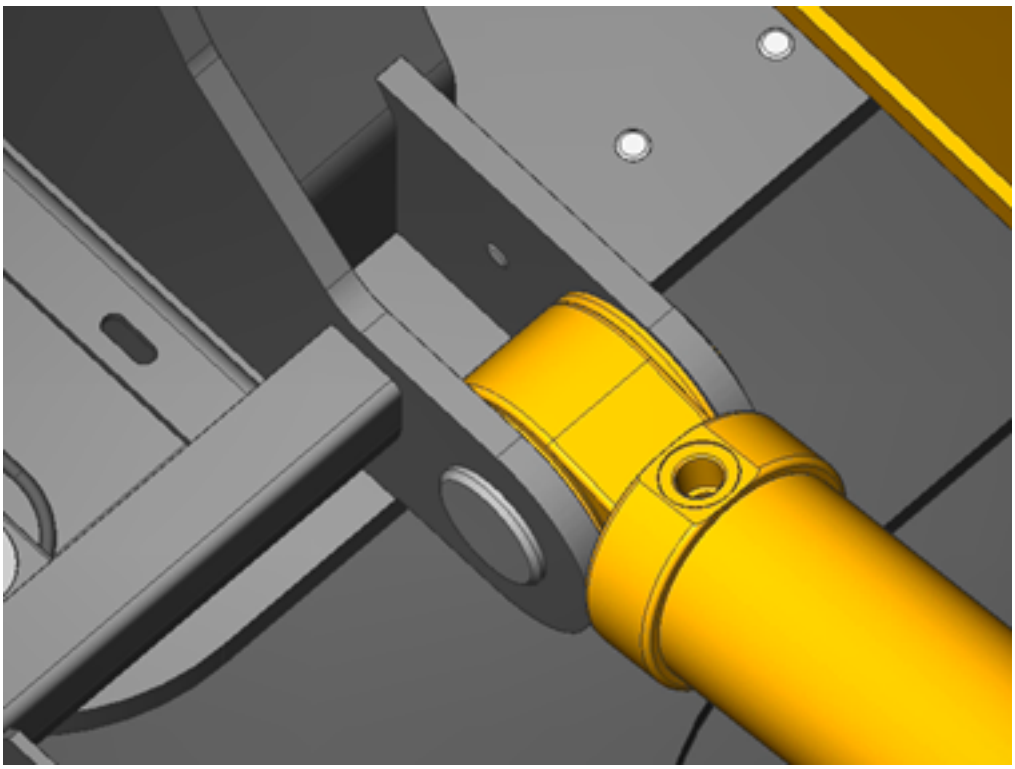


Figure 3.146

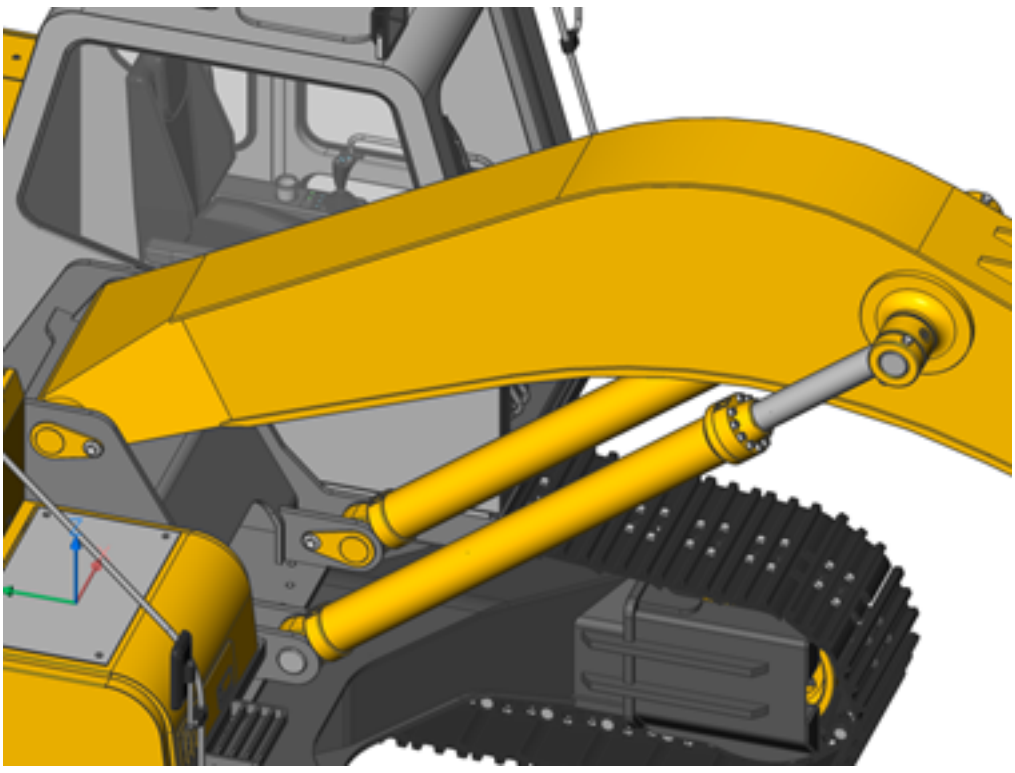


Figure 3.147

Parameter	
Name	Stick_Offset
Expression	300
Value	0
Description	
Exposed	Auto
Units	
BOM status (dbg)	Auto

Figure 3.148

Parameter	
Name	Bucket_Offset
Expression	100
Value	0
Description	
Exposed	Auto
Units	
BOM status (dbg)	Auto

Figure 3.149

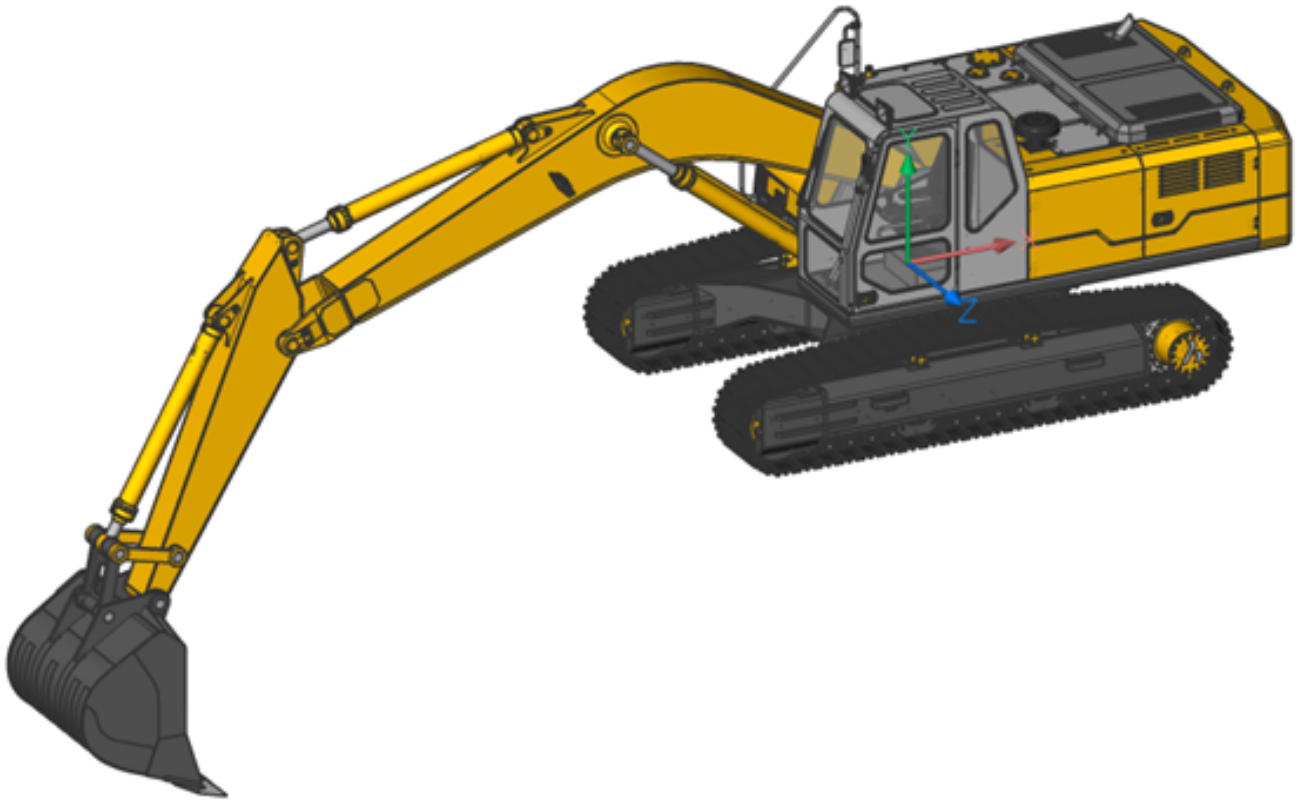
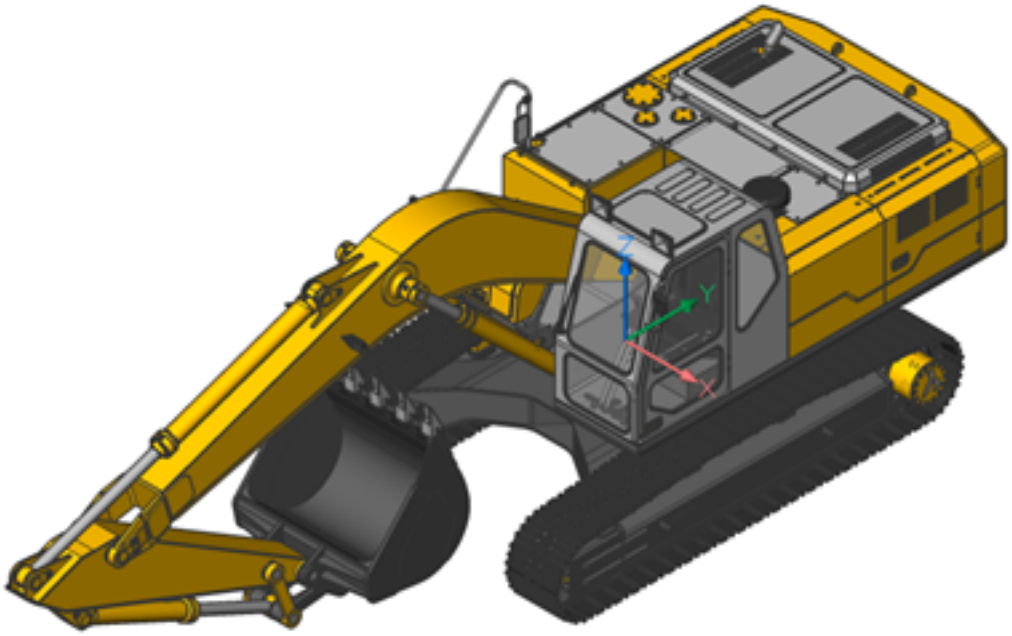


Figure 3.150



3.3.8 Mass properties and BOM

You need**BricsCADPlatinum**and **Mechanical**licenses for this tutorial.

1. Open the **Excavator_Assembly.dwg** file fromExcavator_Assembly.zip. (\vref {fig:C:/Dev/Task-2448/images/helpbricsysc
2. Enable the **UCSOrtho**option inthe Settings (6.27.23)dialog (the UCSORTHO system variable is 1). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039137253/mceclip0.png})
3. Click **Mass Properties (6.21.4)** ( ) on the **Assembly** | **Inquire** panel on the Ribbon (1.13.6).
4. Hit **Enter** to select the entire model.
5. Select the **Yes** option on the **BmMassProp** prompt menu to keep the UCS aligned with the principal moment axes. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040197053/mceclip1.png})
6. Change the **Expression** of the **Rotate_Angle** parameter to **90**.
7. Repeat steps 4-6 for different positions. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36 Make sure that, in all positions, the excavator is stable.
8. Switch to the **Top** view on the **LookFrom** widget to activate **World UCS**.
9. Click Bill of material (6.8.2) ( ) on the **Assembly** | **Inquire** panel on the Ribbon (1.13.6).
10. Select the **Configure, Material, Mass** options on the **BmBom** prompt menu.
11. Click **Back** on the **BmBom** prompt menu to return to the main menu.
12. Select the **Hierarchical** option on the **BmBom** prompt menu.
13. Set the insertion point. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039353034/mc (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039353074/mceclip4.png})
14. Hover over the **BOM table** and click the Export table (6.29.4) ( ) tool button on the**Modify** section fo the Quad (1.12.5).
15. Save the file as **Hierarchical_BOM.csv.**
16. Open the **Hierarchical_BOM.csv** table as a native table with a spreadsheet software package. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040197533/mceclip5.png})

Figure 3.151

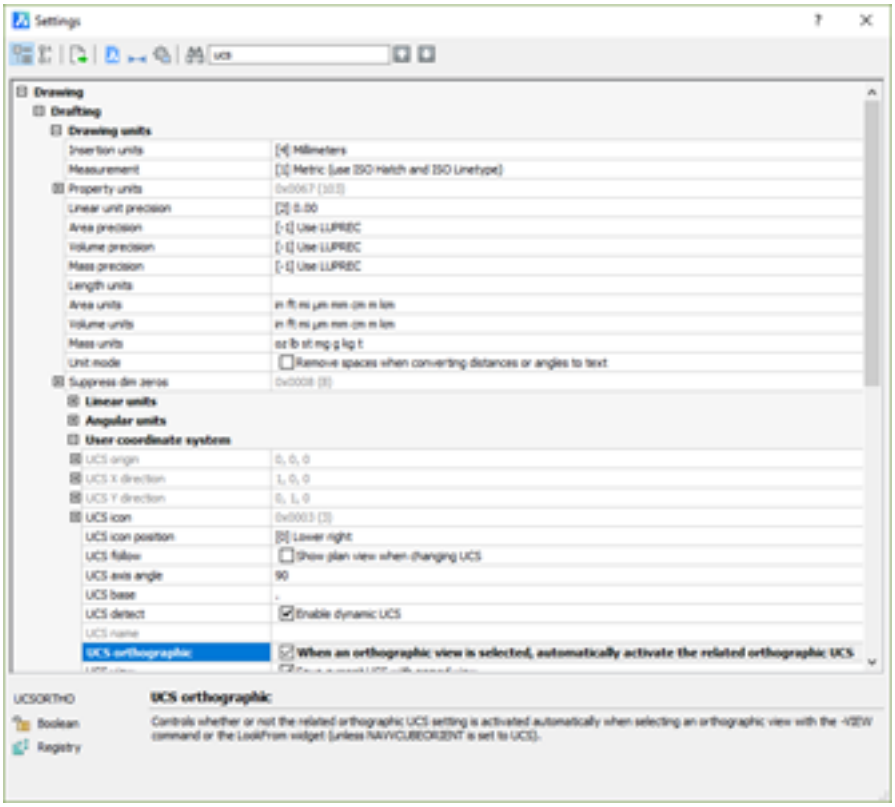


Figure 3.152

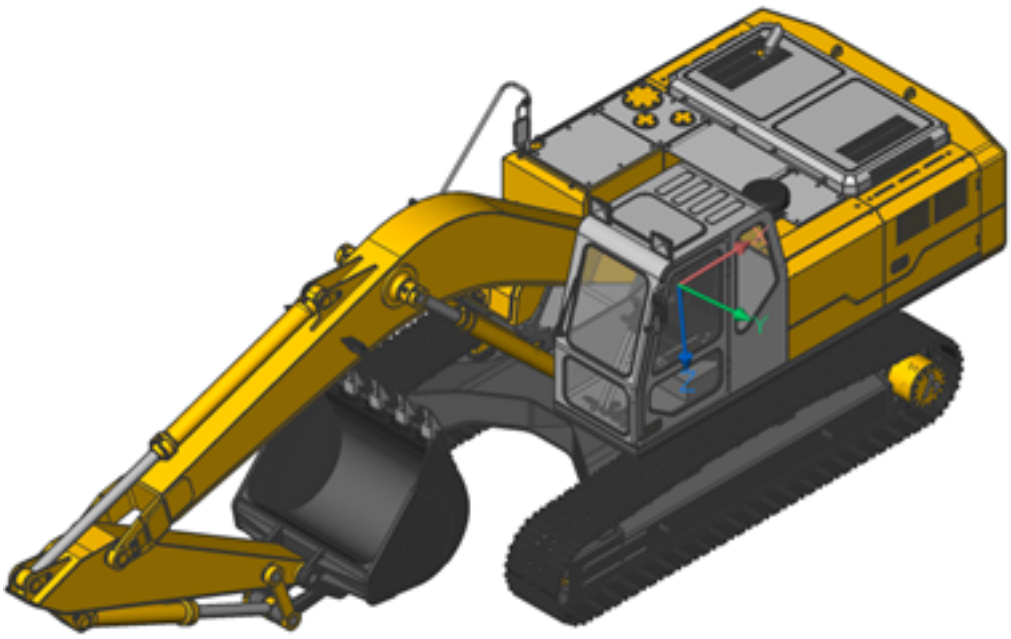


Figure 3.153

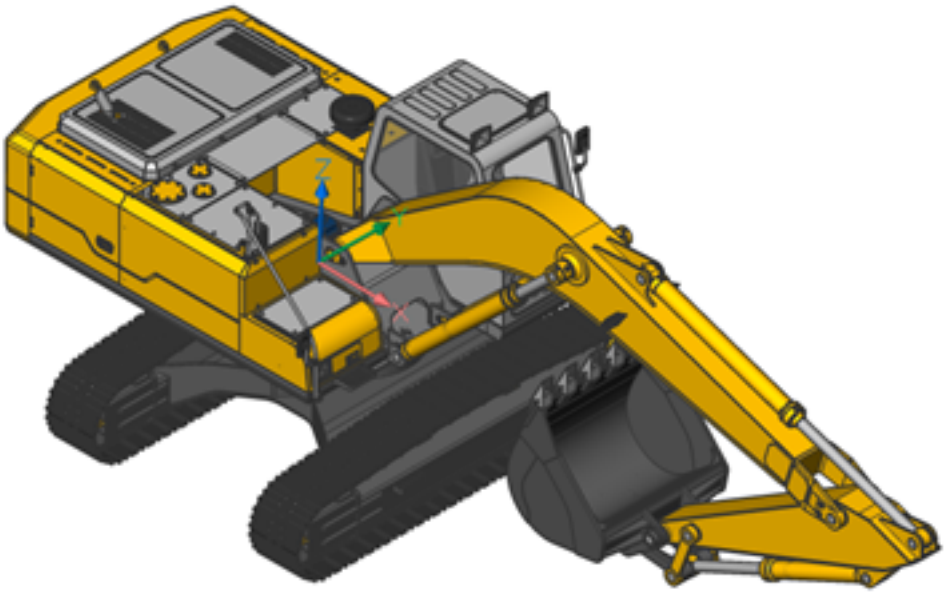


Figure 3.154

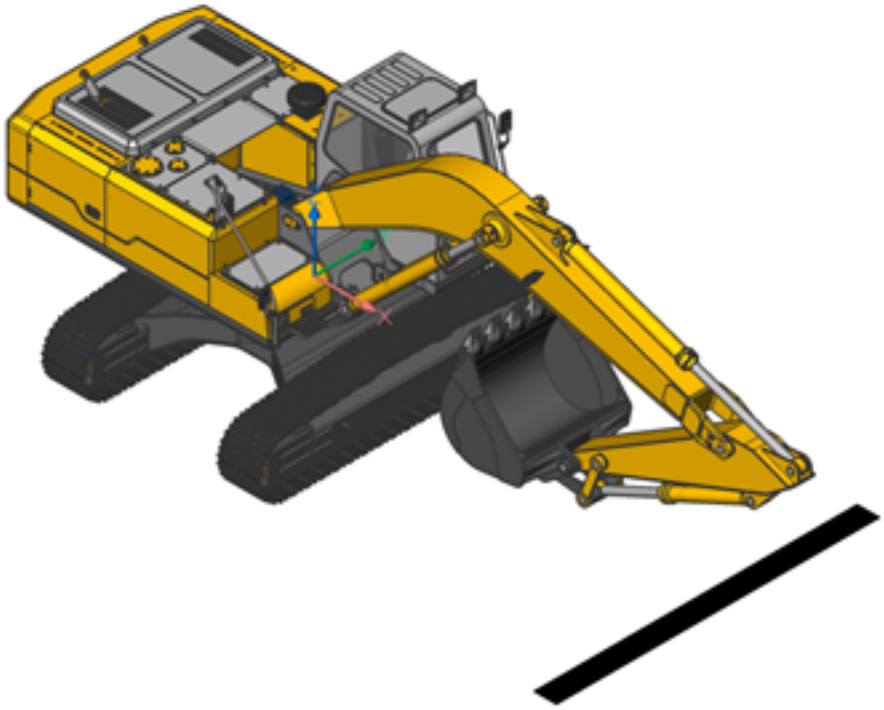


Figure 3.155

Bill of Materials Excavator_Assembly				
No.	Component	Quantity	Material	Mass
1	Boom_Assembly_Parametric	1	Steel	5.78 t
1.1	Boom_Assembly	1	Steel	2.06 t
1.1.1	Boom	1	Steel	1.92 t
1.1.2	Bracket_2	2	Steel	0.37 kg
1.1.2.1	Bracket	2	Steel	0.23 kg
1.1.2.2	DIN 127-B	2	Steel	10.52 g
1.1.2.3	DIN 127-B	4	Steel	1.89 g
1.1.2.4	DIN 933	4	Steel	12.78 g
1.1.2.5	DIN 933	2	Steel	87.63 g
1.1.2.6	DIN 934-B	4	Steel	6.04 g
1.1.3	DIN 934-B	2	Steel	73.10 g
1.1.4	ISO 4016	2	Steel	0.58 kg
1.1.5	LED_Projector_Assembly	1	Aluminium	0.92 kg
1.1.5.1	Body	1	Aluminium	0.44 kg
1.1.5.2	Bracket	1	Steel	0.19 kg
1.1.5.3	DIN 126	2	Steel	1.76 g
1.1.5.4	DIN 127-B	4	Steel	0.43 g
1.1.5.5	DIN 127-B	2	Steel	1.89 g
1.1.5.6	DIN 7985-A	4	Steel	3.53 g

Figure 3.156

	A	B	C	D	E
1	Bill of Materials Excavator_Assembly				
2	No.	Component	Quantity	Material	Mass
3	1	Boom_Assembly_Parametric	1	Steel	5.73 t
4	1.1	Boom_Assembly	1	Steel	2.06 t
5	1.1.1	Boom	1	Steel	1.92 t
6	1.1.2	Bracket_2	2	Steel	0.37 kg
7	1.1.2.1	Bracket	2	Steel	0.23 kg
8	1.1.2.2	DIN 127-B	2	Steel	10.52 g
9	1.1.2.3	DIN 127-B	4	Steel	1.89 g
10	1.1.2.4	DIN 933	4	Steel	12.78 g
11	1.1.2.5	DIN 933	2	Steel	87.63 g
12	1.1.2.6	DIN 934-B	4	Steel	6.04 g
13	1.1.3	DIN 934-B	2	Steel	73.10 g
14	1.1.4	ISO 4016	2	Steel	0.58 kg
15	1.1.5	LED_Projector_Assembly	2	Aluminium	0.92 kg
16	1.1.5.1	Body	2	Aluminium	0.44 kg
17	1.1.5.2	Bracket	2	Steel	0.19 kg
18	1.1.5.3	DIN 126	4	Steel	1.76 g
19	1.1.5.4	DIN 127-B	8	Steel	0.43 g
20	1.1.5.5	DIN 127-B	4	Steel	1.89 g
21	1.1.5.6	DIN 7985-A	8	Steel	3.53 g
22	1.1.5.7	DIN 7985-A	8	Steel	0.60 g
23	1.1.5.8	DIN 933	4	Steel	12.78 g
24	1.1.5.9	DIN 934-B	8	Steel	1.40 g
25	1.1.5.10	Frame	2	Aluminium	38.38 g
26	1.1.5.11	Gasket	2	Rubber	7.95 g

3.3.9 Assembly design using BricsCAD Communicator

Youtube The assembly design functions have been extended in BricsCAD V16. The BricsCAD communicator module enables CAD data exchange by providing import and export capabilities in various native and neutral file formats. Supported formats include neutral ones, such as STEP and IGES; native formats, such as CATIA, Inventor, and SolidWorks; and kernel formats, such as Parasolid. All editions of BricsCAD export drawings and models in ACIS, DWG, 2D PDF, and a variety of raster formats.

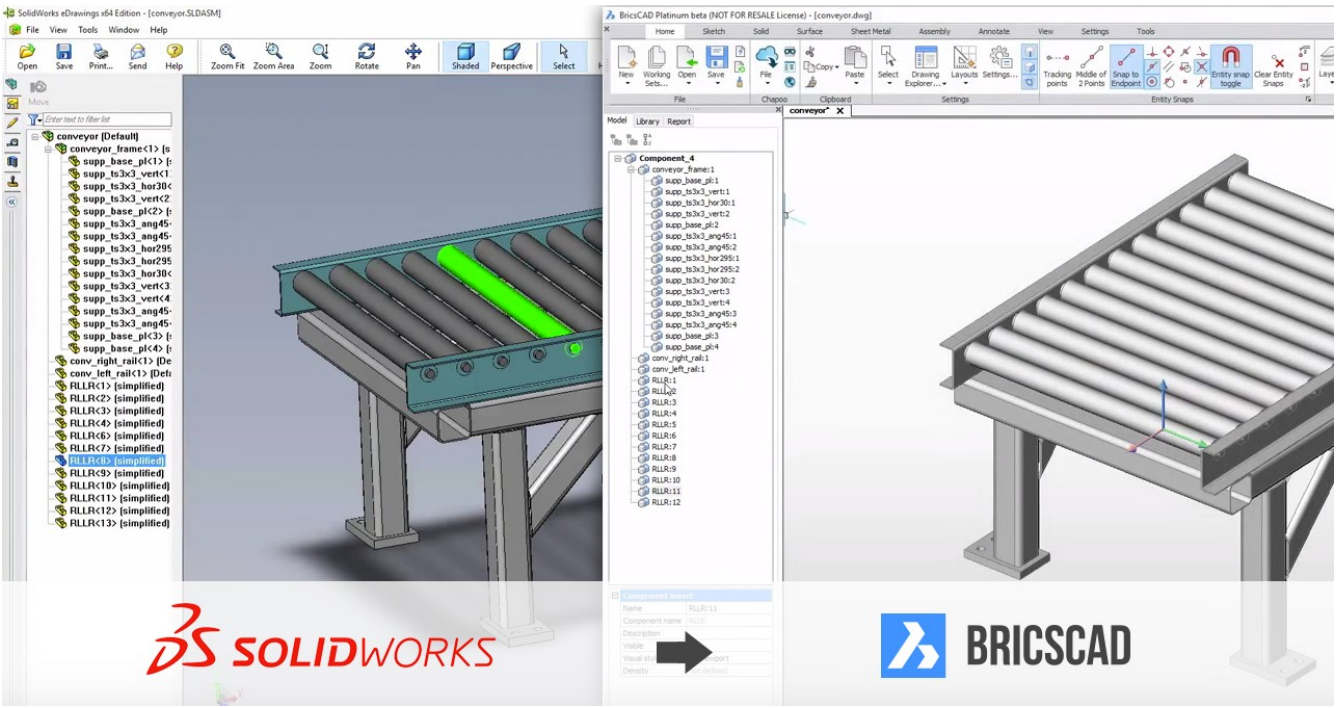
> **In V16 it's now possible to import not only the geometry, but also the product structure from other CAD assembly files.**

In this tutorial, we import a SolidWorks conveyor assembly into BricsCAD. In the settings panel, you can enable "import product structure". BricsCAD now generates the full assembly structure.

Import CAD assemblies including their product structure

You can navigate through the product structure using the Mechanical Browser (3.2.2). Imported parts and subassemblies are represented by local components. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/static/bricsyst

Figure 3.157



Copy local components into a new document using “open as a copy” The “Open as a copy” command allows you to copy a local component into a new document. Change the color of the part and add radius constraints to make this component parametric. After a modification the component can be saved.

Replace components The new “Replace component” function allows you to replace all inserted components with a modified one. Note that all components now have a parameter to control the roller radius. Create a new common parameter to control all these internal parameters simultaneously. All rollers are changed parametrically. By changing one parameter at the assembly level, you can automatically obtain different configurations of the model The “Replace component” function also takes into account the current values of components parameters. While replacing components BricsCAD automatically maintains the values and expressions specified for the assembly. Read more about BricsCAD Communicator Back to Tutorial Overview

3.4 Part Design

3.4.1 Part Design Methods

BricsCAD Mechanical allows you to create and modify 3D models. Ways to create a 3D model.

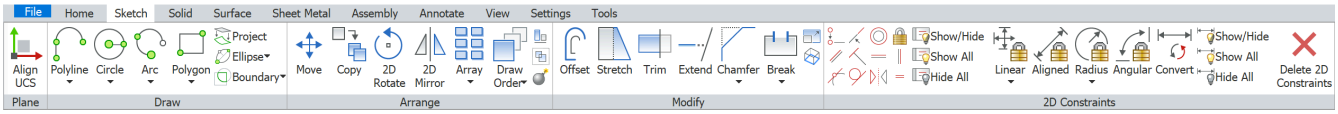
- Create a 3D solid primitive - a box, a cylinder, etc.
- Sketching a 2D profile and extrude or revolve it to make a 3D solid.
- Model a 3D surface and thickening it to make a 3D solid, or stitching several surfaces together to create watertight 3D solid.
- Import existing 3D solid and surface geometry from other CAD programs.

Ways to edit and add more details to your 3D models

- By Pushing/Pulling, moving and rotating faces.
- By creating Fillets or Chamfers on the model edges.
- By applying Holes.
- By adding protrusion and depressions and by using Boolean operations, Direct Editing and Deformable Modeling tools.

No parent child dependencies = history FREE approach BricsCAD Mechanical does not create parent-child dependencies between geometric primitives and features. All changes made to your part are local; they do not force geometry regeneration. This history-FREE approach allows you to concentrate on your design instead of thinking how it can be changed in the future. **3D Constraints = Parametric** BricsCAD Mechanical provides a powerful toolset to make your part parametric by adding 3D constraints to its faces and edges and link them with global parameters using expressions and design tables.

Figure 3.158

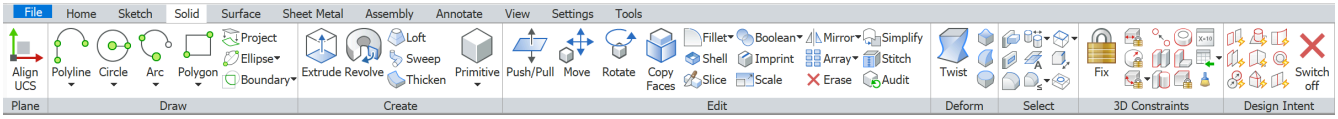


3.4.2 Sketching

Sketching toolset allows you to draw 2d profiles, which can be extruded or revolved to create 3d solids and surfaces or to add/subtract volume to/from existing entities.Tools to work with sketches are available on the **Sketch**tab of the **Mechanical** workspace Ribbon: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Sketc
Sketches are drawn in XY plane of the User Coordinate System (UCS) (1.15.18). By default, the dynamic UCS is enabled, which allows you to draw directly on faces your 2d solids and surfaces. You can also use Align UCS (6.30.2) tool to position UCS as you need. The drawing tools include familiar 2d primitives like a rectangle (6.26.6)/polygon (6.24.43), a circle (6.9.20)/arc (6.4.12)/ellipse (6.13.5), a polyline (6.24.27), a spline (6.27.65), and so on. You can arrange 2d elements using Move (6.21.30), Copy (6.9.50), 2d Rotate (6.26.34), 2d Mirror (6.21.18) and Array (6.4.16) tools. You can modify your sketches using Offset (6.23.3), Stretch (6.27.49), Trim (6.29.26), Extend (6.13.19), Fillet (6.14.7)/Chamfer (6.9.9), and Break (6.6.18) tools. You can make your sketch parametric using 2d constraints (1.15.7).

NOTE	By default, sketching entities are deleted after they wereextruded (6.12.13)/revolved (6.12.22) to create 3d solids/surfaces.To avoid that, setDELOBJ (1.18.1) (Delete Entity) system variable to 0 (never delete) or -1 (prompt before deletion).

Figure 3.159

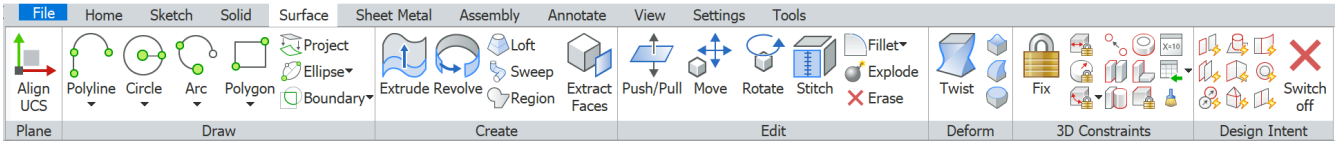


3.4.3 Solid Modeling

BricsCAD Mechanical provides a rich toolset for solid modeling, that includes traditional 3D modeling tools like Extrude or Subtract but also intuitive Direct Editing tools like; Push/Pull, Copy Faces and powerful 3D Constraints that allow to adjust the shape and size of your 3D part. 3D solids tools are available on the **Solid** tab of the **Mechanical** workspace Ribbon: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SolidModelingRibbon
On the Solid tab is where youcan find the sketching (3.4.2) tools as well as tools to create 3D solids from sketches:Extrude (2.4.11), Revolve (6.12.22), Loft (6.20.37), Sweep (6.27.60), andThicken (6.12.31). 3D solids can be created directly, with no underlying sketches using**solid primitives**: Box (6.6.17), Cylinder (6.9.60), Cone (6.9.63), Pyramid (6.24.58), Wedge (6.33.7), Sphere (6.27.64), Torus (6.29.22). All 3D solid can be edited using the one of following tools: Push/Pull (6.12.20), Move (6.12.17), Rotate (6.12.24), Copy Faces (6.12.8), Fillet (6.12.14)/Chamfer (6.12.4), which are applied to their faces and/or edges. Other editing tools that applied to 3D solids: Shell (6.27.39), Slice (6.27.36), Boolean operations (Union (6.30.6), Subtract (6.27.56), Intersect (6.17.19), Separate (6.27.39)), Scale (6.27.6), Mirror (6.21.19), Array (6.4.16). Erase (6.13.8) tool can be applied to the whole 3D solid as well as to a set of its faces forming a **feature** (such a fillet or a hole) and results in the deletion of the selected 3D solid/feature. Audit (6.12.2) tool is useful to find and fix errors in imported geometry. Simplify (6.12.27) tool allows you to automatically recognize **canonical** faces (planar, cylindrical, conical, spherical, toroidal) from spline faces of your 3D solid, which is often needed before applying 3D constraints. Stitch (6.12.28) tool allows to stitch disjoint surfaces and regions into one 3D solid, to form a watertight boundary. Deformable modeling toolset allow you to deform your 3D solid uniformly. It includes Twist (6.12.32), Move Point (6.12.11), Move Edge (6.12.10), and Transform Curve (6.12.9). Smart Selection (6.12.26) tools allow you to select faces and edges of your 3D solids based on particular geometric and topologic criteria - such as length, area, and smoothness. 3D Constraints (1.19.4) can be added to faces and edges of 3D solids to align and resize them, and will make your 3D part parametric (1.19.2). Many geometric 3D constraints like parallel/perpendicular, coaxial, tangent can be applied automatically using Design Intent recognition (1.18.3).

NOTE	BricsCAD is a history-free modeler; it does not create parent-child dependencies between features, nor regenerates geometry on parameter change. Full-blown parametric control can be embedded into any 3D part (including imported one) using 3D constraints (1.19.4).

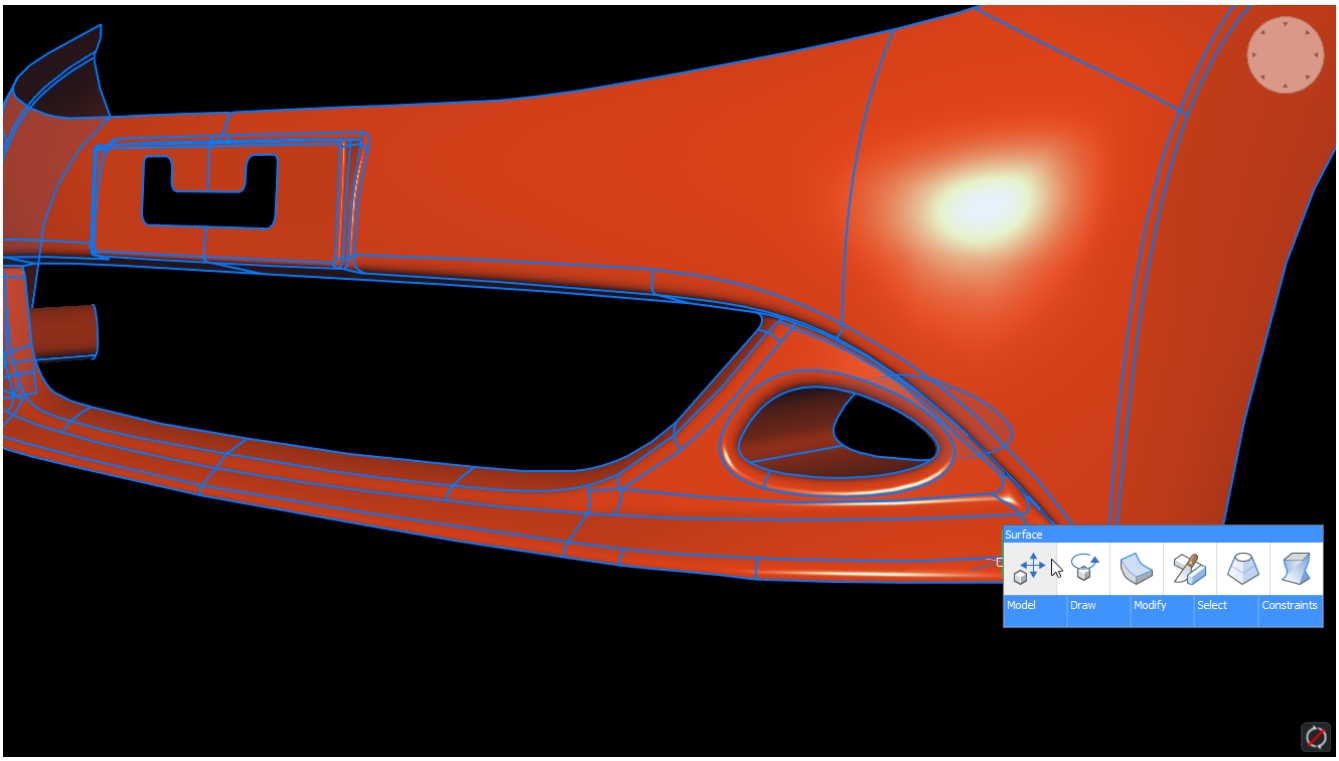
Figure 3.160



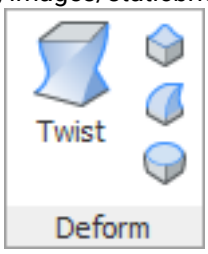
3.4.4 Surface Modeling

BricsCAD Mechanical provides a powerful toolset for surface modeling. Convert of 3D solid to surfaces and vice versa. Surface Modeling tools are available on the Surface tab of the **Mechanical** workspace Ribbon: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SurfaceModelingRibbon.png}) Solid modeling and Surface modeling tools are very identical, in editing (1.18.1) and applying 3D constraints (1.19.4). The only difference is that surface Extrude (6.13.21)/Revolve (6.26.30)/Loft (6.20.37)/Sweep (6.27.60) tools can create surface entities from closed 2D profiles and accept non-cloned profiles. Region tool (1.23.37) will create a region entity from a closed 2D profile and all regions can be stitched (6.12.28) together to create a larger surface. Some other ways to create a surface is to extract (6.34.4) one or more adjacent faces of a 3D solid/surface, and by exploding (6.13.13) a 3D solid into a set of surfaces and regions.

Figure 3.161



3.4.5 Freeform Modeling

Add complex distortions to your solid or surface models by using BricsCAD Mechanical deform tools to creating smoother, or more organic, looking shapes. Deform your solids and surfaces by moving, rotating and twistingtheir edges to change the geometryindependently with no design history. History-FREE approach allows you to concentrate on your design instead of thinking how it can be changed in the future. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/FreeformModeling.png}) **Freeform Modeling - Deform tools** 

- Move Edge (**DmDeformMove (6.12.10)**) - deforms faces by moving and rotating their edges.
- Move Point (**DmDeformPoint (6.12.11)**) - deforms the selected set of faces by moving a point.
- Transform Curve (**DmDeformCurve (6.12.9)**) – deforms faces by changing the geometry of their edges.
- Twist (**DmTwist (6.12.32)**) - deforms 3D object by twisting a portion around a given axis.

Tutorial: Use surface modeling and deformable modeling Click here to watch (1.6.35)

3.5 Sheet Metal - Tutorials

3.5.1 User interface

https://www.youtube-nocookie.com/embed/gl4Mqvn1JxY

TRANSCRIPT Welcome to BricsCAD Sheet Metal. Let's explore the user interface. The ribbon on the top of the screen gives you fast access to the most used tools. Many top-level tools offer submenus. Hover over the icon, left click and hold. The submenu will fly out to show the additional tools. Next, hover your cursor over any model element and the Quad cursor menu will appear. Move your cursor onto the Quad button. The Quad opens and shows you the tools that you can use on the selected element. The panel on the left side of your screen is the Mechanical Browser. It gives you access to sheet metal features and their properties in your model.

3.5.2 BricsCAD respects your design intent

https://www.youtube-nocookie.com/embed/jaPScAM1W_w

TRANSCRIPT During modeling operations, BricsCAD respects your design intent. When creating a flange, its initial position places a body above XY plane. It is like your design is laying on the table. The bottom face of the newly created flange is called a reference face. You can distinguish it with a slightly different color. The reference face maintains its position when you change its thickness. You can control the position of the flange with the Control hotkey. Your overall design intent of the initial flange size is maintained during the creation of other sheet metal features. As you can see here; the overall dimension of L-shape model is still 50. If the length of the vertical flange is 50 and we add a horizontal flange - overall vertical height of the model is 50 Let's switch to the second case. It is dimensioned polyline. While extruding the polyline, you see that all dimensions are kept. This is important for proper design.

3.5.3 Base flange creation

https://www.youtube-nocookie.com/embed/MkswHon9w8M

TRANSCRIPT Let's see how to create a sheet metal base flange. First, you will need a 2D profile and then use the flange base tool. In the control widget, this allows you to control the base flange position: up, middle or down. Use boundary detection for more complex 2D contour boundary selection. By moving the mouse cursor inside the contour, a special highlighting appears. The boundary is detected, and the flange base tool can be used. The thickness can be changed in the properties section of the mechanical browser.

3.5.4 What about edges

https://www.youtube-nocookie.com/embed/HSnYGhKqfr0

TRANSCRIPT By using the flange edge tool, a new flange can be easily created. Just, hover over an edge and use the tool. Length and angle can be specified during mouse cursor movement. Use the Tab hotkey to switch between fields. There are several other parameters which can be used during flange creation. For example; the width of the flange is controlled by using the width option of the tool. If the default radius of the bend is not suitable. It can be changed during creation with radius option. You can also create two flanges simultaneously. Use the taper angle option, if the side faces of the new flange are inclined.

3.5.5 2D polyline to Sheet Metal

https://www.youtube-nocookie.com/embed/IrYSaOp4FVw

TRANSCRIPT Using the Polyline tool is a great way to create Sheet metal bodies. Sheet metal extrude command can easily convert 2D polylines to a sheet metal body. Keeping with our design intent by avoiding possible intersections, the body was added to the outside of other components.

3.5.6 Lofted shapes

https://www.youtube-nocookie.com/embed/5VbGaY4H3nI

TRANSCRIPT BricsCAD can easily work with sheet metal lofted shapes. You can create a variety of different profiles using the sheet metal loft tool. When creating the lofted body, you can control thickness, corners fillet radius, and material position. Each lofted bend feature has the number of subdivision settings which controls the number of bends needed to produce the shape. By adding more subdivisions, will result in the part being smoother.

3.5.7 Working with hard edges

https://www.youtube-nocookie.com/embed/F9Rg_6PkJZU

TRANSCRIPT The model represented is a shell and is not manufacturable. To make the model manufacturable, we need to convert the hard edges. Create corner reliefs to avoid material conflicts. Hover over a flange, using the Quad, select the create relief tool. Convert all hard edges to a bend or a junction edge. Use the Quad cursor, it's fast and easy.

3.5.8 Working with shell

https://www.youtube-nocookie.com/embed/zlSfmSlcCyo

TRANSCRIPT In many cases, we are forced to shell a model to get our sheet metal body. In BricsCAD, the SmConvert command simplifies this, by automatically recognizes flanges, bends and lofted bends. During bending operation, small cuts corners are typically required because of the material conflict. The SmReliefCreate tool automatically creates reliefs on hard edges, bends, and corners for the entire model." To make the model unfoldable, some edges need to be converted to junctions. Use a window or one-by-one selection method to choose necessary edges. Convert the remaining hard edges to bends. Now it's time to generate the unfolded representation with manufacturing information. The unfolded representation of the model can be placed near the initial model.

3.5.9 Unfold a sheet metal body

https://www.youtube-nocookie.com/embed/v8si2_tiyUc

TRANSCRIPT Use the Quad to access tools to create a flat or unfolded representation of your model. For production, you can save your unfolding to a DXF file. The DXF file contains all necessary information, such as: bend positions, radiuses, angles of bending, overall dimensions, material and its thickness. Depending on the material and production tools, the length of unfolded representation may vary. Bend tables are a more reliable way to express material deformation properties and this information can be stored in CSV file, which can be attached to the model. After applying a bend table, you can see the size of unfolding has changed.

3.5.10 Associative unfolded representations

https://www.youtube-nocookie.com/embed/ZuGBEH5woes

TRANSCRIPT In some cases, modifications on sheet metal models can be quite difficult and not evident. The associative mode of sheet metal unfolding can help. In this mode, you can see folded and unfolded representations of your model. Two windows can be placed side-by-side. They have bi-directional associativity. Switching to the sheet metal features, changes the bends that automatically propagates to the unfolded state. Some changes can be done precisely only in unfolded representation. In the unfolded representation state allows you to use the direct editing tools for modifications. Switch to a folded model state to see modifications.

3.5.11 Working with reliefs

https://www.youtube-nocookie.com/embed/ldT1gAOKGIE

TRANSCRIPT Use Corner Reliefs to convert corners to circular, rectangular or V-type reliefs for production. Corner reliefs are built on corners which have three or more adjacent flanges. The size can be controlled by corner relief feature properties. Corner relief can be deleted or re-created if necessary. There are other types of corner relief. Use Bend reliefs to avoid material conflicts during bending operations and are created on the start and end of a flange edge. Apply rip, round or smooth shapes. The width of the bend relief is controlled by the properties. Bend and Corners reliefs can be applied to the manufacturing process.

3.5.12 Direct editing of a sheet metal model

https://www.youtube-nocookie.com/embed/i5oQPLg_qHk

TRANSCRIPT You can use BricsCAD Direct modeling functionality on your sheet metal bodies. For example: to cut a slotted hole, all you need is a 2D profile and use the direct editing extrusion feature. Unnecessary details of the model can be simply deleted. During Push/Pull operations, the flange thickness is maintained automatically. The corresponding angle of the existent flange can be changed, also. If cutouts are not needed anymore, use the window selections in combination with the erase tool to delete them.

3.5.13 Working with junctions

https://www.youtube-nocookie.com/embed/usewkcdZlOs

TRANSCRIPT Junction is a sheet metal feature where two adjoining flanges are met, but not connected. The junction has two types of connection: flanges or symmetric. The type can be switched by using Quad or in the properties of the mechanical browser. All bends can be changed to a junction or deleted. The junction gap is controlled by the value in the properties section of the mechanical browser. It can be changed individually for each junction feature or for the whole model. Junctions can be created in case of curved edges either.

3.5.14 Working with bends

https://www.youtube-nocookie.com/embed/XYGnRYflixU

TRANSCRIPT Bend features are important components of sheet metal design. Reordering bends and junctions can be useful for optimization and future manufacturing. Bends have several parameters, such as: radius which can be global, absolute or depend on material thickness. If the bend radius is too large, it can be substituted by a lofted bend for sequential bending operations. On the unfolded representation multiple bends appear. The number of sub-bends is controlled by the number of subdivisions property.

3.5.15 How to bend an existing flange

https://www.youtube-nocookie.com/embed/zywiRHgi3sg

TRANSCRIPT All existent flanges can be bent by using the flange bend tool. Using the Quad, launch the tool by hovering over the target flange. Draw or select a bend line and then specified which side to move. The bending angle is the final step. The target flange can have cutouts, also.

3.5.16 How to split a flange

https://www.youtube-nocookie.com/embed/8ZtFFrLcHFw

TRANSCRIPT Using the Split tool is an effective way to help in the unfolding of your sheet metal parts. The split tool makes a thin cutout to separate flange that creates a miter feature. Use the smart split tool to automatically find the closest corner. Use the control modifier to change the split between diagonal, vertical and horizontal. The miter properties control its size and position. Use a polyline to create a custom split path.

3.5.17 How to fill the gap in the corner

https://www.youtube-nocookie.com/embed/N4O8gvpoyTY

TRANSCRIPT In some cases, non-orthogonal connections of flanges corners are not actually closed. Use the Flange connect tool to close the corners. Select two side faces and the tool will connect flanges. The Flange connect tool does more than just 45 degrees angles. When creating junctions, use the Sheet Metal Imprint tool to extend to several side faces. This will add more edges to your model, giving you more control to modify.

3.5.18 Split lofted bend

https://www.youtube-nocookie.com/embed/QF92fDWMq_E

TRANSCRIPT Lofted bends are widely used when the bend radius must be significantly big. Following the goal of material usage and optimal nesting, you can split lofted bends. To make a split you just need to select the lofted bend and specify a point on it. The split will be created along a straight line through a given point.

3.5.19 Convert an imported sheet metal design

https://www.youtube-nocookie.com/embed/s2OxAL5PRCg

TRANSCRIPT BricsCAD Communicator can import 3D CAD data from all the major MCAD programs. After the import, converted the model to Sheet Metal bodies for unfolding. All import models are not alike. BricsCAD detects defects and has the tools to fix problems. In this model, the bends are not correct to unfold. Some are non-concentric, some have hard edge instead of an internal cylinder. Use the Repair command to convert the wrong bends to regular bends.

3.5.20 Creating parametric sheet metal model

https://www.youtube-nocookie.com/embed/Ur-dAzY0AtY

TRANSCRIPT With BricsCAD you can create parametric sheet metal models using 3D constraints. The Parametrize tool creates a consistent set of geometrical and dimensional constraints for a selected sheet metal part. You can easily change the size of the model with respect to all other dimensions. To modify the shape, choose a constraint and change its value.

3.5.21 Smart selection

https://www.youtube-nocookie.com/embed/g-PghoJ9Znk

TRANSCRIPT Use Smart Selection tool helps reduce the number of steps when modifying your model. With one click BricsCAD selects all hard edges in the model. Smart Selection tool also allows you to select faces and edges of your 3D solids based on geometric and topologic criteria.

3.5.22 Working with form features

https://www.youtube-nocookie.com/embed/6NiBVV805GM

TRANSCRIPT BricsCAD has a library of parametric Form Features, to access Form Features, open components panel. To use a Form Feature in the model just choose a necessary type and position it on a face. The parameters control the dimensions of each Form Feature such as length, width, height, and diameter. To position a form feature, dynamic dimensions can be used. Use the Replace tool to replace one Form Feature with another. Need to create an array of a Form Feature, you can with the Array option when inserting. An array of Form Features can be oriented along a custom direction, like along the edge.

3.5.23 Working with ribs by trajectory

https://www.youtube-nocookie.com/embed/IqkXNSWn4vo

TRANSCRIPT Rolled ribs provide additional strength to the sheet metal component. To create a Rib Feature, first draw the rib path and select the target flange and then click on the path. Ribs can have open and closed paths. The rib has two parameters: profile and round radius. All Ribs have associative curve paths and are updated automatically when its defining profile is changed. BricsCAD can identify ribs on imported models and use a curve path for modifications.

3.5.24 Assembly batch processing

https://www.youtube-nocookie.com/embed/8AQhDbw2e0A

TRANSCRIPT BricsCAD can process assemblies in batch mode. The result of batch processing is HTML report which gives the user all necessary information about sheet metal components in the assembly. The command validates each component for correctness by indicating poor sheet metal components and non-sheet metal components.

3.5.25 Sheet Metal context: browser, properties & settings

https://www.youtube-nocookie.com/embed/ZyzqdXD34Os

TRANSCRIPT When Sheet Metal context is initialized you see the L-shape icon near the root of the mechanical browser. Once Context is initialized the Sheet Metal properties section appears in the mechanical browser. This section contains all necessary Sheet Metal parameters, such as material thickness, bend radius, size of reliefs, K-factor and bend table. Sheet Metal settings can be expanded into four categories. The first is Commands, this is where the behavior of particular sheet metal command can be controlled. The second is Feature Colors, these are the colors of Sheet Metal features can be controlled. The third is the Initial Parameters section, this is where the values are located. The fourth is the Modeling section, which controls model behavior during modeling operations. Layers and Unfolding section controls the display representation of annotations by layers. Sheet Metal features colors can be turned ON or OFF by ribbon button.

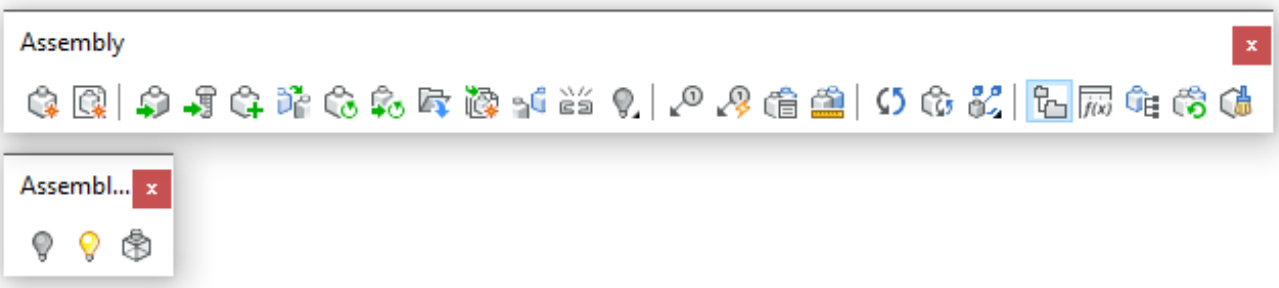
3.6 Assembly Design

3.6.1 Assembly Design Commands and Toolbars

Tools to work with mechanical components are available on the **Assembly** toolbar and its **Assembly Visualization** flyout toolbar, the **Assembly** menu and on the **Assembly** tab of the **Mechanical** workspace Ribbon:

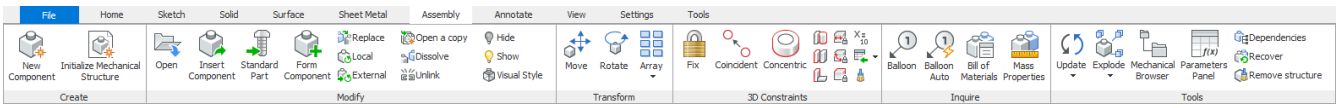
Toolbars (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/AssemblyD

Figure 3.162



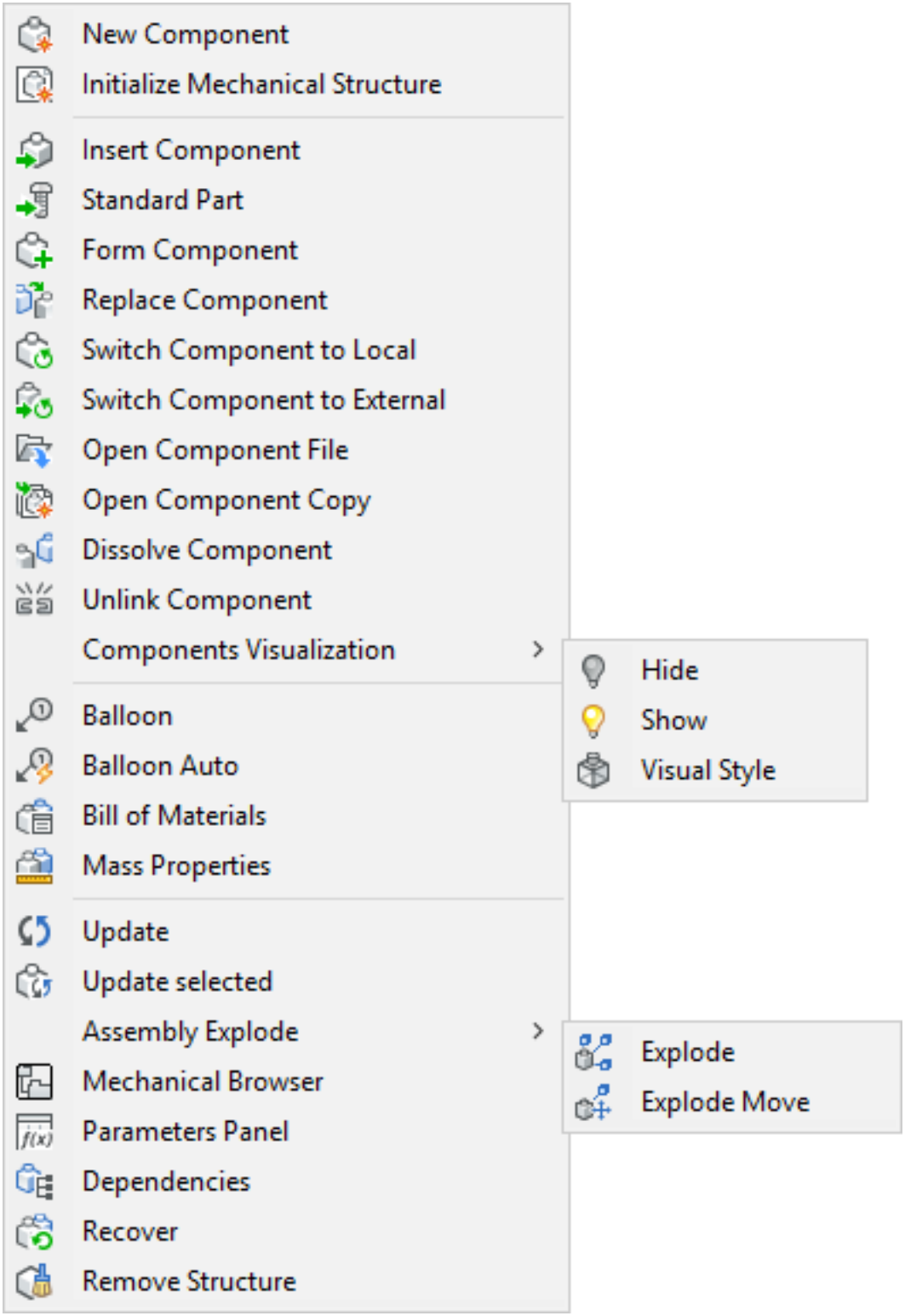
Ribbon (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/AssemblyDes

Figure 3.163



Menu (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/AssemblyDesi

Figure 3.164


New Component
Initialize Mechanical Structure
Insert Component
Standard Part
Form Component
Replace Component
Switch Component to Local
Switch Component to External
Open Component File
Open Component Copy
Dissolve Component
Unlink Component
Components Visualization
Hide
Show
Visual Style
Balloon
Balloon Auto
Bill of Materials
Mass Properties
Update
Update selected
Assembly Explode
Explode
Explode Move
Mechanical Browser
Parameters Panel
Dependencies
Recover
Remove Structure

3.6.2 Bottom-up and Top-down Assembly Design

Assembly modeling represent the structure of a designed product and consists of using components (1.22.22) to group geometric entities in hierarchies. Assembly modeling is commonly used in Mechanical Computer-Aided Design (MCAD) applications, it simplifies the design of complex products containing thousands or millions of geometric entities. The following terms are used across this topic:

- An **assembly** is a component that contains inserts of other components and a subassembly is separate assembly that is inserted into another assembly.
- A **part** is a component that does not contain inserts of another component.

An assembly with inserted components (which in turn can be assemblies with other inserted components) forms a tree that is called a mechanical structure of the 3D model. This tree is displayed in the Mechanical Browser (3.2.2). There are two main ways to design mechanical products: **top-down** and **bottom-up** approaches. You can also add mechanical structure to existing geometry, designed in BricsCAD or imported from another source. BmNew (6.8.16) command is called to create a new document for a new mechanical component. BmMech (6.8.15) command can be used to convert existing blocks and external references to components, while BmUnmech (6.8.24) does the opposite.

Bottom-Up Design

With bottom-up approach users start to design a product by detailing its low-level component. These components are then grouped into higher-level components. The process is continued until the very top component is corresponding to the product itself. The main command for bottom-up design is BmInsert (6.8.12), which is used to insert an existing component into the current drawing.

Top-Down Design

With top-down approach users start to design a product by defining but not detailing its first-level components. Each component is then refined in more details by defining its sub-components. The process is continued till all low-level components are defined and detailed. BmForm (6.8.9) command creates a new subcomponent for an existing component.

Adding Mechanical Structure to Existing Geometry

BricsCAD users can add component hierarchy to any existing drawing created in BricsCAD or imported from a different CAD format. BmForm (6.8.9) converts a selected set of geometric entities to a subcomponent of the current drawing.

Changing Mechanical Structure

During design process users may realize that a particular component should be moved up or down in the component hierarchy. BmForm (6.8.9) command forms a new subcomponent for the current drawing that will contain a selected set of a component. BmDissolve (6.8.5) deletes a component and moves all its subcomponents to the level of the dissolved component.

Using Standard Components

Some of the standard components are: screws, nuts, washers, bearings and holes. BricsCAD supplies a huge library of standard components, more than 30,000 details in a variety of sizes. Users can significantly increase design performance by using standard parts. Use the Components panel (6.9.61) to insert standard components into your parts and assemblies.

Assembly Constraints

Assembly constraints are ordinary 3D Constraints (1.19.4), that link faces and edges of bodies to represent the components of an assembly. You can link 3D Constraints (1.19.4) at the top-level and at the subcomponents level.

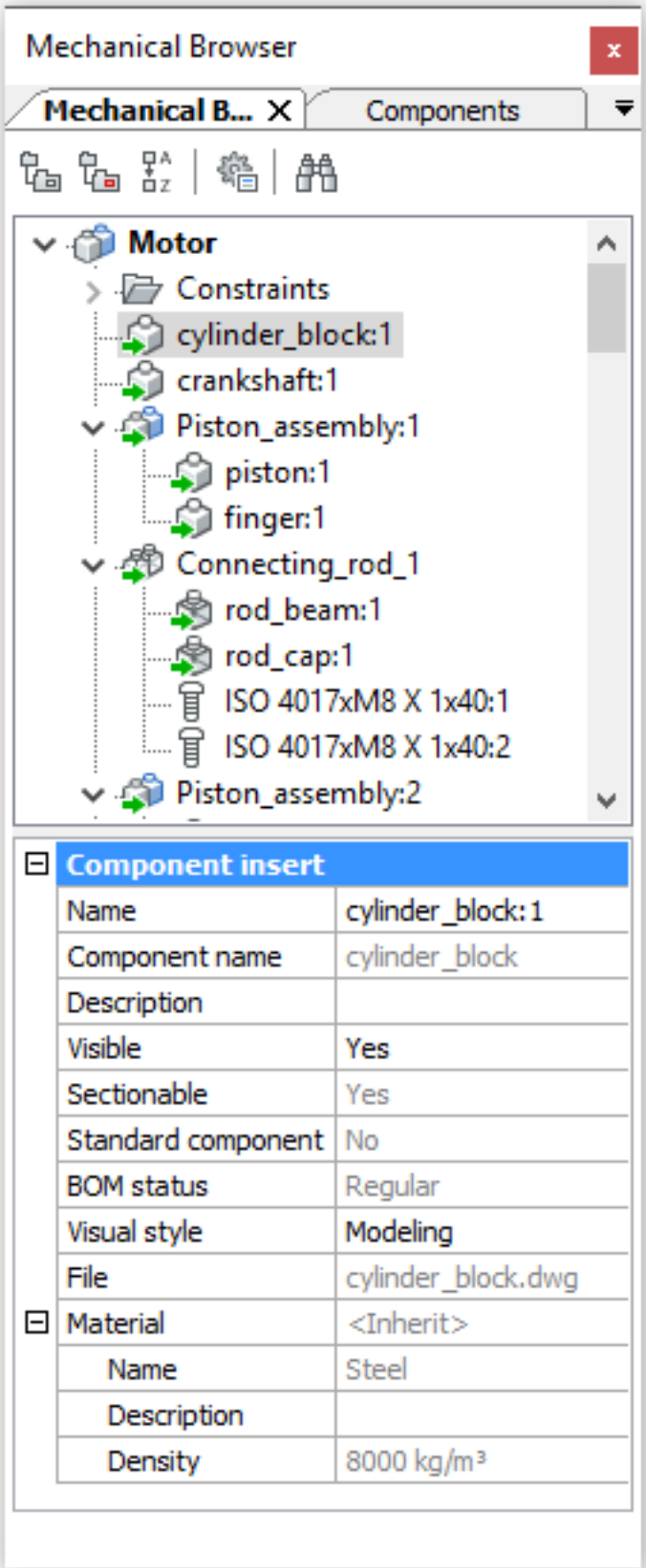
Component Visibility

A component within a complex assembly may seem invisible due to other components surrounding it. For example, a piston is invisible inside a cylinder block. To make such a component visible user may need to hide the other components surrounding it. Users can hide any subcomponent inserted into the current drawing or any subcomponents, on any level of nesting. You can also make a component transparent and/or visualize it as a wireframe model. BmHide (6.8.11) hides a particular subcomponent. BmShow (6.8.22) unhides a hidden subcomponent.

3.6.3 Component Properties

Component properties and context menus are in the Mechanical Browser (6.8.29). Component context menus are displayed by right mouse click on the component, and component properties are displayed automatically when selecting a component in the Mechanical Browser. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mech
Components can be: Main, Insert and Subcomponent. **Icons**

Figure 3.165


Mechanical Browser
Mechanical B...
Components
Motor
Constraints
cylinder_block:1
crankshaft:1
Piston_assembly:1
piston:1
finger:1
Connecting_rod_1
rod_beam:1
rod_cap:1
ISO 4017xM8 X 1x40:1
ISO 4017xM8 X 1x40:2
Piston_assembly:2
Component insert
Name
cylinder_block:1
Component name
cylinder_block
Description
Visible
Yes
Sectionable
Yes
Standard component
No
BOM status
Regular
Visual style
Modeling
File
cylinder_block.dwg
Material
<Inherit>
Name
Steel
Description
Density
8000 kg/m³

Rendered visual style

Wireframe visual style

Hidden Component that does not contain subcomponents * * * Component that contains subcomponents * * * Standard Part Component * External components have a green arrow added: or

Context Menus

Main Component context menu

Right click the main component name: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360018456

- Update : Updates the hierarchy of mechanical components for the current drawing in case referenced drawing files of sub-components have been modified.
- Visual Style : All by Viewport: Applies the current Visual Style (6.31.13) to all components in the assembly (see the ShadeMode (6.27.29) command). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/
- Switch all to local : Switches all components to internal components (see the BmLocalize (6.8.13) command).
- Switch all to external : Switches all components to external components (see the BmExternalize (6.8.8) command).
- Add new parameter : Creates a new parameter (6.24.5) in the **assembly.**
- Create exploded view (6.8.6) : creates a block with an exploded representation of the current assembly
- Collapse all : collapses the main component and all components and subcomponents.
- Expand all : expands the main component and all components and subcomponents.

Component context menu

Right click a component name. External components context menu (left), local components context menu (right): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse

- Open: Opens the referenced drawing (see the BmOpen (6.8.17) command).
- Open a copy: Opens a copy of a component insert as a new drawing (see the BmOpenCopy (6.8.18) command).
- Update: Reloads all referenced components from external files and updates BOM tables (see the BmUpdate (6.8.25) command).
- Replace: Replaces a component insert (see the BmReplace (6.8.21) command). Replacing a local insert turns it into an external insert.
- Replace all inserts: Replaces all inserts that refer to the same source (see the BmReplace (6.8.21) command). Replacing local inserts turns them into external inserts.
- Switch to local: Switches an external component to an internal component (see the BmLocalize (6.8.13) command).
- Switch to external: Switches an internal component to an external component (see the BmExternalize (6.8.8) command).
- Set material to component... : Assigns a physical material to local component.
- Removes material from component... : Removes the physical material definition from local component.
- BOM status: Parameter to control the appearance of the component in BOM tables (6.8.2)
- Dissolve: Dissolves a mechanical component inserted in the current drawing (See the BmDissolve (6.8.5) command).
- Unlink: Breaks the connection between a component insert (e.g. a Window) and a 3D solid (e.g. a Wall) retaining the opening.
- Hide ( ) / Show ( ): Hides or shows the selected component.
- Exclude all inserts from section: Sets the **Sectionable** property of all similar inserts to **NO**. Defines whether an insert is affected by the ViewSection (6.31.10) command.
- Visual Style: Displays the **Visual Style** menu. Visual Styles (6.31.13) that are saved in the current drawing are available. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/
- By Viewport: Renders the selected component according to the current viewport shademode

Subcomponent context menu	External subcomponents context menu (left), local subcomponents context menu (right): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse • Open: Opens the referenced drawing (see the BmOpen (6.8.17) command) • Open a copy: Opens a copy of a component insert as a new drawing (see the BmOpenCopy (6.8.18) command). • Hide ( ) / Show ( ): Hides or shows the selected subcomponent. • Exclude all inserts from section: Sets the **Sectionable** property of all similar inserts to **NO**. Defines whether an insert is affected by the ViewSection (6.31.10) command. • Visual Style : See Component Visual Style (6.8.29). • Zoom to: Zooms to the selection set. • Collapse all: collapses the main component and all components and subcomponents. • Expand all: expands the main component and all components and subcomponents.
Component parameter context menu	Right click a parameter: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse • Link to parameter: Establishes a link between one or more component parameters and assembly parameters. • Collapse all: collapses the main component and all components and subcomponents. • Expand all: expands the main component and all components and subcomponents.

Properties

Main component properties	Click the main component name: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse • Name: Type a name for the main component. By default the name of the main component equals the file name. • Description: Optional description of the main component. • File: Shows the path and name of the dwg. • Extension type: Allows to identify the component as a BIM Component or a Sheet Metal Feature. • Insert as: Select either Internal Component or External Component. • Sectionable: Defines whether the component is affected by the **ViewSection** command. • Standard component: defines whether this component standard or not. • BOM status:defines behavior of this component in BOM table. • Material: Material of the main component. The **Name**, **Density**, and the optional **Description** are defined in the **Physical Materials** dialog box. Select the **Material** node, then do one of the following: • Click the **Browse** button ( ) to open the **Physical Materials** dialog and choose a material in the **Project** or **Library** list. The **Density** property of the selected material is used by the BmMassProp (6.8.14) command. • Click the **Delete** button ( ). The **Material** field reads **<Inherit>**. When the component is inserted in an assembly, the **Material** definition is copied from the main component of the assembly.

Component insert properties	Click the component name: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse • Name: Type a name for the component. By default the name of the component equals the source file name. • Component name: Name of the component as defined in the source file. • Description: Description of the component as defined in the source file. • Visible: Allows to show or hide the component in the assembly. Click the **Visible** field, then click the down-arrow button and choose **Yes** or **No**. or Double click the **Visible** field to toggle the visibility of the component. • Sectionable: Defines whether the component is affected by the ViewSection (6.31.10) command. • Standard component : defines whether this component standard or not. • BOM status:defines behavior of this component in BOM table. • Visual style: Allows to control the visual style for the component. By default the visual style is inherited from the viewport. Click the Visual style field, then click the down-arrow button and choose a visual style in the list. • File: Shows the path and name of the referenced dwg. • Material: Shows the material of the component. If the in the material the source file is set to <Inherit>, the material is copied from the main component. If the material is defined explicitly in the source file, this material is used.

Subcomponent properties	Click the subcomponent name: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse • Name: Name of the component as defined in the source file of the parent component. • Component name: Name of the component as defined in the source file. • Description: Description of the component as defined in the source file. • Visible: Allows to show or hide the component in the assembly. Click the **Visible** field, then click the down-arrow button and choose **Yes** or **No**. or Double click the **Visible** field to toggle the visibility of the component. • Sectionable: Defines whether the component is affected by the ViewSection (6.31.10) command. • Standard component : defines whether this component standard or not. • BOM status :defines behavior of this component in BOM table. • Visual style: Allows to control the visual style for the component. By default the visual style is inherited from the viewport. Click the Visual style field, then click the down-arrow button and choose a visual style in the list. • File: Shows the path and name of the referenced dwg. • Material: Shows the material of the subcomponent. If the in the material the source file is set to <Inherit>, the material is copied from the main component or the parent component. If the material is defined explicitly in the source file, this material is used.
Selection Methods	
Select a component in the model.	The component highlights in the Mechanical Browser.
Click a component in the Mechanical Browser	The component highlights in the drawing. Use Space button to select component.

Figure 3.166

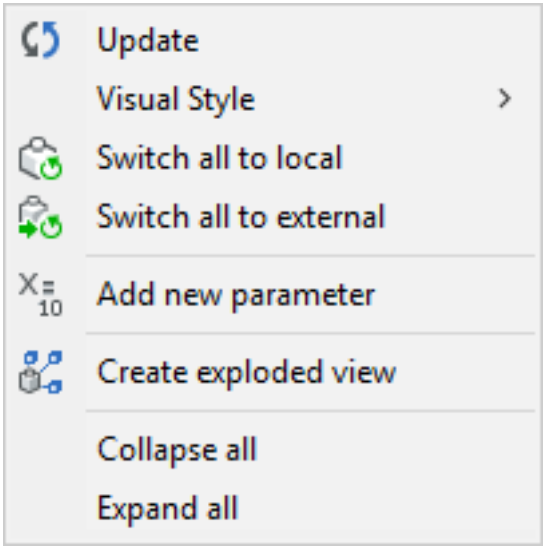


Figure 3.167



Figure 3.168

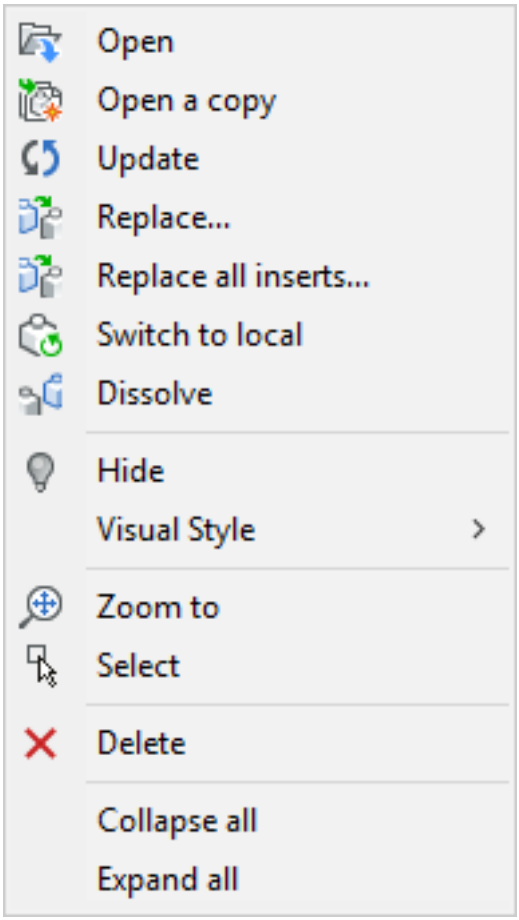


Figure 3.169

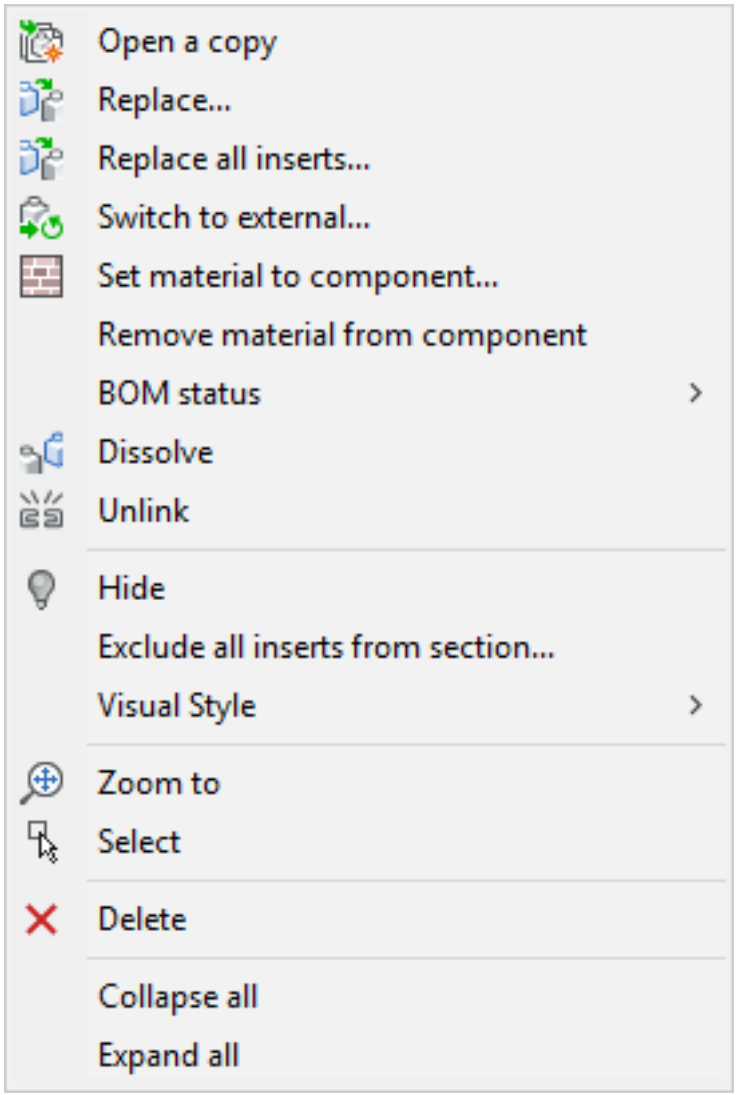

Open a copy
Replace...
Replace all inserts...
Switch to external...
Set material to component...
Remove material from component
BOM status
Dissolve
Unlink
Hide
Exclude all inserts from section...
Visual Style
Zoom to
Select
Delete
Collapse all
Expand all

Figure 3.170

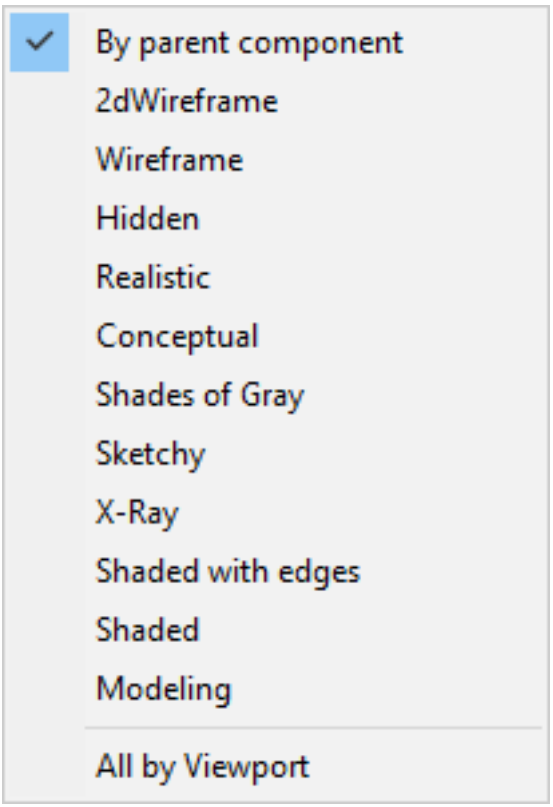


Figure 3.171

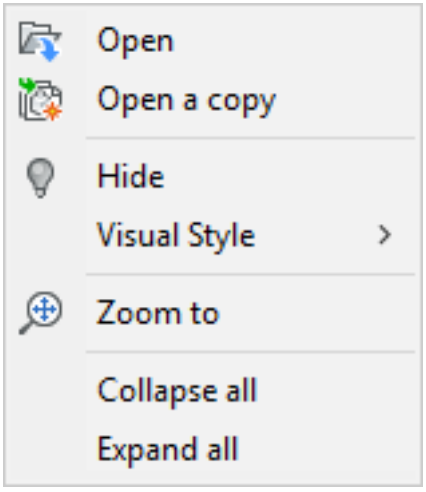


Figure 3.172

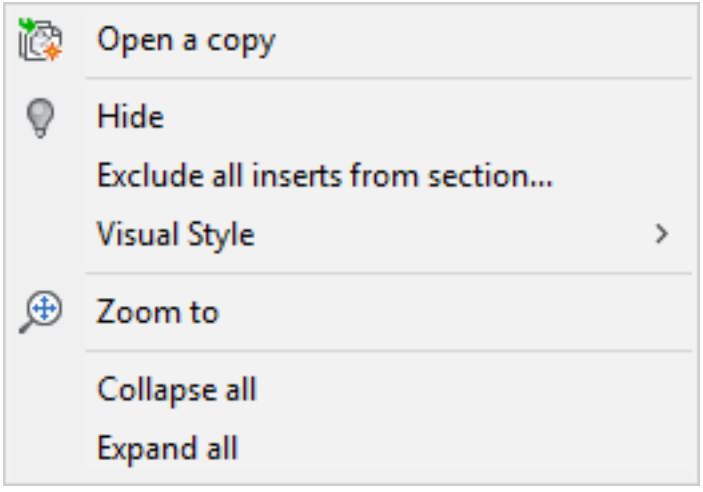


Figure 3.173

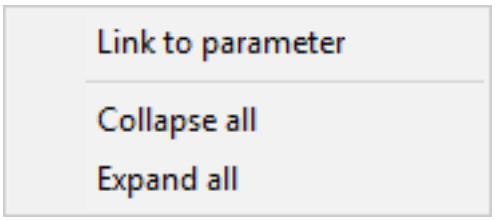


Figure 3.174

Component	
Name	Motor
Description	
File	D:\BricsysHelp\V8_engine\Engine
Extension type	
Insert as	External component
Sectionable	Yes
Standard component	No
BOM status	Regular
Material	Steel
Name	Steel
Description	
Density	8000 kg/m³

Figure 3.175

Component insert	
Name	Connecting_rod_assembly:3
Component name	Connecting_rod_assembly
Description	
Visible	Yes
Sectionable	Yes
Standard component	No
BOM status	Regular
Visual style	By parent component
File	Connecting_rod_assembly.dw
Material	<Inherit>
Name	Steel
Description	
Density	8000 kg/m³

Figure 3.176

Component insert	
Name	rod_beam:1
Component name	rod_beam
Description	
Visible	Yes
Sectionable	Yes
Standard component	No
BOM status	Regular
Visual style	By parent component
File	rod_beam.dwg
Material	<Inherit>
Name	Steel
Description	
Density	8000 kg/m³

3.6.4 Arrays of Components

Commands: ARRAY (6.4.16), ARRAYRECT (6.4.25), ARRAYPOLAR (6.4.23), ARRAYPATH (6.4.21), BMINSERT (6.8.12)
BricsCAD allows you to insert the same component multiple times in a regular way: all inserts can be distributed uniformly (at the same distance) along a line, a circle, or an arbitrary curve. Higher-order patterns (for example, rectangular or cylindrical) are also supported. This is achieved by using the technology of **associative arrays:**rectangular, polar, and path. Associative arrays are persistent entities, which can be edited (manually or automatically) at any moment after creation - to change the number of elements or distance between them. All arrays in your model are listed in the Mechanical Browser, where they can be edited. Any component can be arrayed - including parts, subassemblies, standard parts, and holes.

Creating an Array of Components

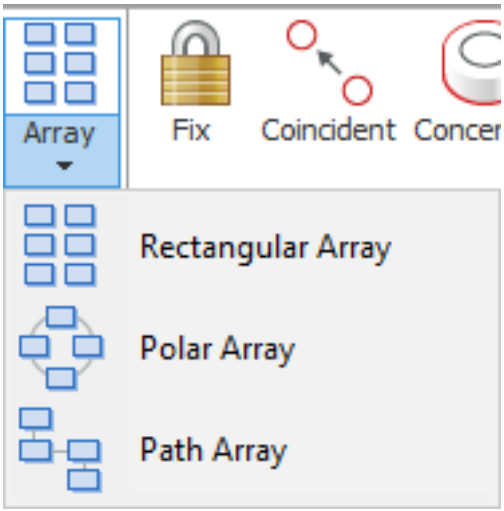
Use one of the array tools available in Assembly ribbon panel of Mechanical profile:

- Rectangular Array (6.4.25)
- Polar Array (6.4.23)
- Path Array (6.4.21)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/ArraysComponentArra
Another way to create an array of component is to use Multiple/Array option of Insert Component (6.8.12) tool:



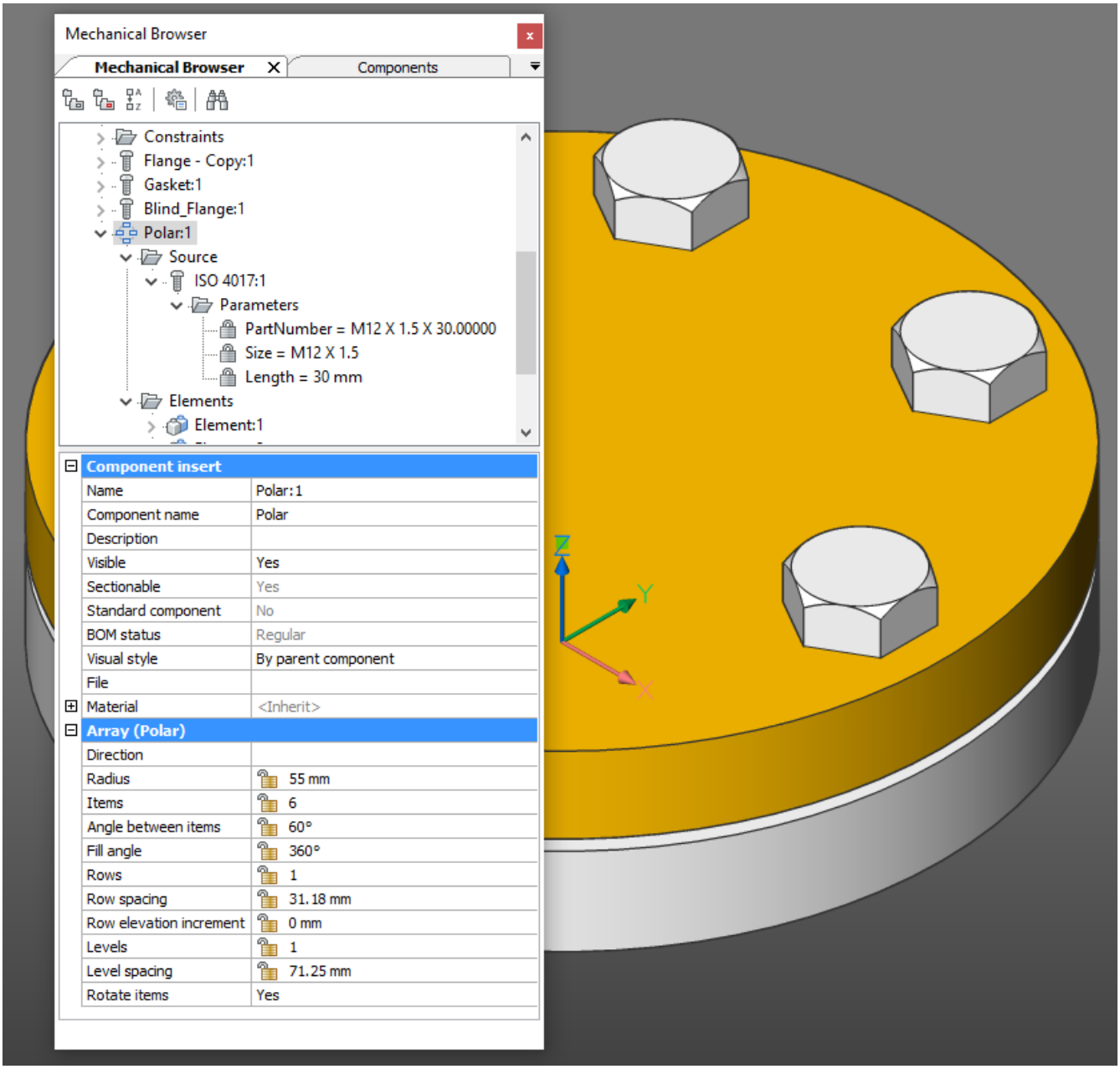
Figure 3.177



Arrays of Components in the Mechanical Browser

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/ArraysComponentArra
Arrays of components are represented as special nodes in the Mechanical Browser: they are named after the array type (Rectangular, Polar, or Path) followed with semicolon and the index of the array (for example, **Rectangular:1**, **Polar:2**, **Path:3**). These nodes have the sameproperties (3.6.5) as regular associative arrays. The properties can be used to change the number of elements in the array, distance/angle between them, and other array parameters. You can also assign a persistent expression to any property - it will be used to automatically recompute the value of the property upon changing its driving parameters. Each array node has two child nodes: **Source** and **Elements**. Under Source node you can find the inserted component and its parameters, while Elements node can be expanded into a list of array elements with the possibility to hide or change visual style of a particular element inside the array.

Figure 3.178


Mechanical Browser
Mechanical Browser
Components
Constraints
Flange - Copy:1
Gasket:1
Blind_Flange:1
Polar:1
Source
ISO 4017:1
Parameters
PartNumber = M12 X 1.5 X 30.00000
Size = M12 X 1.5
Length = 30 mm
Elements
Element:1
Component insert
Name Polar:1
Component name Polar
Description
Visible Yes
Sectionable Yes
Standard component No
BOM status Regular
Visual style By parent component
File
Material <Inherit>
Array (Polar)
Direction
Radius 55 mm
Items 6
Angle between items 60°
Fill angle 360°
Rows 1
Row spacing 31.18 mm
Row elevation increment 0 mm
Levels 1
Level spacing 71.25 mm
Rotate items Yes
Z
Y
X

Arrays of Components and Bill of Materials

Arrays of components are transparent for product structure: components inside arrays are considered as inserted directly into the same assembly. So in a top-level BOM components inserted into top-level arrays are listed as a top-level components. The **Quantity** column in BOM is filled with the total number of inserts of the same component including the components inside arrays.

Arrays of Parametric Components, Standard Parts, and Holes

If a parametric component is inserted as an array, its parameter are still available for editing and linking with global parameters. Technically the parameters are associated with the source element of an array, and when this source element is edited parametrically, all other elements of the array are changed after it. Standard Parts and Holes are parametric components and you can edit their parameters inside the arrays. You can easily create a part with many holes and you can control the hole number, positions, and size parametrically plus insert a parametric bolt. This powerful technology allows you to create parametric flange assembly with multiple configurations.

3.6.5 Array Properties

Component array is an associative array made of a parametric component. Component arrays have special representationin the Mechanical Browser. (6.8.29) Component array have two main subnodes:

- Source -node that contains source object and its parameters. Source object can be presented as acomponent or subassembly with their own properties (3.6.3).
- Elements - node that contains all insstances of source object

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/ArrayPropertiesMechB
Component array context menu displayed by right mouse click on the array node while array (6.4.16) properties displayed automatically when selecting an array in the Mechanical Browser.

Context Menu

Component array context menu

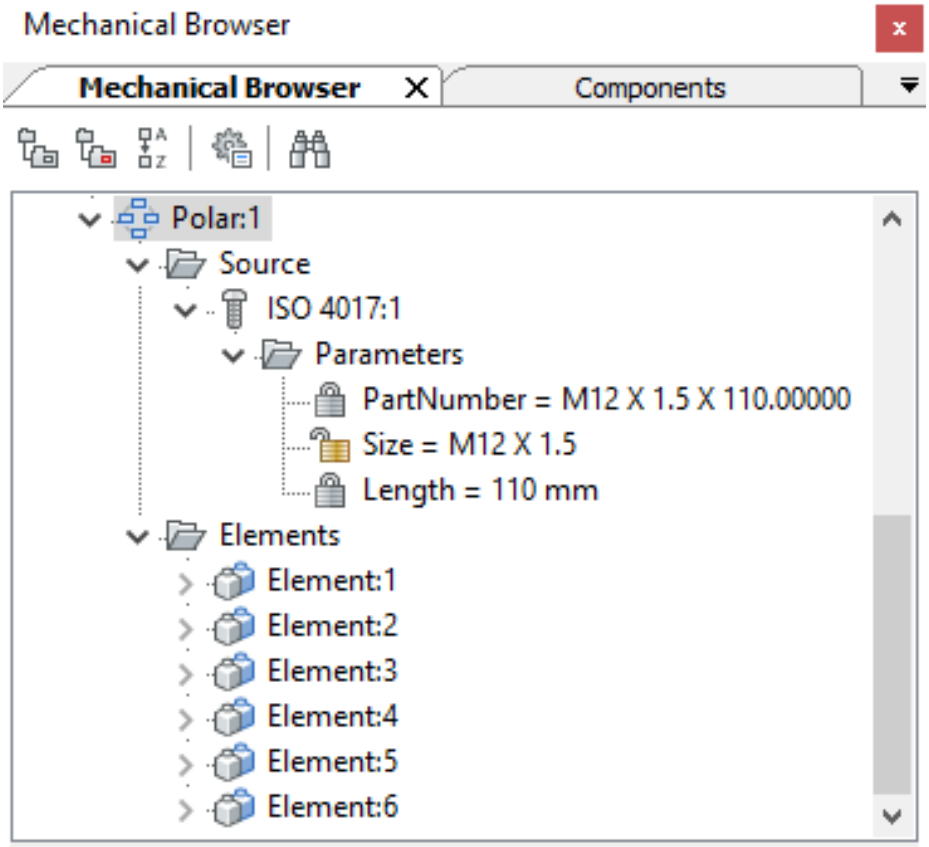
Right click an array: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse

- Dissolve: Dissolves an array created in the current drawing (See the BmDissolve (6.8.5) command).
- Unlink: Unlinks an array from 3D solid by dissolving the underlying component-based feature.
- Hide ( ) / Show ( ): Hides or shows the selected array.
- Exclude all inserts from section: Sets the **Sectionable** property of all similar inserts to **NO**. Defines whether an insert is affected by the ViewSection (6.31.10) command.
- Visual Style: Displays the **Visual Style** menu. Visual Styles (6.31.13) that are saved in the current drawing are available.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse

- By Viewport: Renders the selected array according to the current viewport shademode (see the ShadeMode (6.27.29) command).
- All by Viewport: Renders all arrays according to the current viewport shademode (see the ShadeMode (6.27.29) command).
- Zoom to: Zooms to the selection set.
- Select: Selects the array.
- Delete: Removes the selected array from the assembly.
- Collapse all: collapses the main component and all components and subcomponents.
- Expand all: expands the main component and all components and subcomponents.

Figure 3.179



Component insert	
Name	Polar:1
Component name	Polar
Description	
Visible	Yes
Sectionable	Yes
Standard component	No
BOM status	Regular
Visual style	By parent component
File	
Material	<Inherit>
Array (Polar)	
Direction	
Radius	55 mm
Items	6
Angle between items	60°
Fill angle	360°
Rows	1
Row spacing	30 mm
Row elevation increment	0 mm
Levels	1
Level spacing	120 mm
Rotate items	Yes

Related Commands

3dArray (6.3.2) - creates 3D arrays. -Array (6.4.15) - creates a polar or rectangular array of entities; displays prompts at the command bar. ArrayClassic (6.4.17) - creates a static polar or rectangular array of entities through a dialog box. ArrayEdit (6.4.20) - allows editing associative array entities and their source entities. ArrayPath (6.4.21) - associatively distributes entity copies evenly along a path into multiple rows, and levels. ArrayPolar (6.4.23) - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. ArrayRect (6.4.25) - associatively distributes entity copies into any number of rows, columns, and levels. MInsert (6.21.17) - inserts arrays of blocks.

Figure 3.180

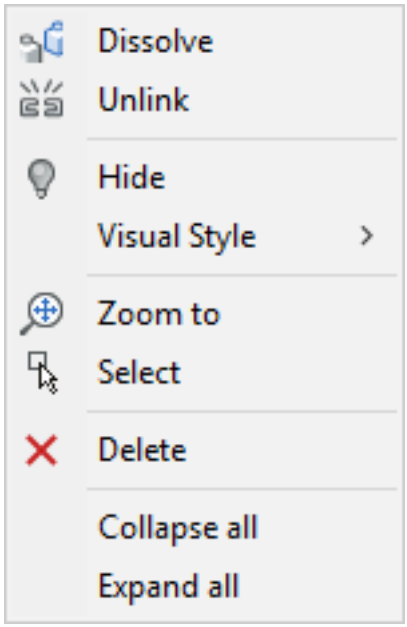
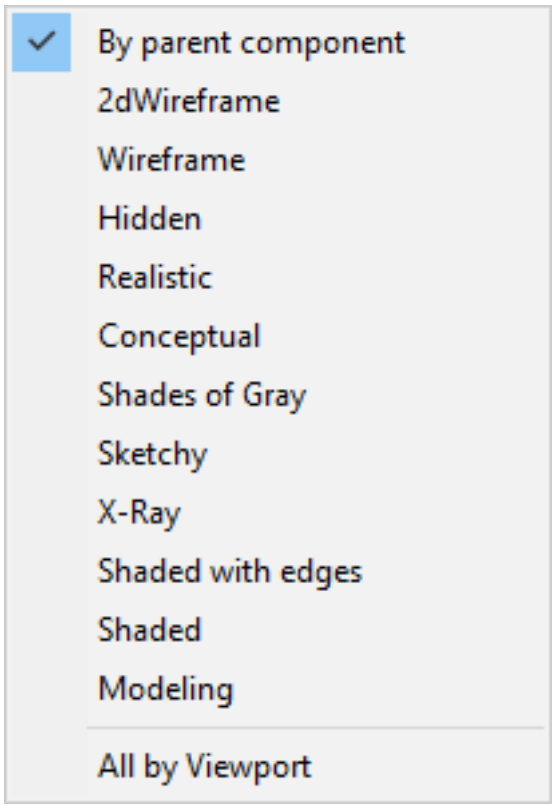


Figure 3.181



3.6.6 Bill Of Materials

Command: BMBOM (6.8.2) Engineering Bill of Materials (BOM) is a list of unique mechanical components used in a drawing. The list is represented as a BricsCAD table entity (see Tables (1.21.25)) and can be placed at any location in the drawing area. Each entity has its own serial number (starting with 1), name, and the number of occurrences.

Bill of Materials types

There are three difference ways to list Bill of Materials in BricsCAD. The difference between these is the way in which components are listed.

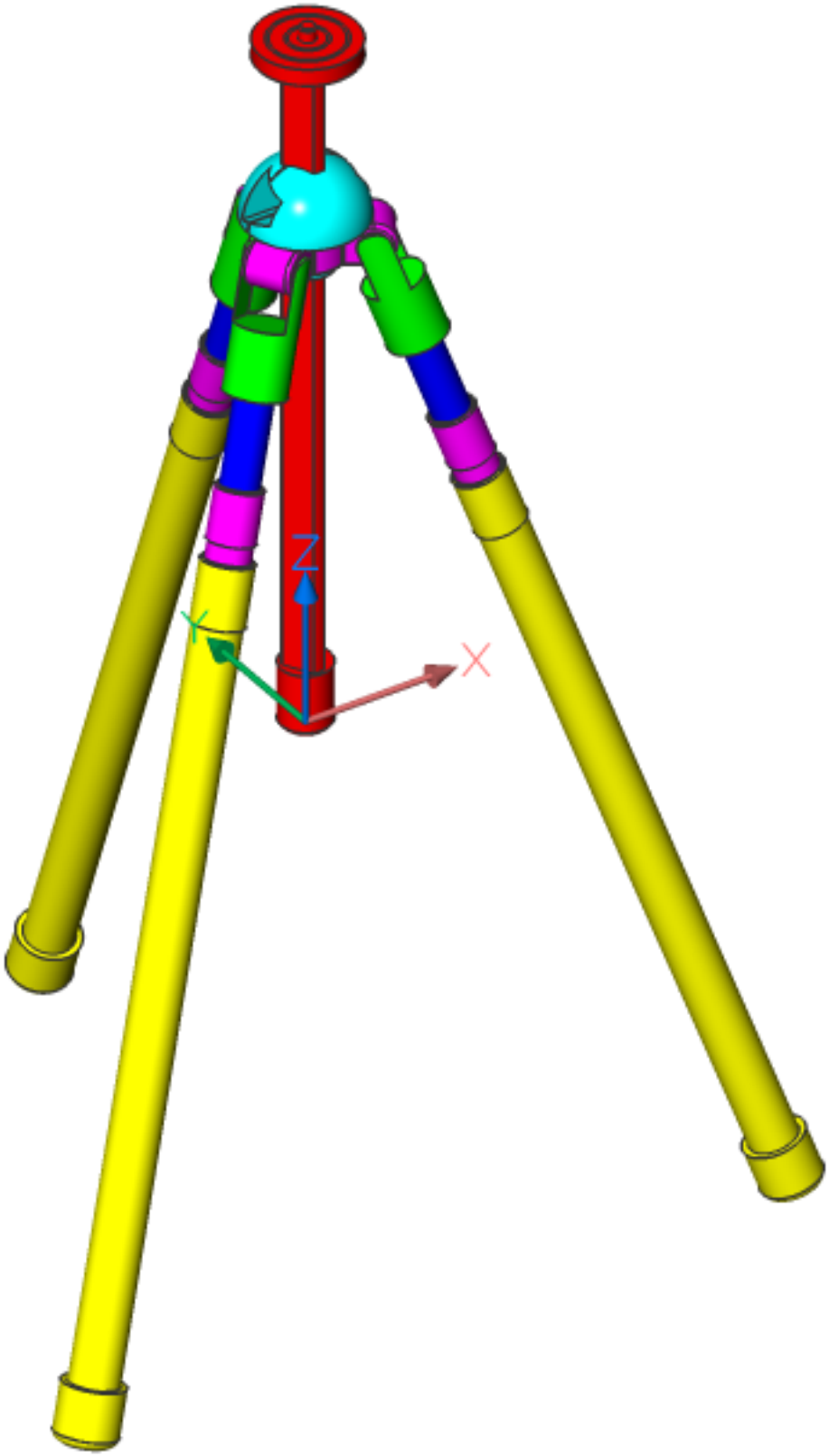
- **Top level** lists top-level components only.Top level components are components inserted directly into the main assembly. This type of BOM lists only main components, without going into the details of what those components consist of.
- **Bottom level**listsbottom-level (parts) only.Bottom level components are terminal parts, which do not contain other parts or subassemblies. This type of BOM lists all parts in the main assembly, regardless of what component those parts belong to.
- **Hierarchical**lists all components so that, for each component has its subcomponents listed directly below. In the hierarchical mode, serial numbers for subcomponents will be formed as a sequence of numbers separated by dots (for example, ”1.2.3”). This type of BOM allows users to see the assembling hierarchy in the Bill of Material table.

NOTE	It is possible to control which parts should be included in the Bill of Material using the**BOM status** property. The level of a component is deduced in BOM with respect to the **BOM status**of all related components. For example, a component with its BOM status set to**Terminal**will be always considered bottom-level even if it contains other parts or subassemblies.

Example: Tripod assembly (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDes

Type	**Bill of Material**
Top level	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse
Bottom level	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse
Hierarchical	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse

Figure 3.182



BOM columns

By default, the BOM table contains only three columns: number of a component, its name, and the number of inserts of this component in the assembly. The Configure option of the BmBom command gives the option to add more columns; Description, Density, Volume, Mass, Material, Thickness (for sheet metal parts) and Parameters (for inserts of parametric components (1.22.22)). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanic
BOM with Density, Volume, and Mass. Multiple inserts of the same parametric component (1.22.22) are grouped in separate BOM rows depending on the values of their parameters. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

Figure 3.183

Bill of Materials tripod_main_assy					
No.	Component	Quantity	Density	Volume	Mass
1	tripod_axis_assy	1	8000.00 kg/m³	330.77 cm³	2.65 kg
1.1	tripod_axis_prt	1	8000.00 kg/m³	194.42 cm³	1.56 kg
1.2	tripod_crosspiece_prt	1	8000.00 kg/m³	60.06 cm³	0.48 kg
1.3	tripod_sphere_prt	1	8000.00 kg/m³	76.29 cm³	0.61 kg
2	tripod_foot_assy	3	8000.00 kg/m³	166.24 cm³	1.33 kg
2.1	bond_assy	3	8000.00 kg/m³	21.32 cm³	0.17 kg
2.1.1	tripod_bond_prt	3	8000.00 kg/m³	21.32 cm³	0.17 kg
2.2	tubes_assy	3	8000.00 kg/m³	144.92 cm³	1.16 kg
2.2.1	lower_tube_prt	3	8000.00 kg/m³	62.85 cm³	0.50 kg
2.2.2	middle_tube_prt	3	8000.00 kg/m³	34.83 cm³	0.28 kg
2.2.3	upper_tube_prt	3	8000.00 kg/m³	47.24 cm³	0.38 kg

Figure 3.184

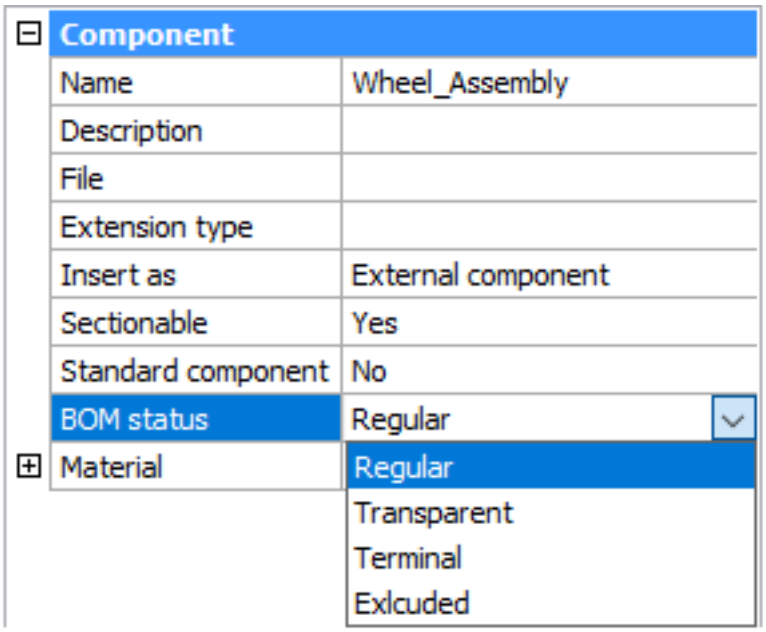
Bill of Materials Bookshelf					
No.	Component	Quantity	L	T	W
1	Frame Board	2	360.0	20.0	200.0
2	Frame Board	2	700.0	20.0	200.0
3	Shelf Board	2	266.7	2.5	200.0
4	Shelf Board	2	466.7	2.5	200.0

BOM status

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/BOMStatus.png}) Each component has a **BOM status** property, which can be used to control the behavior of the component in **Bill of Materials** tables.This propertymay take one of the following values:

- **Regular** - the component will appear in the Bill of Materials with respect of its position in the assembly structure and the selected mode for the Bill of Materials - default value.
- **Transparent**- the component itself will not appear in the Bill of Material, but their subcomponents may appear in the BOM, depending on their BOM status, positions in the assembly structure and the mode of the BOM table. Effectively, it means that the component acts as a transparent container for its subcomponents.
- **Terminal** - the component will be treated as a leaf component (that is, having no subcomponents) regardless of the actual presence of the subcomponents. This is useful when modeling a complex part or a standard component.
- **Excluded** - neither the component itself nor its subcomponents will appear in the Bill of Materials.

Figure 3.185



To set BOM Status of an external or root component

1. Open the file containing the component.
2. Select the root component in theMechanical Browser (6.8.29).
3. Set the **BOM status** property to the required value.

To set BOM Status of a local component

1. Select the component in theMechanical Browser (6.8.29).
2. Right click on the selected node to open the context menu.
3. Select the required BOM status from theBOM Status submenu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h

NOTE	For local components inserted into an external component, it is not possible to set the BOM status via the context menu. To do that, it is necessary to open the component that directly contains the required local component.

Figure 3.186

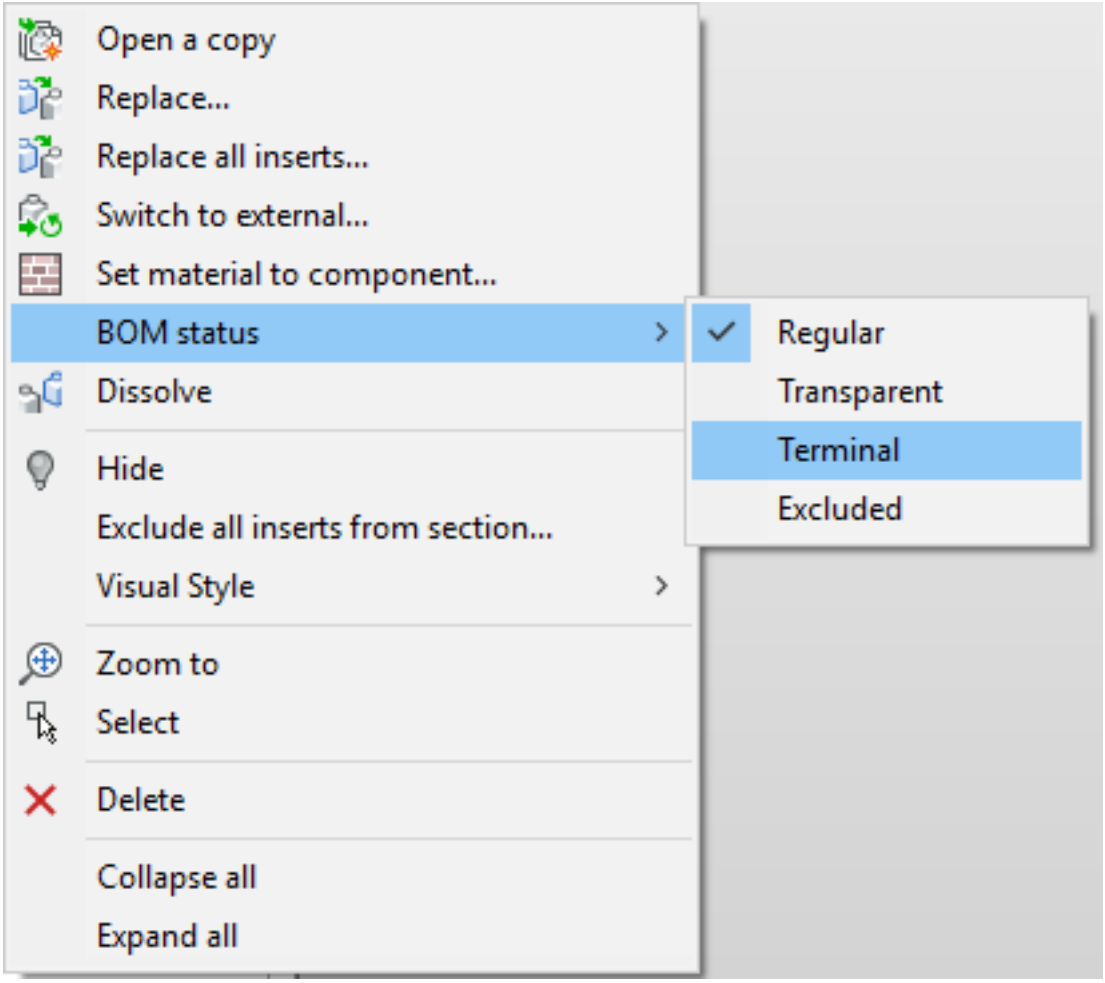


Figure 3.187

Bill of Materials tripod_main_assy		
No.	Component	Quantity
1	tripod_axis_assy	1
2	tripod_foot_assy	3

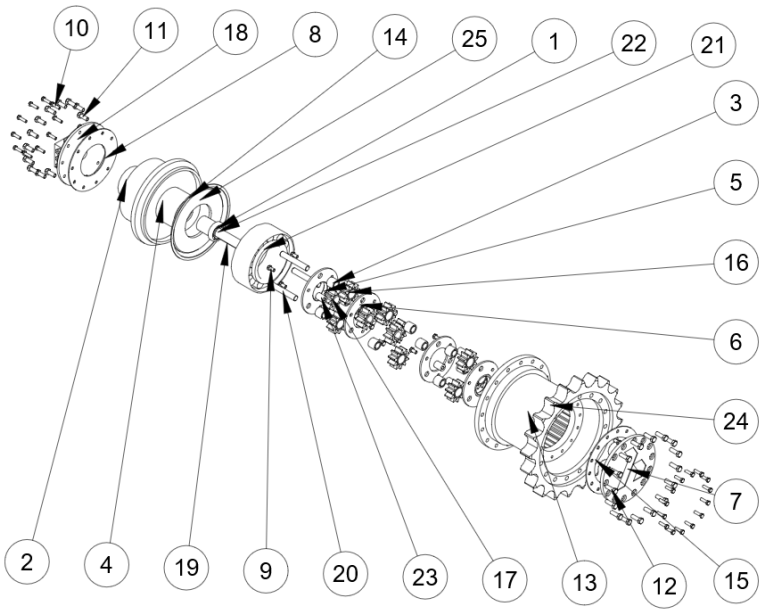
Figure 3.188

Bill of Materials tripod_main_assy		
No.	Component	Quantity
1	lower_tube_prt	3
2	middle_tube_prt	3
3	tripod_axis_prt	1
4	tripod_bond_prt	3
5	tripod_crosspiece_prt	1
6	tripod_sphere_prt	1
7	upper_tube_prt	3

Figure 3.189

Bill of Materials tripod_main_assy		
No.	Component	Quantity
1	tripod_axis_assy	1
1.1	tripod_axis_prt	1
1.2	tripod_crosspiece_prt	1
1.3	tripod_sphere_prt	1
2	tripod_foot_assy	3
2.1	bond_assy	3
2.1.1	tripod_bond_prt	3
2.2	tubes_assy	3
2.2.1	lower_tube_prt	3
2.2.2	middle_tube_prt	3
2.2.3	upper_tube_prt	3

Figure 3.190



Bill of Materials Drive_Assembly		
No.	Component	Quantity
1	BBRG-03	2
2	BBRG-03	1
3	BBRG-60	9
4	Body	1
5	Carrier_1	2
6	Carrier_2	2
7	Cover	1
8	Cover_Interal_Seal	1
9	DIN 933	12
10	DIN 933	24
11	DIN 933	16
12	DIN 933	16
13	Drum	1
14	External_Medium_Seal	1
15	External_Seal	1
16	Gear_Planet	9
17	Gear_Sun	3
18	Internal_Cover	1
19	Key	1
20	Pin	3
21	RLR BRG-32	1
22	Shaft	1
23	Sleeve	6
24	Sprocket	1
25	Washer	1

3.6.7 Exploded Representations

Commands: BMEXPLODE (6.8.6), BMEXPLODEMOVE (6.8.7),VIEWBASE (6.31.3). Youtube The exploded representation functionality creates associative exploded representations of assemblies without modifying the assembly itself. Exploded representations are stored in dedicated blocks which can be edited and inserted according to the user's needs.You can also generate drawing views from the exploded representations. Exploded representations are associative. Each part in an exploded representation is linked with the corresponding part in the assembly. Using the BMBALLOON (6.8.1) command, you can update exploded representations to the current state of the assembly as well as to place balloon annotations on corresponding drawing views. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/ExplodedRepresentationDriveAssemblyExplo

Levels of the exploded representation

Exploded representations contain references to the components which makes them quite lightweight.Two levels of the exploded representation are supported: Top or Bottom.

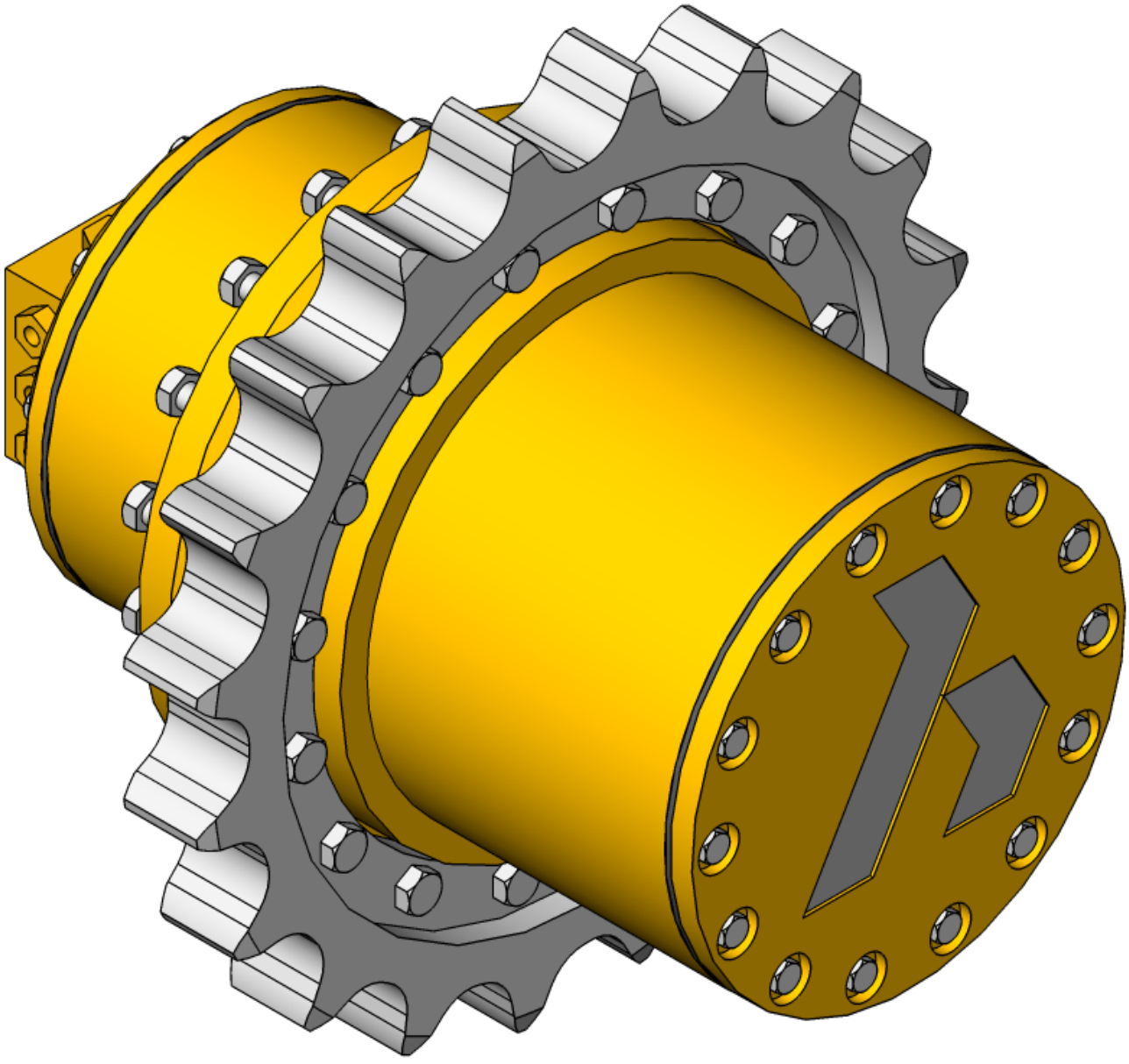
- **Top Level**: The exploded representation contains references to top level components only. Top level components are components inserted directly into the main assembly. A top level component in such exploded representation is not exploded further - all its parts and subassemblies remain assembled. Top level exploded representation allows you to see the basic composition of your assembly.
- **Bottom level**: The exploded representation contains references to bottom-level components only. Bottom level components are terminal parts, which do not contain other parts or subassemblies. Bottom level exploded representation allows you to see all parts isolated.

NOTE	If **Bottom level** is selected and there are some middle-level components containing not only inserts of components but also other entities, such entities will be lost in the exploded representation, because it is not possible to reference them correctly. Those entities should be converted into corresponding bottom-level components first.

Algorithms

BricsCAD provides two algorithms to automatically create exploded representations and a Manual mode to create customized exploded representations manually.

Figure 3.191



Steps and animations

Exploded views may contain one or several steps, which describe a particular assembly or disassembly sequence. Each step corresponds to a set of components that should be moved at this particular stage in order to assemble or disassemble the main assembly. All exploded representation algorithms automatically create necessary steps. It is also possible to add the user's steps as well as delete, merge, split or move them. Each step has a unique name that can be used to describe the step. It is possible to animate a particular step as well as the entire sequence of steps, both in direct and reverse directions (to get disassembly and assembly animations correspondingly). In order for the animation to work properly, an initial step that defines the initial state of the assembly must be present in the exploded representation. It is possible to automatically hide components not important for a particular step during the animation using the**Auto hide** property of the exploded representation.

Editing of exploded representations

Because each exploded representation is stored in a block, it is possible to edit the exploded representation using theblock editor (1.22.3). Do not useREFEDIT (6.26.12)command to edit exploded representations, as this command is incompatible with exploded representation steps. The fastest way to open the exploded representation for editing is to right-click at the required exploded representation in theMechanical Browser (6.8.29)and select "Edit" command from the context menu. Theblock editor (1.22.3)will be opened automatically. It is also possible to open the exploded representation for editing at the end ofBMEXPLODE (6.8.6)command. When the exploded representation is opened, it is possible to change the state of thecurrent step of the exploded representation. By default, this is the last step made by the algorithm. However, the current step can be changed using the**Current step** property of the exploded representation in the browser, or by right-click at the desired step in theMechanical Browser (6.8.29)and selecting "Apply" command from the context menu.

NOTE	Changing the current step will discard all the unsaved modifications, as well as any operation on steps except saving. Make sure that all necessary modifications are saved before changing the current step or performing any other operation on exploded representation steps.

DMMOVE (6.12.17),MOVE (6.21.30),DMROTATE (6.12.24), andBMEXPLODEMOVE (6.8.7)commands can be used to move components, inside the exploded representations, to their desired positions.In the case ofBMEXPLODEMOVE (6.8.7), all necessary steps will be added automatically after the current step. For other operations the current step must be saved afterward. Only the difference between the current step and the previous one is stored. It is also possible to change the exploded representation by performing operations on steps. For example, if a step is removed, all components moved at this step will be returned to their positions before that step. After editing is finished, it is necessary to save the current step, otherwise all changes will not be taken into account and could be discarded.

To create an exploded representation

The BMEXPLODE (6.8.6) command creates an exploded representation of the current assembly. **Note:** It is necessary to have at least one insert of a part in the assembly for the command to work.

1. To launch the BMEXPLODE (6.8.6) command, do one of the following:
   - Click the **Explode** tool button ( )on **Tools** panel of the **Assembly** ribbon tab;
   - Click the **Explode** tool button ( )on the **Assembly** toolbar;
   - Select **Explode** in the **Assembly** menu;
   - Type **bmexplode** in the command bar. **You are prompted:** Select explosion algorithm [Table by types/Linear/Manual
2. (Option) Set the **Level** and the **Name** of the exploded representation using the **Settings** option.
3. Select the algorithm. **You are prompted:** Select exploded view behavior [Edit/Generate drawing views/Finish]:
4. Select **Edit** to edit the exploded representation. Select**Generate drawing views**to generate drawing views of the exploded representation. Select**Finish**to finish the command.

To generate drawing views of the exploded representation

1. Launch the VIEWBASE (6.31.3) command. **You are prompted:** Select objects or [Entire model/preseTs/Special views] <Entire model>:

2. Select**Special views** option. **You are prompted:**Select view [Exploded view/Back] <Back>:
3. Select**Exploded view** option. A dialog to select the exploded representation will be opened. Select the exploded representation from the dialog. **You are prompted:**Enter new or existing layout name to make current <Layout1>:
4. (Optional) Enter the**Layout name** of the layout for the exploded representation.
5. Position the generated drawing views of the exploded representation in the paper space layout.

NOTE	To use balloons (see BMBALLOON (6.8.1) command) with drawing views of exploded representations, the GENERATEASSOCVIEWS (1.13.11) system variable must be ON.

To update an exploded representations

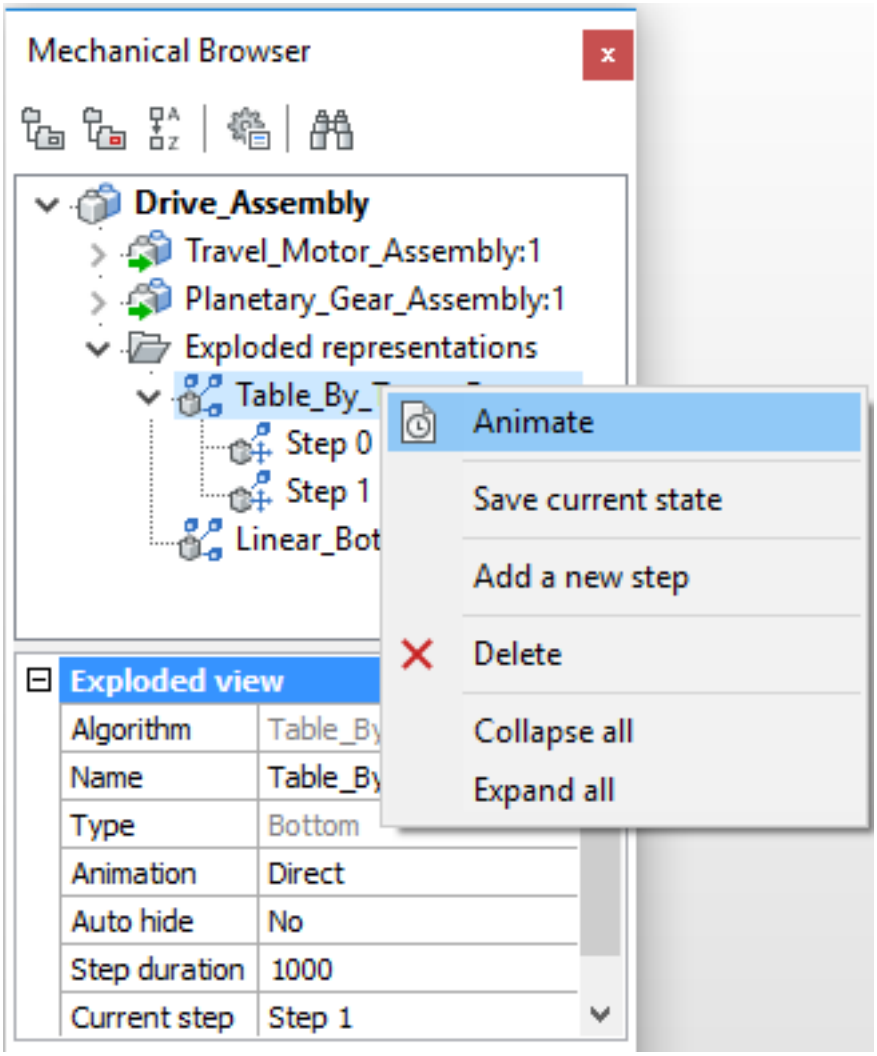
Execute the BMUPDATE (6.8.25) command to synchronize exploded representations with the current state of the assembly. If necessary, adjust the exploded representation. Such adjustments, for example, may be required because of addition and/or deletion of some parts.

NOTE	The position of parts in exploded representations created in **Manual** and**Linear**modes are not updated automatically, even if no editing has been done.

To manage exploded representations

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/ExplodedRepresentatio
Open theMechanical Browser (6.8.29)to see all exploded representations in the document. All exploded representation will be listed under the**Exploded representations** group. Open the context menu for the selected exploded representations to see all supported operations in this context.

Figure 3.192


Mechanical Browser
Drive_Assembly
Travel_Motor_Assembly:1
Planetary_Gear_Assembly:1
Exploded representations
Step 0
Step 1
Animate
Save current state
Add a new step
Delete
Collapse all
Expand all
Exploded view
Algorithm
Name
Type
Bottom
Animation
Direct
Auto hide
No
Step duration
1000
Current step
Step 1

Properties of exploded representations Each exploded representation has a set of properties, some of them can be edited by the user.

- **Algorithm -** the name of the algorithm used to create this exploded representation.
- **Name -** Name of the exploded representation. Must be unique for all exploded representations as well as for all the blocks in the model.
- **Type -** Type of the exploded representation: **Top** or**Bottom**.
- **Animation** - the direction of animation. Can take one of two values:
  - **Direct** - from start to end.
  - **Reverse** - from end to start.
- **Auto hide** - if set to**Yes**, all parts not important for a particular step will be hidden during the animation.
- **Step duration** - default duration of each step during the animation, in milliseconds. Default value is 1000, which means 1s duration.
- **Current step** - the current step that corresponds to the state of the exploded representation.

Operations on exploded representations The supported operations depend on if the exploded representation is opened for editing. If the exploded representation is not opened for editing, following operations are possible:

- **Edit** - opens the exploded representation for editing. Alternatively, it is possible to edit the exploded representation usingBedit (6.6.7)
- **Generate views** - launchesViewBase (6.31.3)command for the selected exploded representations. The name of the exploded representation will be used as the name of the layout. Current orientation is used for the exploded representation automatically.
- **Delete** - removes the exploded representation and the associated block from the document.

If the exploded representation is opened for editing, following operations are possible:

- **Animate** - animates the entire sequence of steps.
- **Save current state** - saves the current positions of parts in the current step.
- **Add a new step** - adds a new step after the last one. Note: the new step will be automatically set as current, and all unsaved modifications of the exploded representation will be lost.
- **Delete** - removes the exploded representation and the associated block from the document.

In addition, all steps will be listed in theMechanical Browser (6.8.29)for this exploded representation.

To manage exploded representation steps

Open theMechanical Browser (6.8.29)and open the required exploded representation for editing. All steps will be listed under the corresponding node in the tree.

Properties of steps

- **Name** - name of the step. Must be unique for this exploded representation.
- **Duration, ms** - the duration of the step. By default, equal to the**Step duration** set for the exploded representation.
  - **Duration type** - specifies if the duration should be taken from the exploded representation, or is specific for this particular step.
  - **Duration value**(only for**Absolute** duration type) - the duration for this step in milliseconds.

Operations on steps

- **Apply** - sets the step as the current and updates the parts in the exploded representation according to this step.
- **Animate** - animates the selected step.
- **Set as previous** - clears the step, so the positions of all parts will be the same as at the previous step.
- **Merge with previous** - merges this and the previous step, then removes the previous step.
- **Split** - splits the step to a sequence of steps, each of them corresponding to exactly one part.
- **Move step up** - moves the step one position up in the tree.
- **Move step down** - moves the step one position down in the tree.
- **Delete** - completely removes the step as well as the associated movements of parts. After that, it updates positions of parts in the exploded representation accordingly.

To edit exploded representations

1. Select the exploded representation in theMechanical Browser (6.8.29).
2. Right-click at the exploded representation and select**Edit**from the context menu **Or** RunBedit (6.6.7)command and select the block with the exploded representation
3. Perform the necessary operations to get the desired exploded representation. Do not forget to save the current step when necessary.
4. Save changes usingBclose (6.6.4)with the option**Save**.

NOTE	Avoid using the **Discard** option ofBclose (6.6.4), as it can lead to an inconsistent state of the block and exploded representation steps. UseUndo (6.30.5)instead.

Figure 3.193

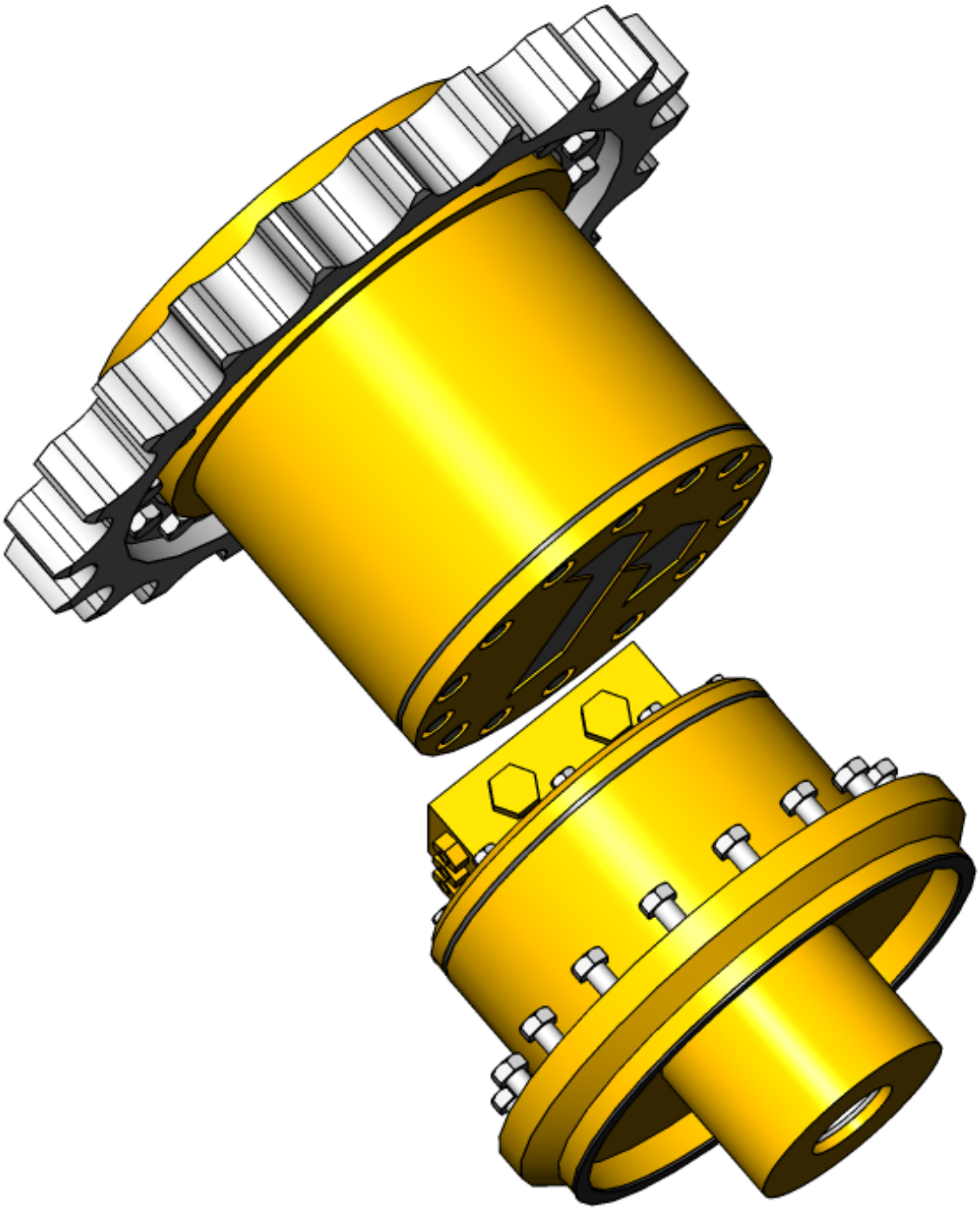


Figure 3.194

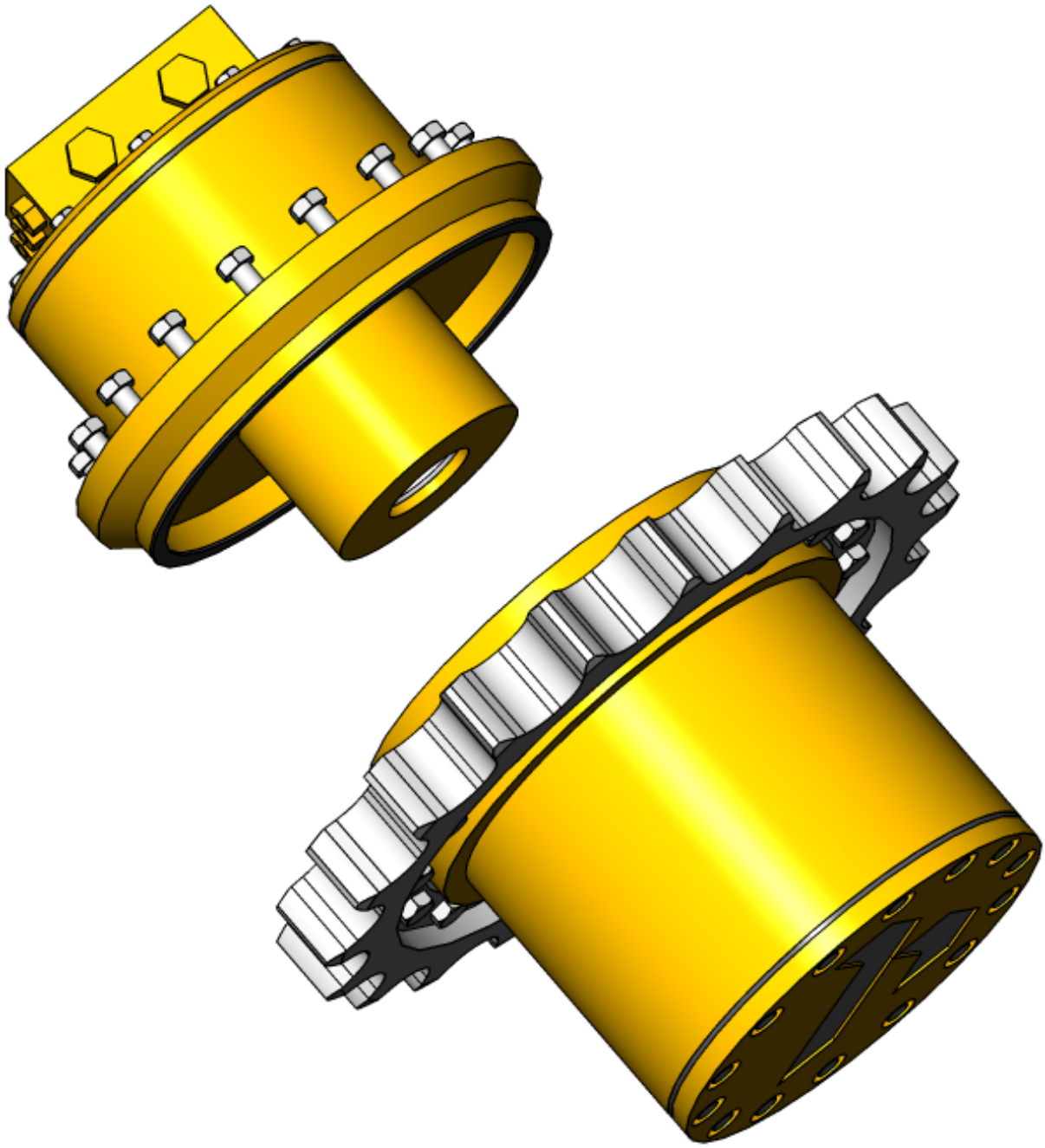


Figure 3.195

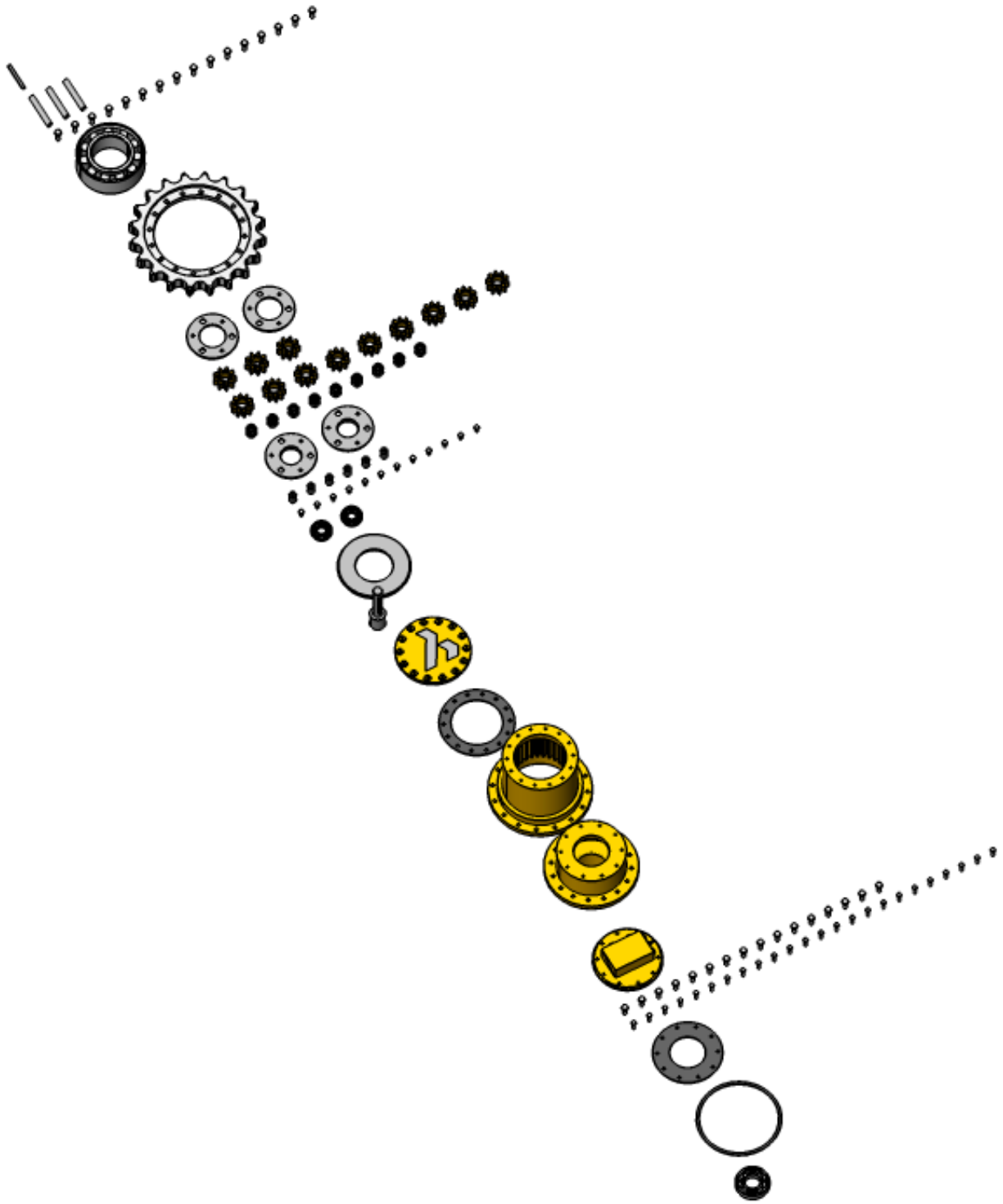


Figure 3.196



3.6.8 Assembly mass properties

Command: BmMassProp (6.8.14) An assembly can consist of multiple parts made from different materials. To correctly compute the mass, center of gravity (centroid), and inertia of such assemblies, BricsCAD supports individual densities for components. A physical material (2.6.10) for a mechanical component is defined in the Material (1.22.22) property in the Mechanical Browser (3.2.2). By default, a component inherits the density of the assembly, in which it was inserted into or formed from.

To set the mass units

1. Open the Settings dialog box (1.13.11).
2. Expand the **Property units** settings under **Drawing / Drafting / Drawing** units. (\vref {fig:C:/Dev/Task-2448/images/staticb
3. Check the **Format mass properties** checkbox to enable formatting of mass values with appropriate units.
4. Select the **Mass units** field, then click the **Browse** button (...) and specify the desired mass units in the **MASSUNITS** dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesig
5. Specify the desired precision in the **Mass precision** field.
6. Close the Settings dialog box (1.13.11).

NOTE	The **Mass units** setting affects the mass values only. Other mass properties such as density or moments of inertia will be formatted in SI units for the metric system and in imperial units for the imperial system, regardless of the value of the **Mass units** setting.

Figure 3.197

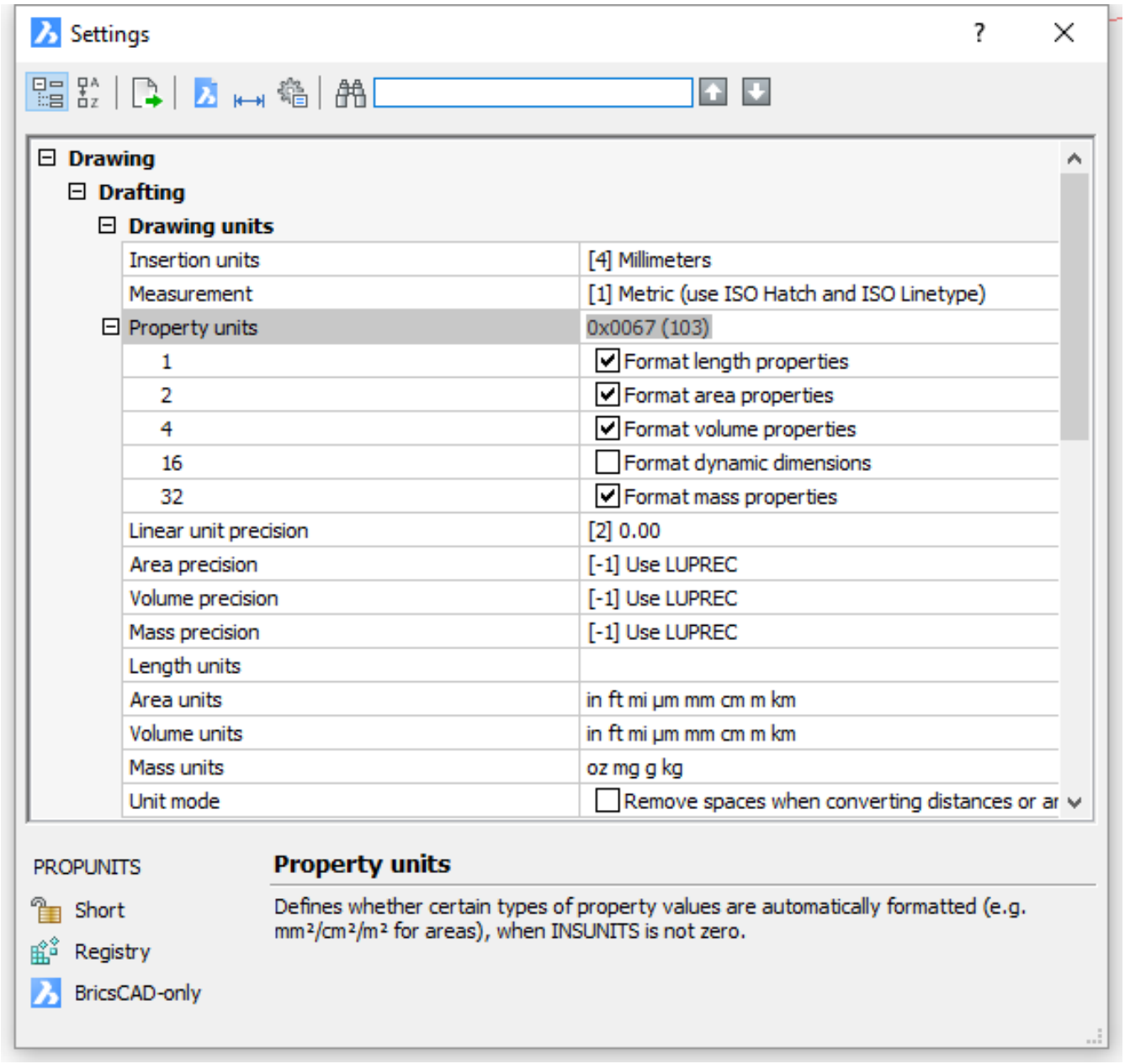
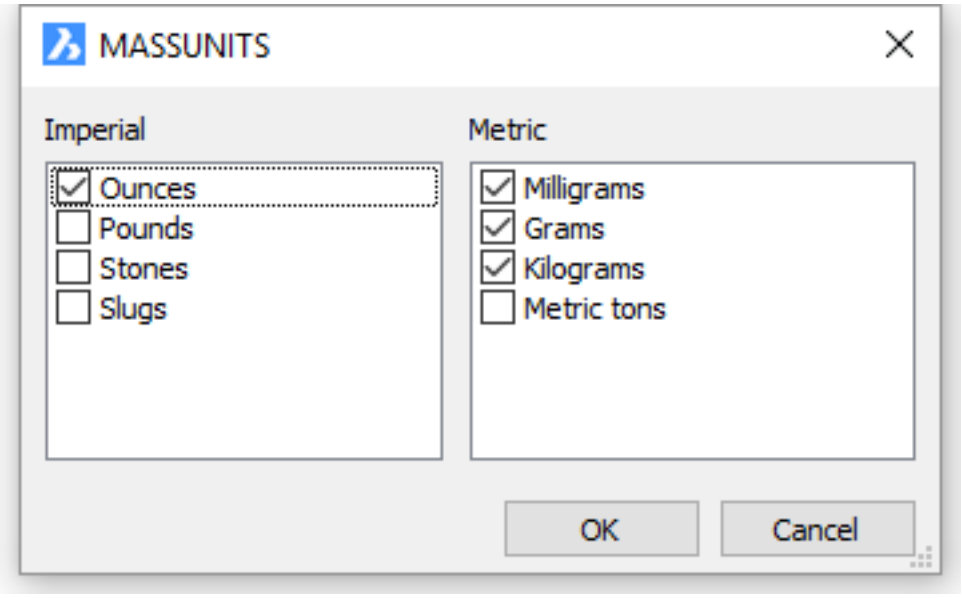


Figure 3.198



To compute the mass properties

Execute the BmMassProp (6.8.14) command. You can compute mass properties for the entire model or for a selection of 3D solids and/or subcomponents. The values of mass, volume, centroid, moments of inertia, products of inertia, radii of gyration, principal moments and X-Y-Z directions about centroid are reported in the command bar. Optionally you can align the User Coordinate System with the inertia axes, as a result, the center of masscentroid of the component lies at the origin of UCS, which can be useful to investigate the behavior of a product under gravity force.

3.6.9 Standard Mechanical Components

Commands: ComponentsPanelOpen (6.9.61) Youtube BricsCAD Mechanical contains a library of 30,000 standard mechanical components of 800+ different types. They are available in Components panel (6.9.61), and can be inserted into your assembly with drag'n'drop. You can adjust the parameters by using the Properties panel (6.24.49), which temporarily lists parameters of a component being inserted or after insertion by using Mechanical Browser (6.8.29). Components panel is configured in the Mechanical profile to contain three categories:

- Standard Parts
- Holes
- Sheet Metal

Standard Parts (Hardware)

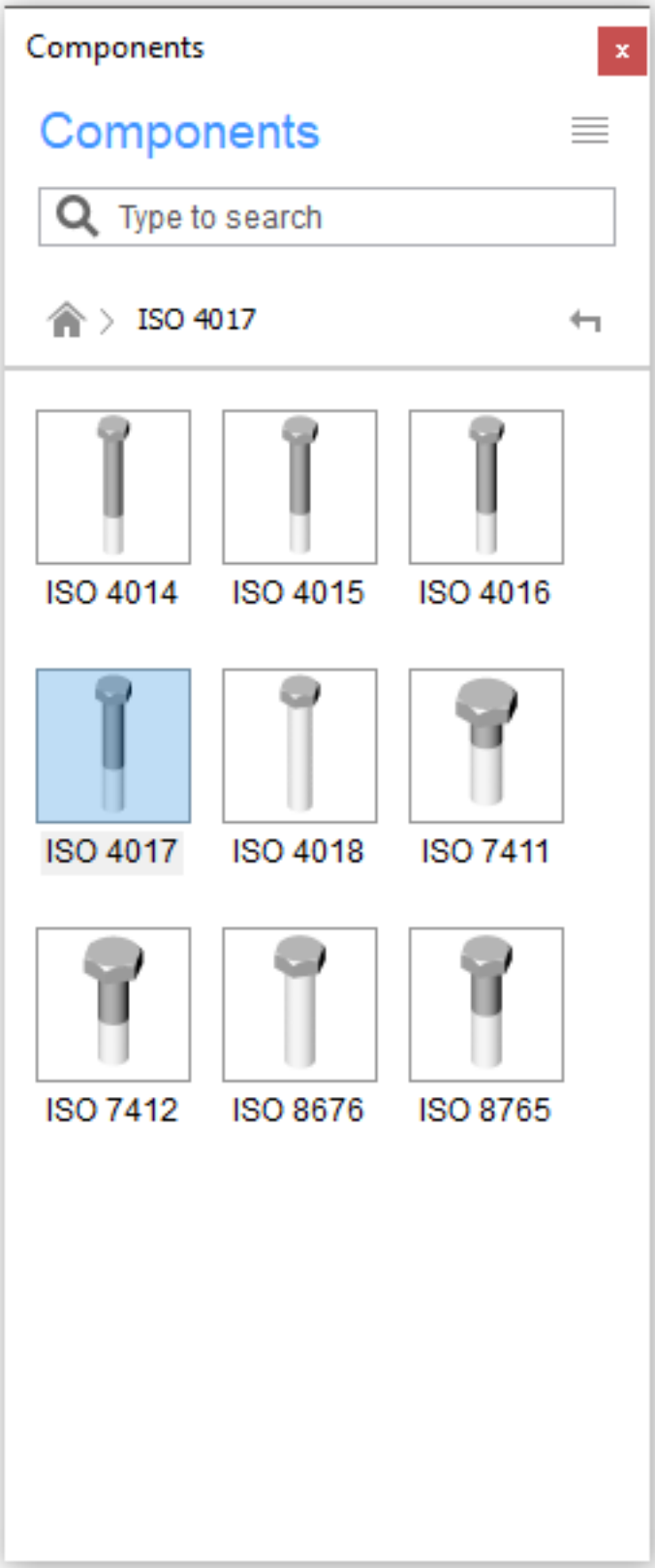
Standard Parts category contains parametric standard hardware such as bolts, nuts, bearings, gears, and etc. The content of Standard Parts category is further structured by type, subtype, and standard (example: Standard Parts / BOLT / HEX HEAD / ISO / ISO 4017). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Asse

NOTE

There are two specific settings you can configure for Standard Parts in Settings (6.27.23) dialog:

- **Thread representation** is used to represent 3D thread for bolts and nuts or do not represent it (the latter option allows to decrease the memory consumption and increase the performance for large assemblies);
- **Maximum number of gear teeth** is used to limit the number of teeth for gears (sprocket weals, which are available under Standard Parts / SPROCKET), which also allows to decrease memory consumption and increase the performance.

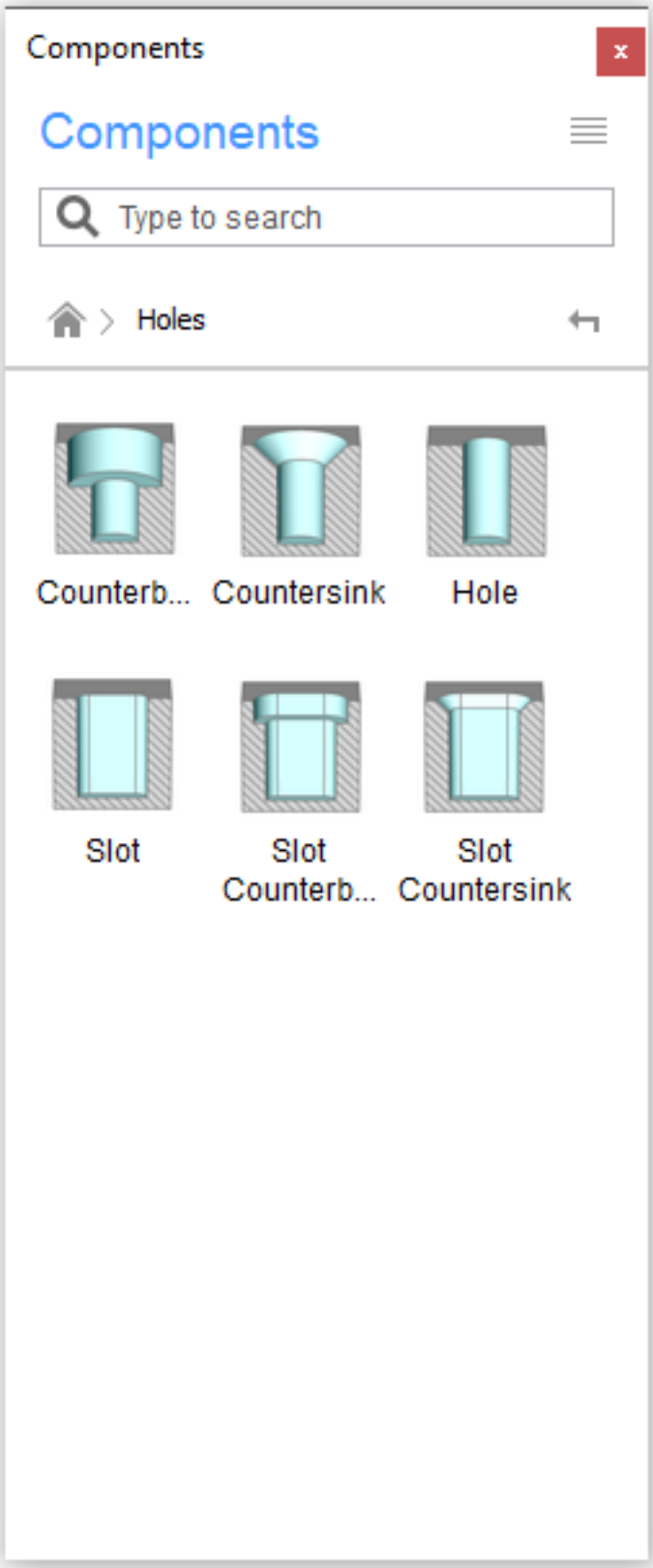
Figure 3.199


Components
Components
Type to search
ISO 4017
ISO 4014
ISO 4015
ISO 4016
ISO 4017
ISO 4018
ISO 7411
ISO 7412
ISO 8676
ISO 8765

Holes

Holes category contains parametric library of standard holes such as slots, counterbore, countersink, and etc. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/StandardMechanicalCompo

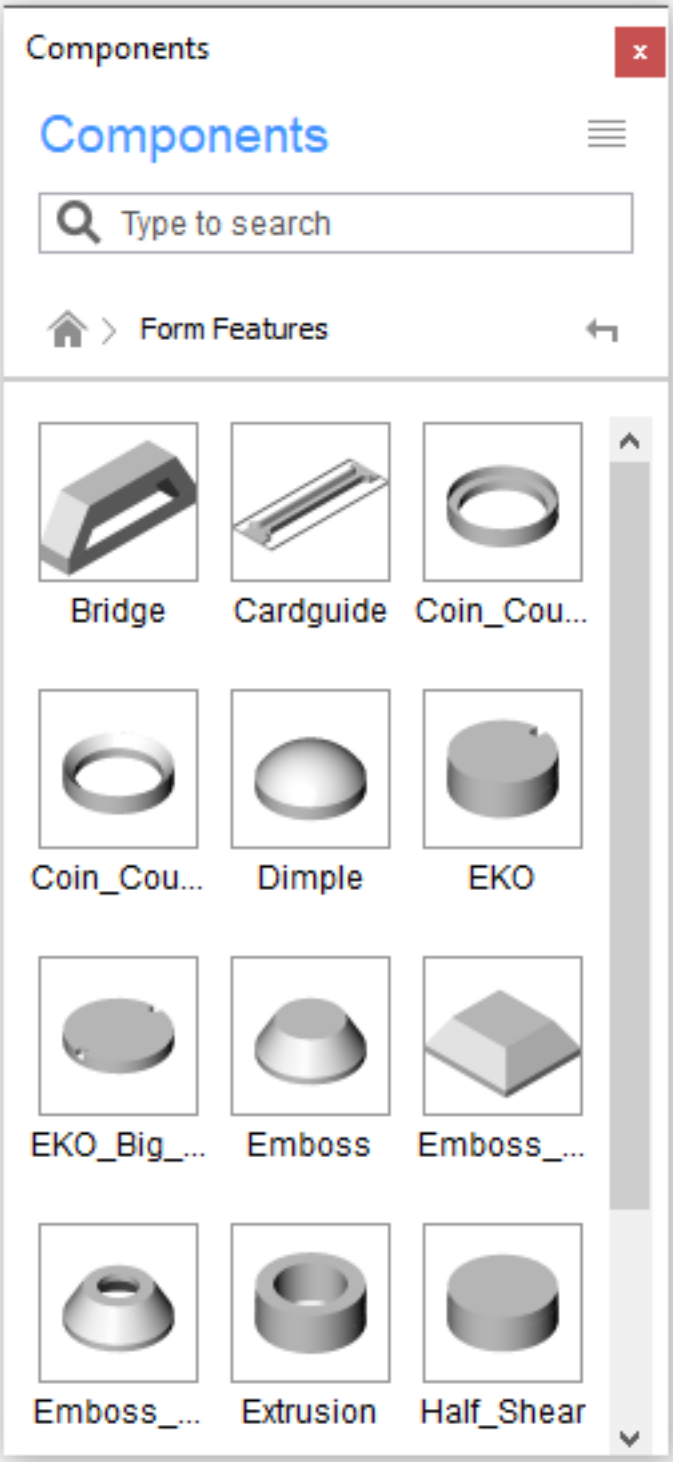
Figure 3.200


Components
Components
Type to search
Holes
Counterb...
Countersink
Hole
Slot
Slot Counterb...
Slot Countersink

Sheet Metal Form Features

Sheet Metal category contains parametric library of sheet metal form features such as bridges, embosses, and etc.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/AssemblyDesign/StandardMechanicalC

Figure 3.201


Components
Components
Type to search
Form Features
Bridge
Cardguide
Coin_Cou...
Coin_Cou...
Dimple
EKO
EKO_Big_...
Emboss
Emboss_...
Emboss_...
Extrusion
Half_Shear

To place a standard component into the current drawing:

1. Choose a category.
2. Choose a standard.
3. Select a type.
4. Click on desired standard component.
5. Select appropriate parameters in properties panel.
6. Click in the drawing area to insert the selected component with chosen parameters.

To create your own standard component:

You can create your own standard parts and add them to Components panel using BmCreateComponent (6.8.3) command.

3.7 Sheet Metal - Workflows

3.7.1 Feature coloring in BricsCAD Sheet Metal

Youtube With feature coloring you can easily distinguish one sheet metal feature from another and visually validate your sheet metal parts.Each of the sheet metal features – flanges, bends, junctions and reliefs – has its own assigned color. You can easily re-set colors in the settings menu. Back to Tutorial Overview

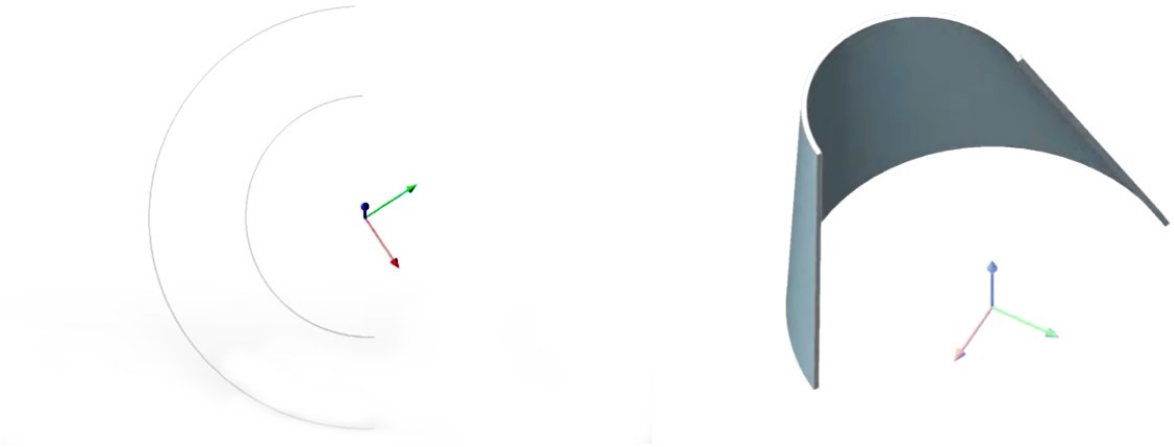
3.7.2 Lofted bends in BricsCAD Sheet Metal

Youtube BricsCAD creates sheet metal parts with lofted surfaces representing smooth transitions between two profiles, such as between lines, arcs, circles, rectangles, polylines, and splines. Imported sheet metal parts with lofted bends can be edited and unfolded. BricsCAD supports sheet metal parts with lofted bends, which are manufactured by a series of regular bends. Supported profiles include lines, arcs, circles, rectangles, polylines and splines.

A lofted bend can be created from two curves

(\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2016/12/imageContent3-1-091132.jpg})

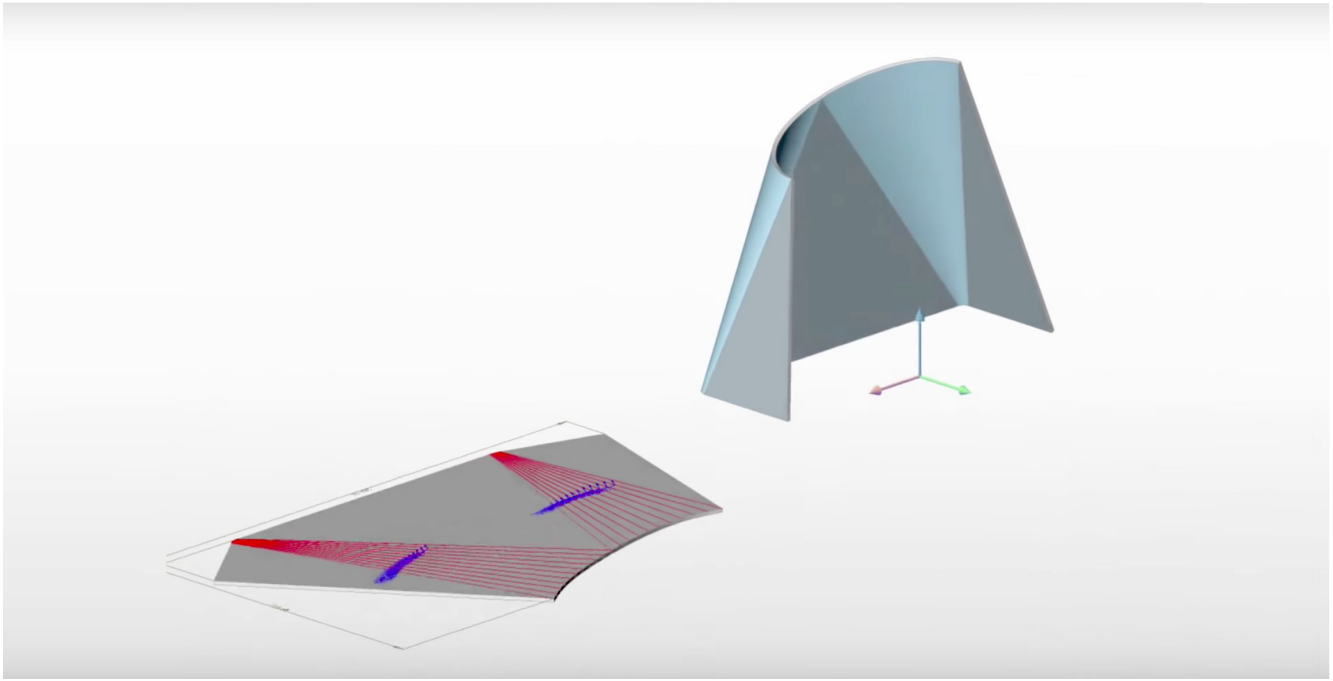
Figure 3.202



Imported sheet metal parts with lofted bends can be edited and unfolded

(\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2016/12/imageContent6-091132.jpg})
Back to Tutorial Overview

Figure 3.203



3.7.3 Split thickness faces into regions

https://www.youtube-nocookie.com/embed/G5Q80Li-7-w When usingjunction features in your designs, Sheet Metal for BricsCAD will allow youto split thickness faces into regions. Thus enhancing your ability to work with junction features. Back to Tutorial Overview

3.7.4 Making a parametric workbench - Part 4

https://www.youtube-nocookie.com/embed/u2skl7fx4f8 In the final part of this series, we take on the role of a sheet metal fabricator to prepare our workbench for unfolding, then generate the flattened profiles. Back to Tutorial Overview

3.7.5 Batch processing in BricsCAD Sheet Metal

https://www.youtube-nocookie.com/embed/1fjMsqGpmbk Batch processes of sheet metal parts and assemblies can be**automated in large numbers**using a**dedicated LISP API.**This automation feature accomplishes tasks, such as producing families of sheet metal parts of different sizes. Example LISP scripts are included with BricsCAD.
Back to Tutorial Overview

3.7.6 Working with lofted bends of an imported model

https://www.youtube-nocookie.com/embed/cnnop3X3PqU In this tutorial, we will show how an airconditioning system is manufactured using assembly modeling and lofted bends. Back to Tutorial Overview

3.7.7 Rework an imported model in BricsCAD Sheet Metal

https://www.youtube-nocookie.com/embed/WA7cfBKKJas In this tutorial, we'll investigate the creation of a sheet metal air box from an existing solid model. Back to Tutorial Overview

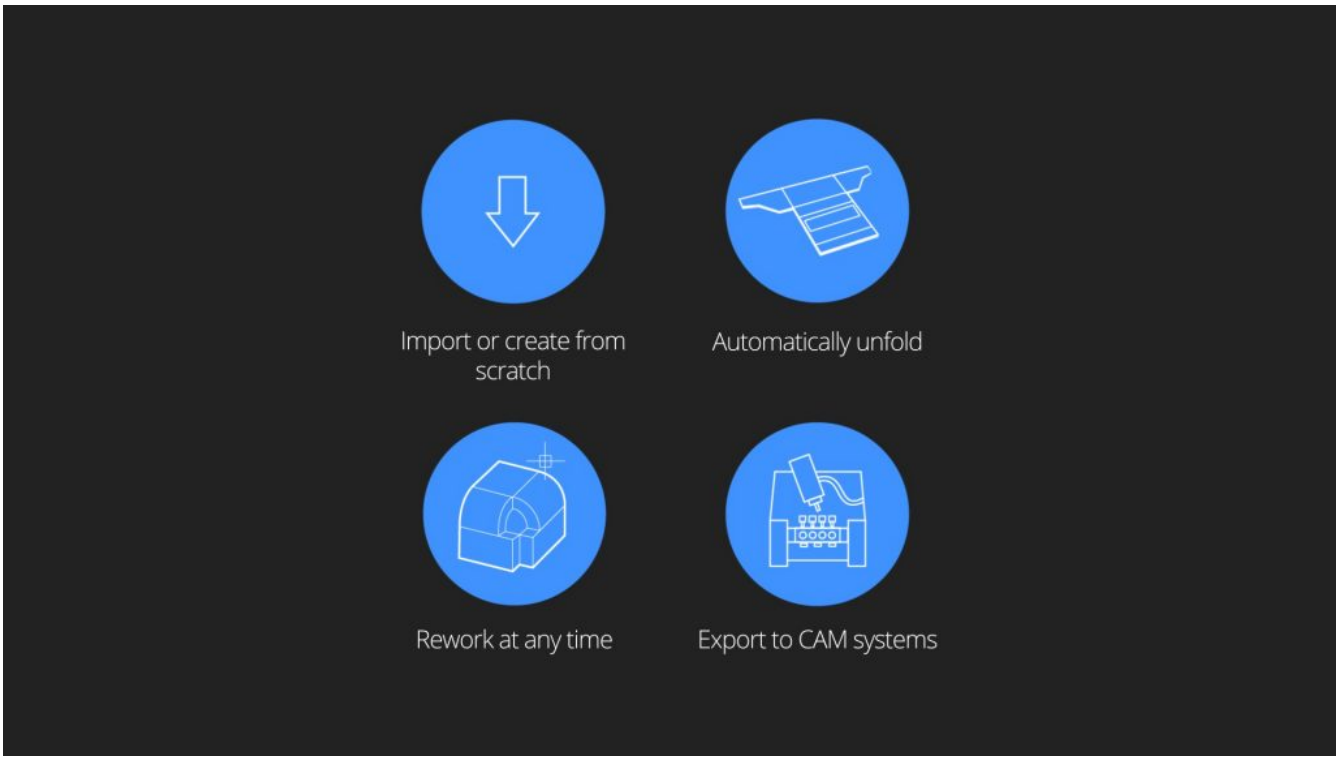
3.7.8 Designing from scratch in BricsCAD Sheet Metal

https://www.youtube-nocookie.com/embed/zPTzEKV5IrI This video demonstrates how to create a sheet metal part from scratch and then unfold it to export it to a CAM system. Creating new sheet metal parts or reworking existing ones? BricsCAD is your best choice. It is free of conceptual mistakes of history-based MCAD software.You will never need to restart your design from scratch, because we guarantee maximum flexibility for changing your models by combining the freedom of direct modeling with the power of feature-based design. Back to Tutorial Overview

3.7.9 Start from an imported model in BricsCAD Sheet Metal

Youtube Combining the simplicity of direct modeling with the power of parametric features, sheet metal parts are generated by applying direct editing operations to the geometry. Edit feature parameters in the browser. Go beyond the limits imposed by traditional history-based modeling systems with BricsCAD's interactive sheet metal toolset.
Back to Tutorial Overview

Figure 3.204



3.7.10 Introducing BricsCAD Sheet Metal

Youtube BricsCAD Sheet Metal lets you create, import, rework and unfold using the most intuitive 3D direct modeling techniques. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent2-1024x576.jpg})

Import from any CAD system

Import from any CAD system. Supported formats include neutral ones, such as STEP and IGES; native formats, such as CATIA, Inventor, SolidWorks and kernel formats. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads 1024x576.jpg}) (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent6-1024x576.jpg})

Figure 3.205

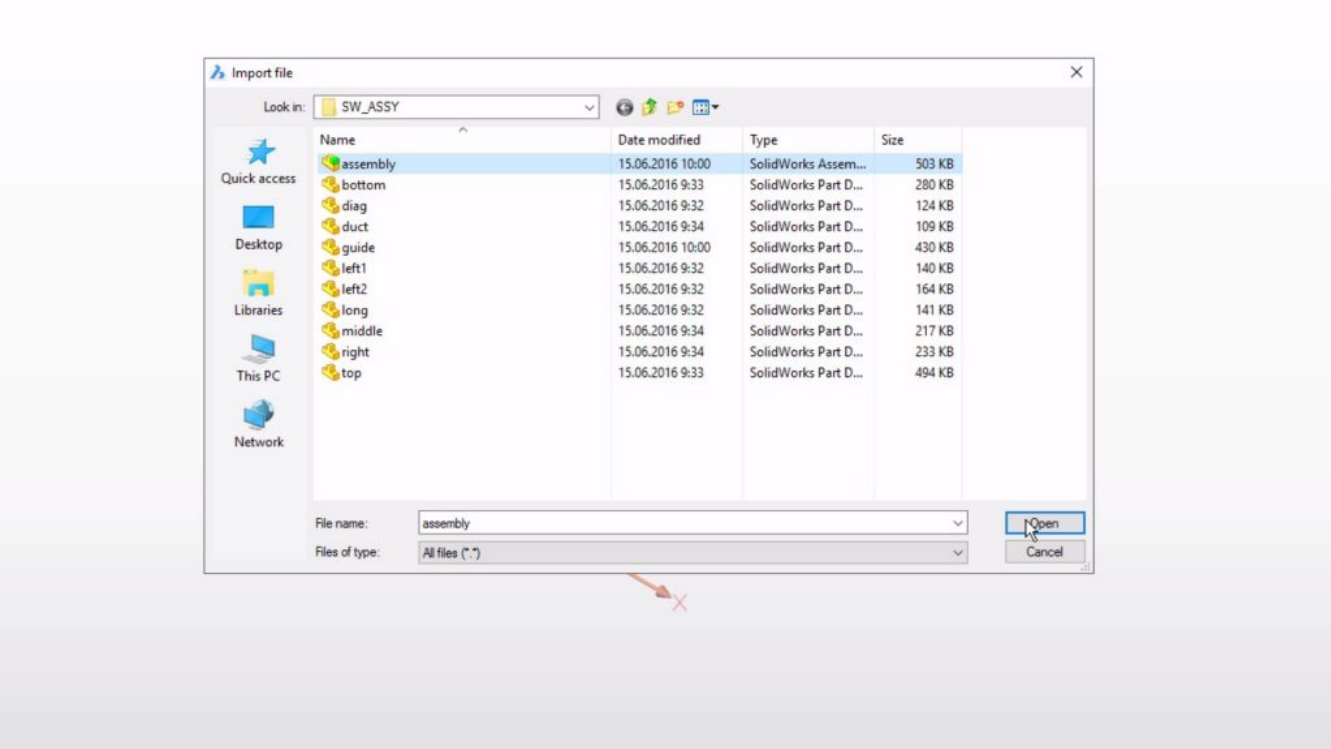
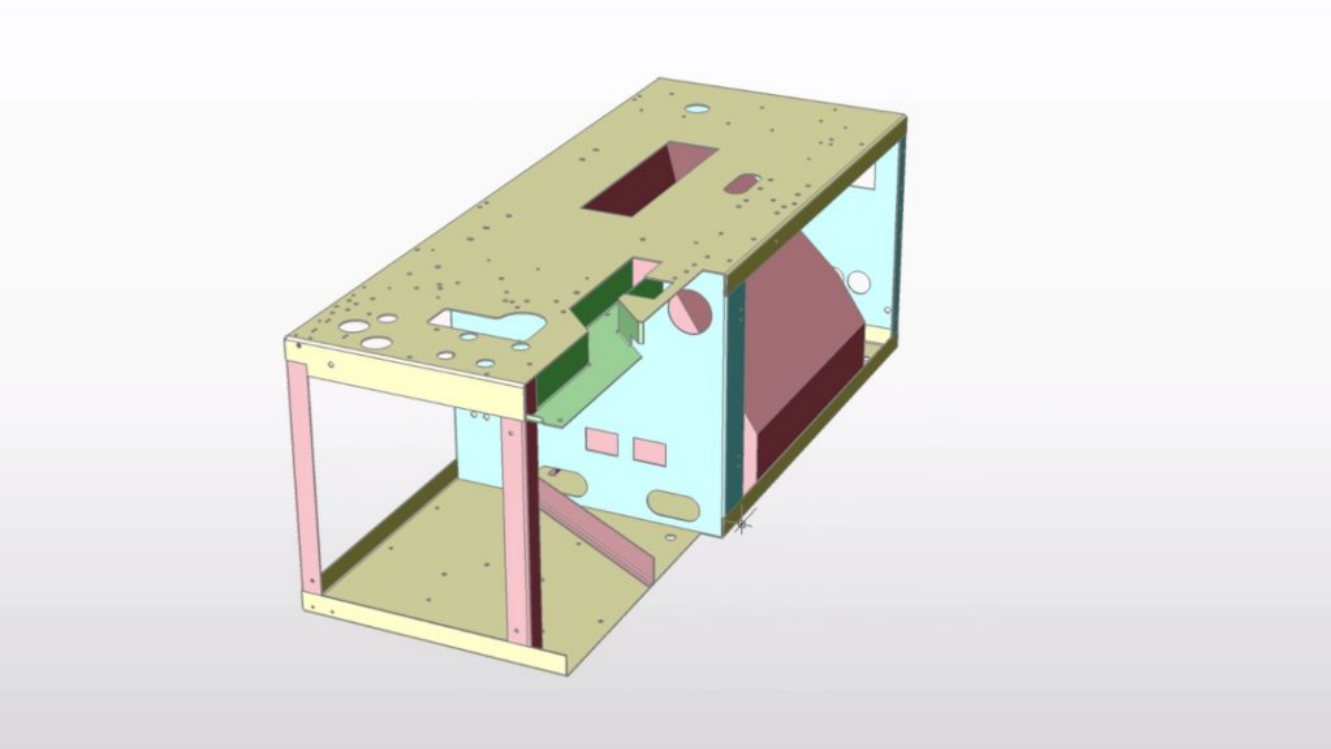


Figure 3.206



Prepare for production

By converting your parts to sheet metal, you can easily distinguish one sheet metal feature from another and visually validate them. Each of the sheet metal features – flanges, bends, junctions, and reliefs – has its own assigned color.
(\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent9-1024x576.jpg})
(\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent10-1024x576.jpg})

Figure 3.207

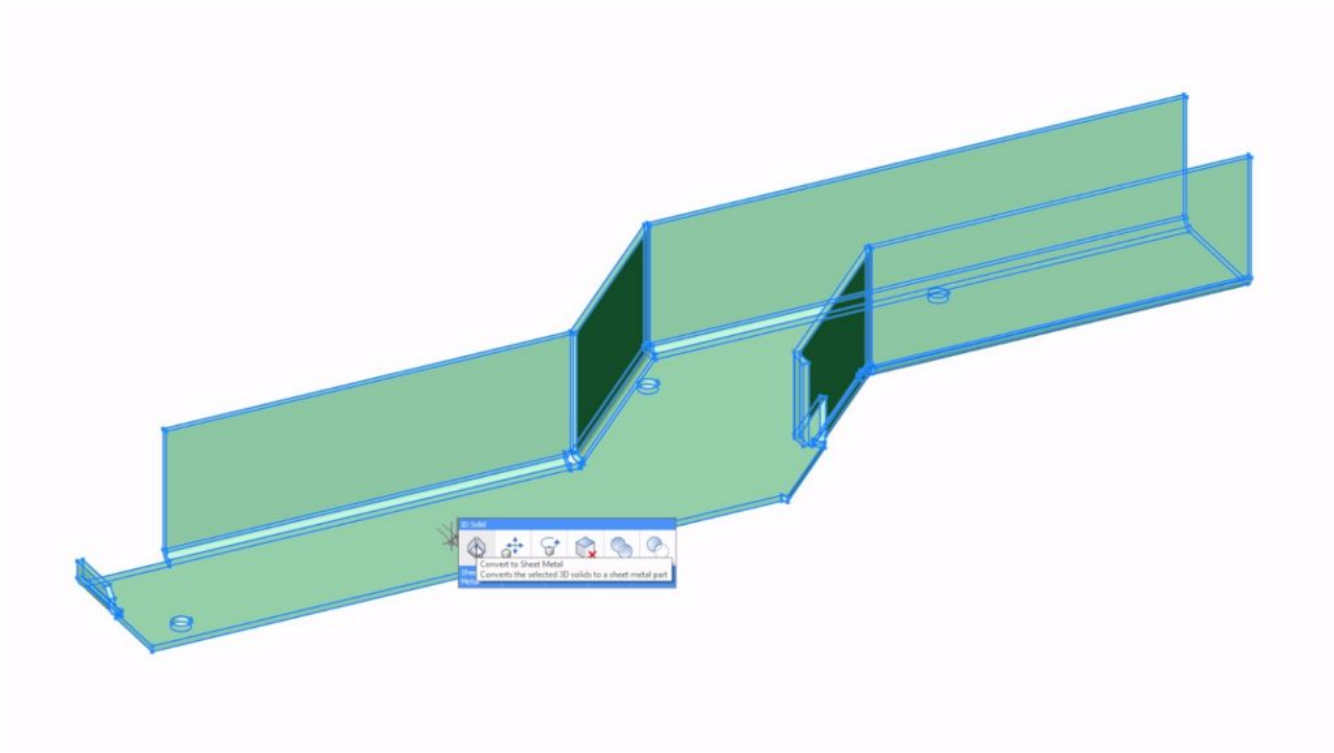
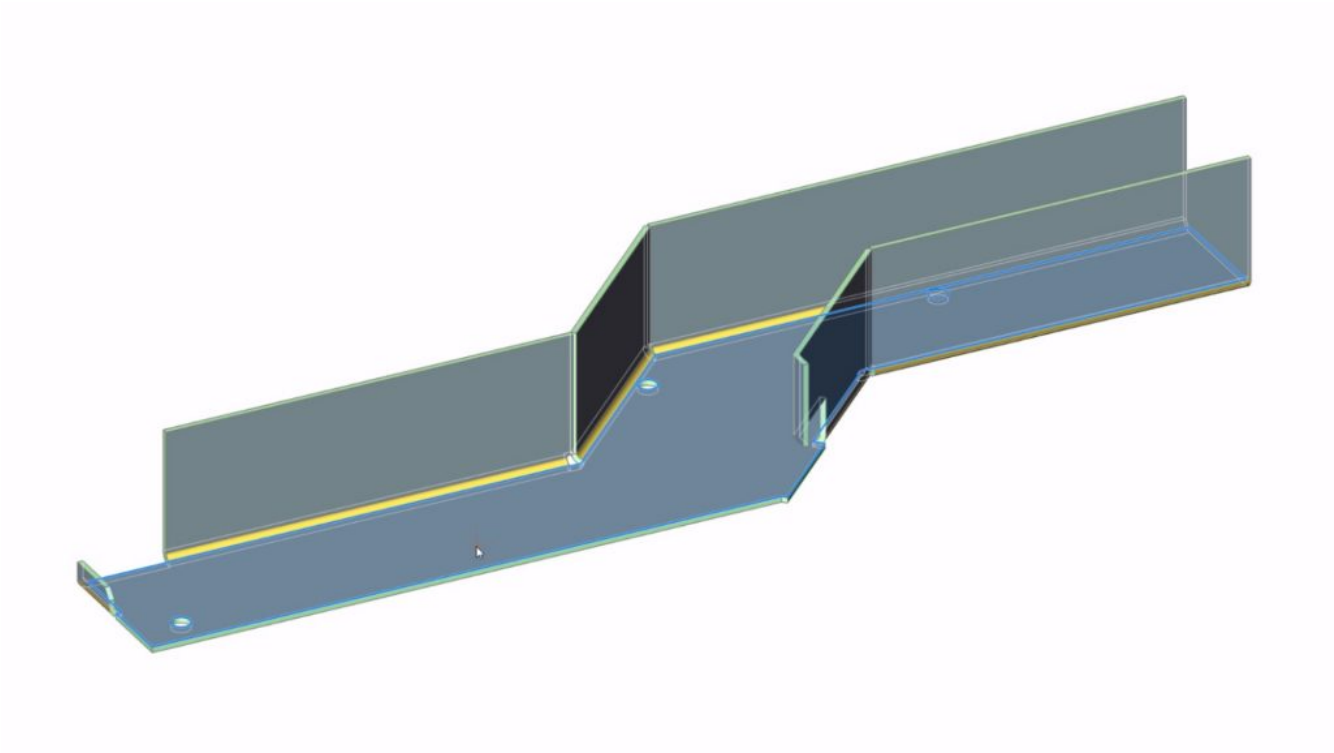


Figure 3.208



Automatically unfold

The model needs to be reworked

> "In manufacturing more than 80% of the imported sheet metal parts from a CAD program are not ready to be unfolded"

Rework at any time In this case, we have to split the part in two to make it ready for production. Note that with other CAD software, you need to restart from scratch to rework your model. With BricsCAD Sheet Metal, you can keep reworking the model until it's ready for production. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent17-1024x576.jpg}) (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent18-1024x576.jpg}) (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent19-1024x576.jpg})

Figure 3.209

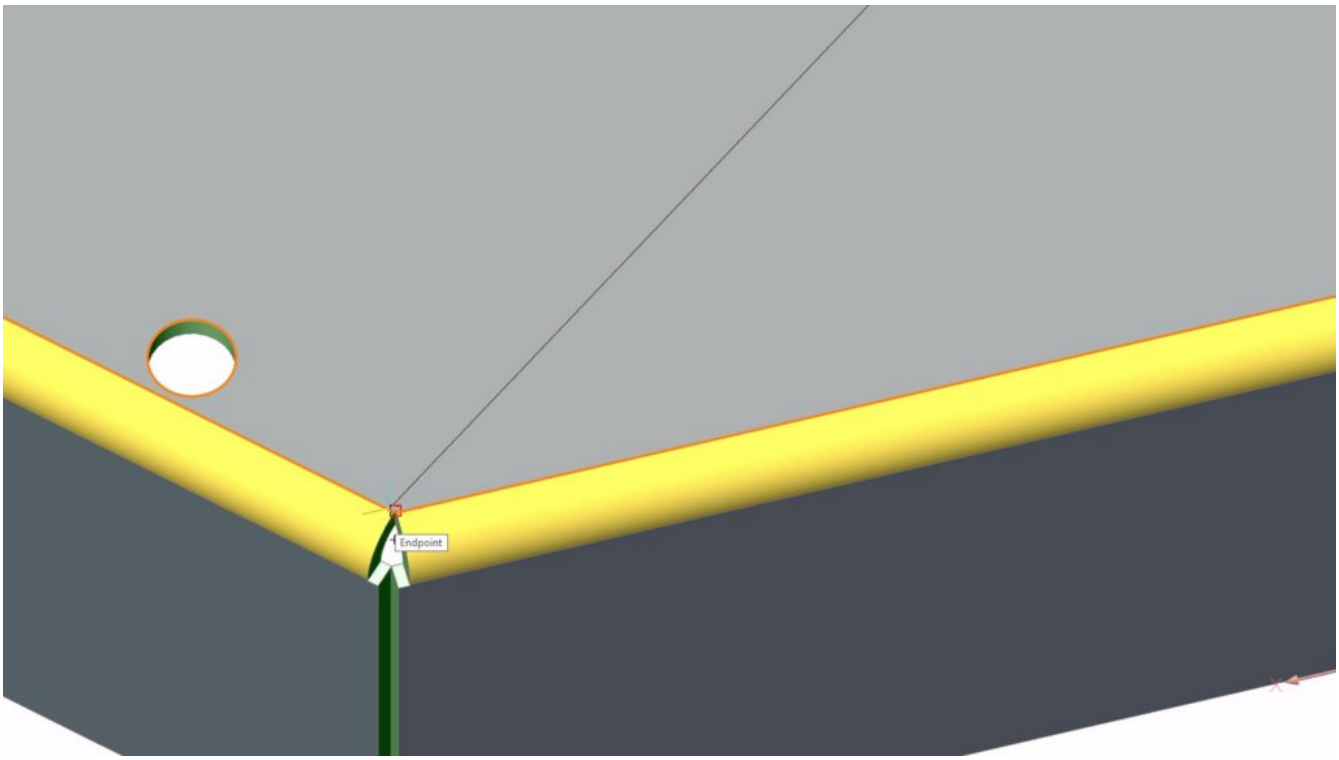


Figure 3.210

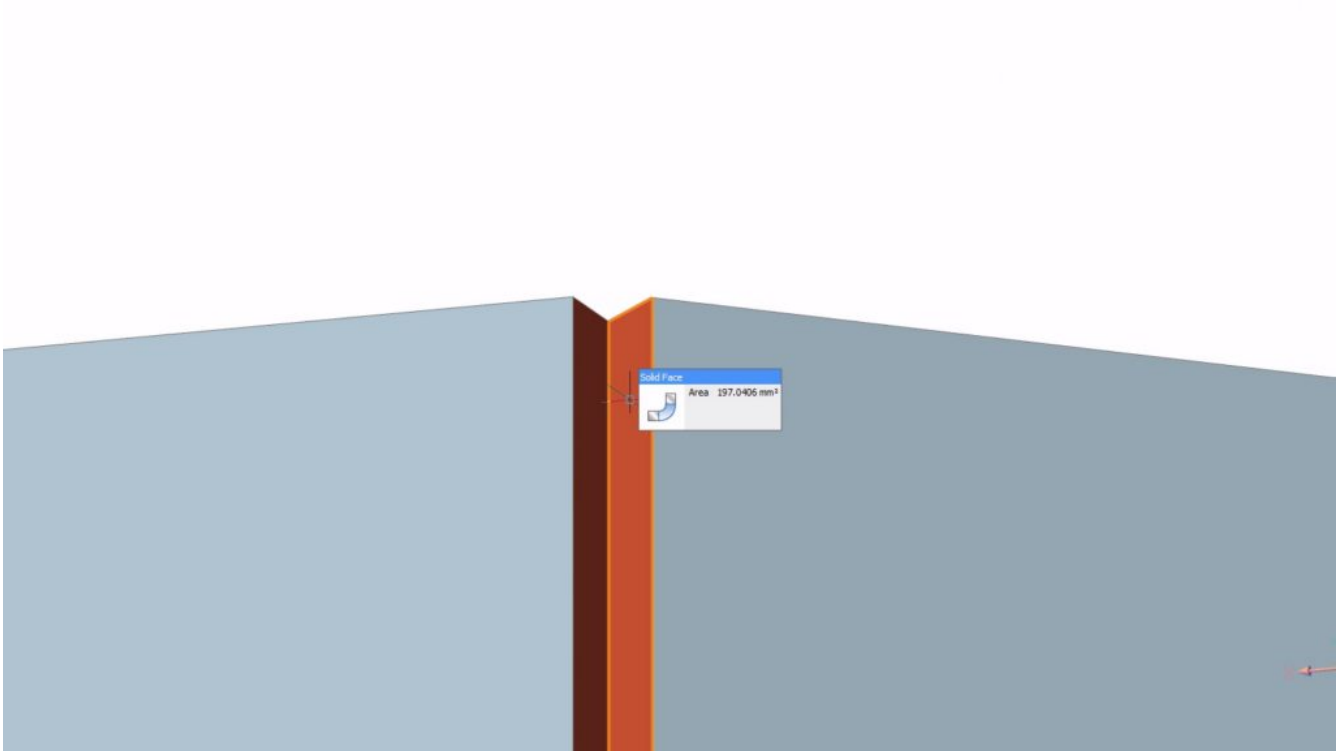
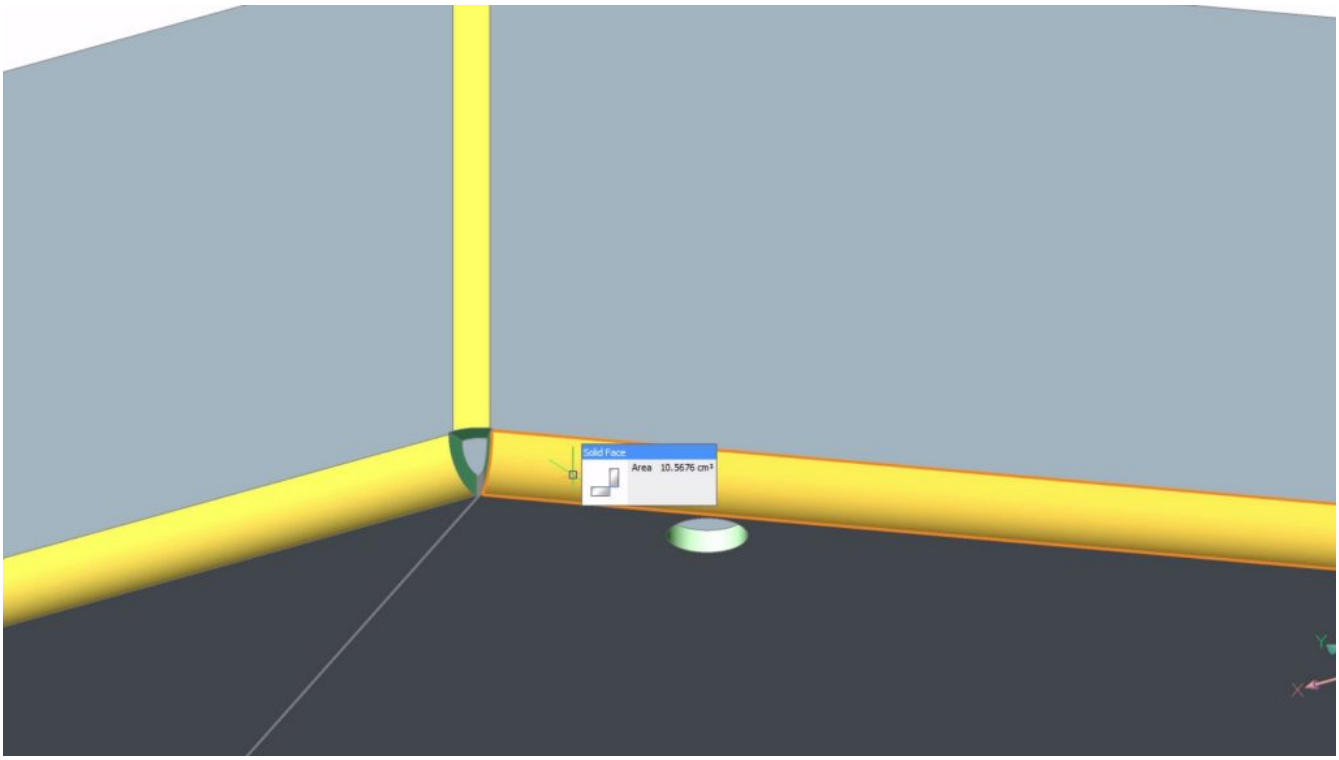
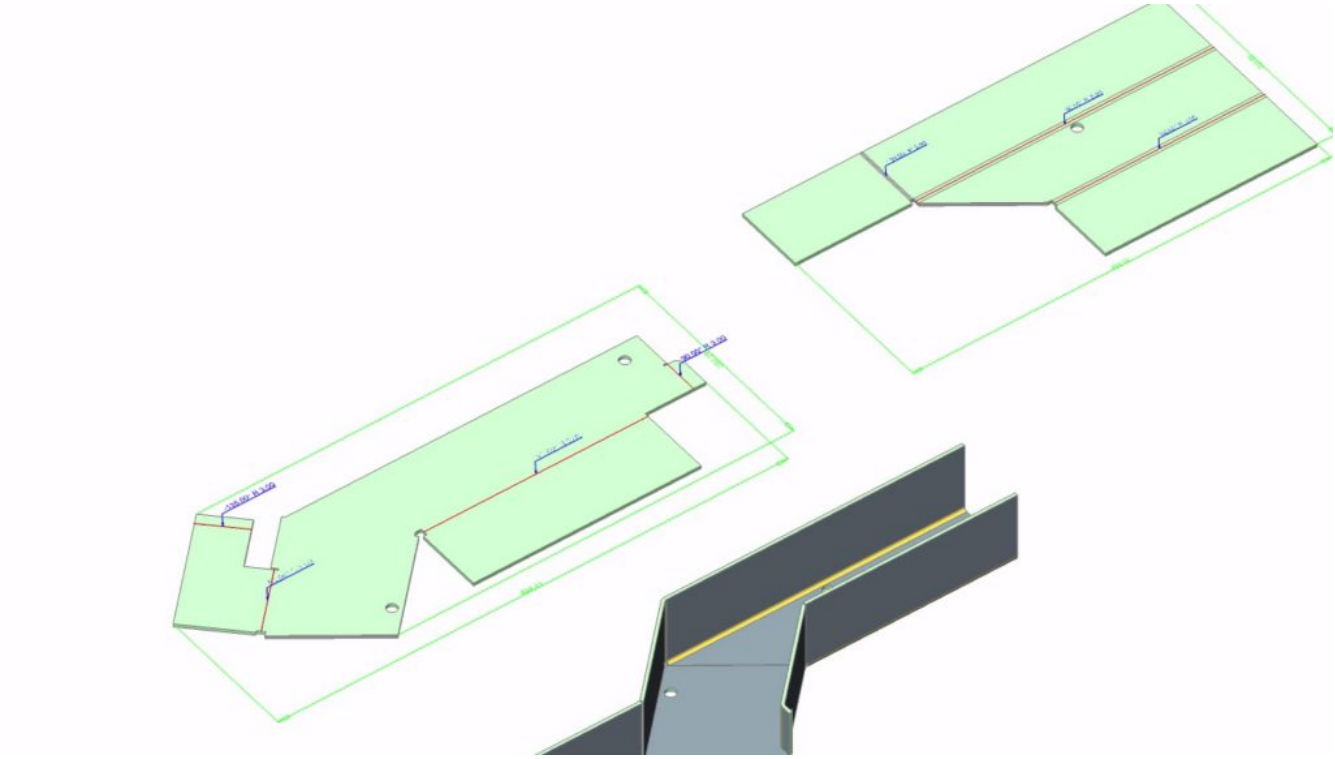


Figure 3.211



We moved a flange from one part to the other Note that by switching a junction to a bend on the left part and switching a bend to a junction on the right part, we moved a flange from one part to the other. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent21-1024x576.jpg})

Figure 3.212



Export to CAM system

Once your design is ready for production, you can export your unfolded model to any CAM system. (\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent24-1024x576.jpg})
(\vref {fig:C:/Dev/Task-2448/images/blogbricsyscom/wp-content/uploads/2017/06/imageContent25-1024x576.jpg})
Back to Tutorial Overview

Figure 3.213

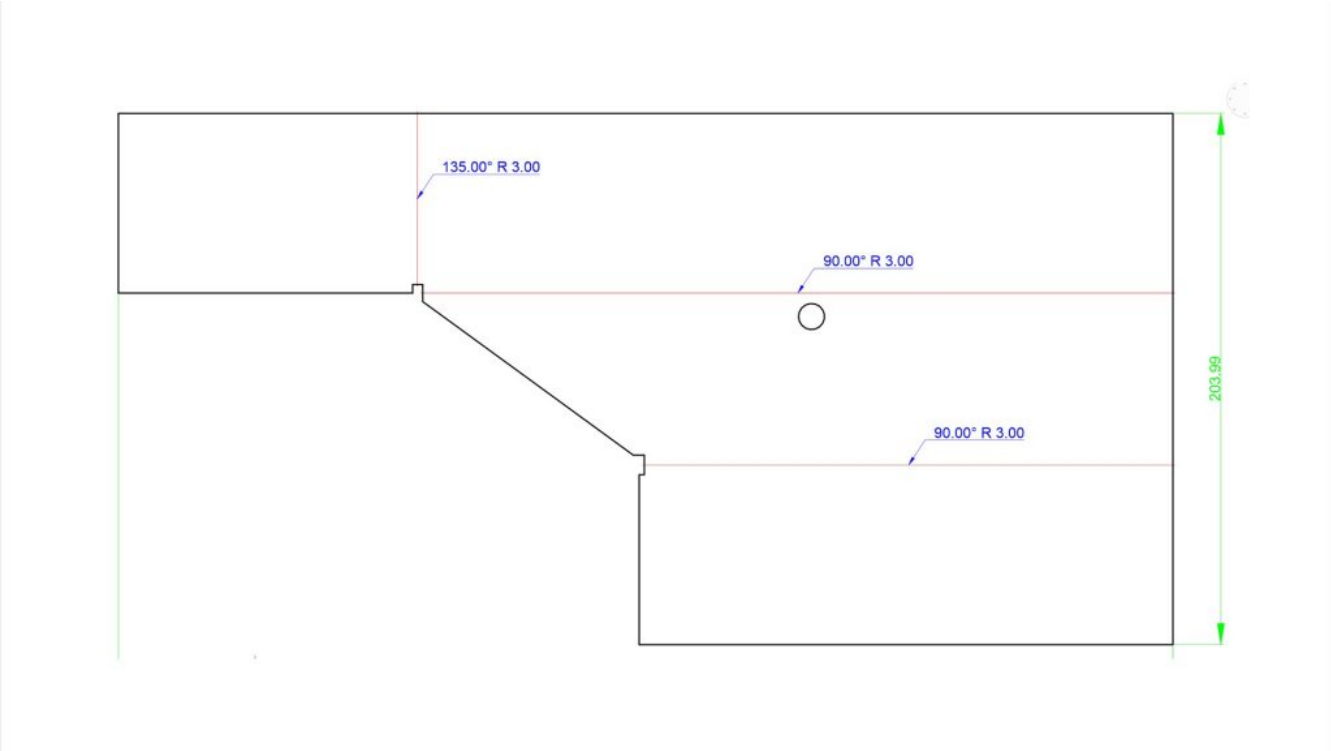
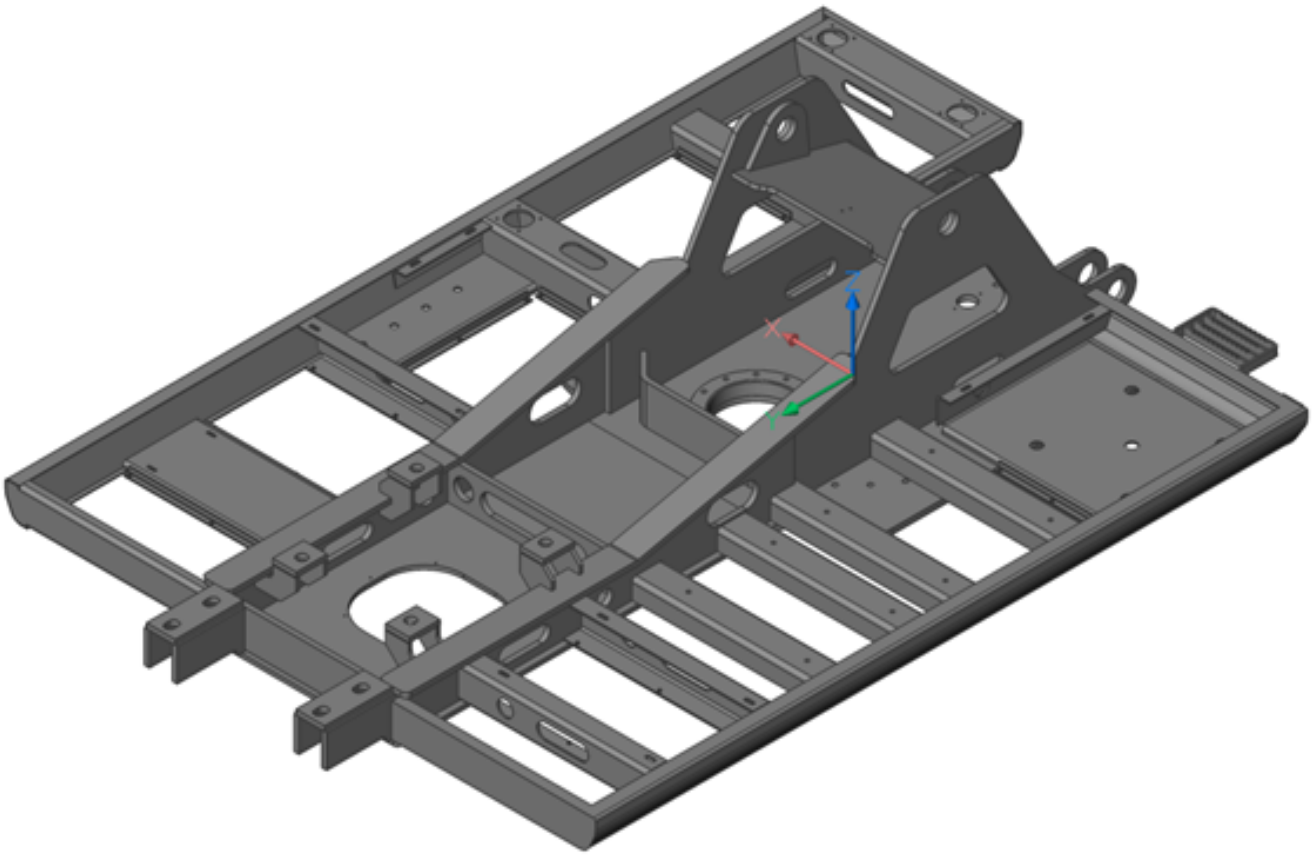


Figure 3.214



Figure 3.215



3.7.11 Poor sheet metal rework

1. You need **BricsCAD Platinum** and **Mechanical** licenses for this tutorial.
2. Open the **Upper_Frame_Assembly.dwg**file from Poor_Sheet_metal_rework_files.zip. **(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040957354/mceclip11.png})**
3. Hover over **Border_3**, click the **Open (6.23.8)** ( ) tool button on the Quad (2.2.3)| **Model**.

3.7.12 1. Rework the left side of the border

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041786353/mceclip10.png})

1. Click the **Convert to Sheet Metal (6.28.4)** ( ) tool button on the **Create** panel, of the **Sheet Metal** tab, on theRibbon (1.13.6). Hit **Enter**.
2. Click the **Smart (6.28.28) Split** ( ) tool button on the **Modify** panel, of the **Sheet Metal** tab, on the Ribbon (1.13.6).
3. Select the top flange and the **endpoint** in the corner. Hit the CTRL key twice to cycle through the options, then press **Enter** to accept the **Right**split. **(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360**
4. Repeat previous step on the bottom flange.
5. Click the **Create Bend (6.28.2)** ( ) tool button on **Modify** panel, of the **Sheet Metal** tab, on the Ribbon (1.13.6).
6. Select the top and bottom left edges. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/3600
7. Click the **Create Junction (6.28.18)** ( ) tool button on the **Modify** panel, of the **Sheet Metal** tab, on the Ribbon (1.13.6).
8. Select the left vertical face to create the curved junction: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/article

Figure 3.216

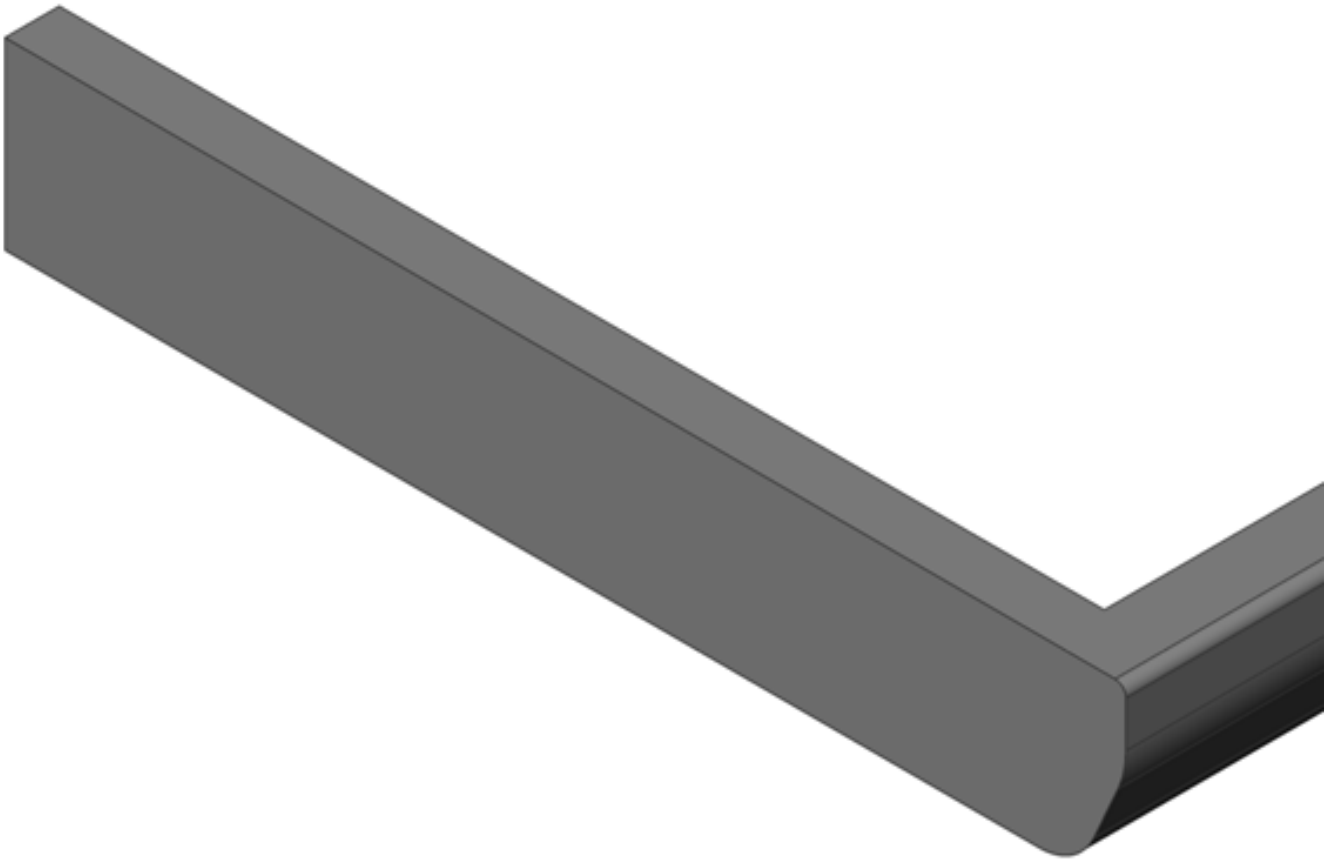


Figure 3.217

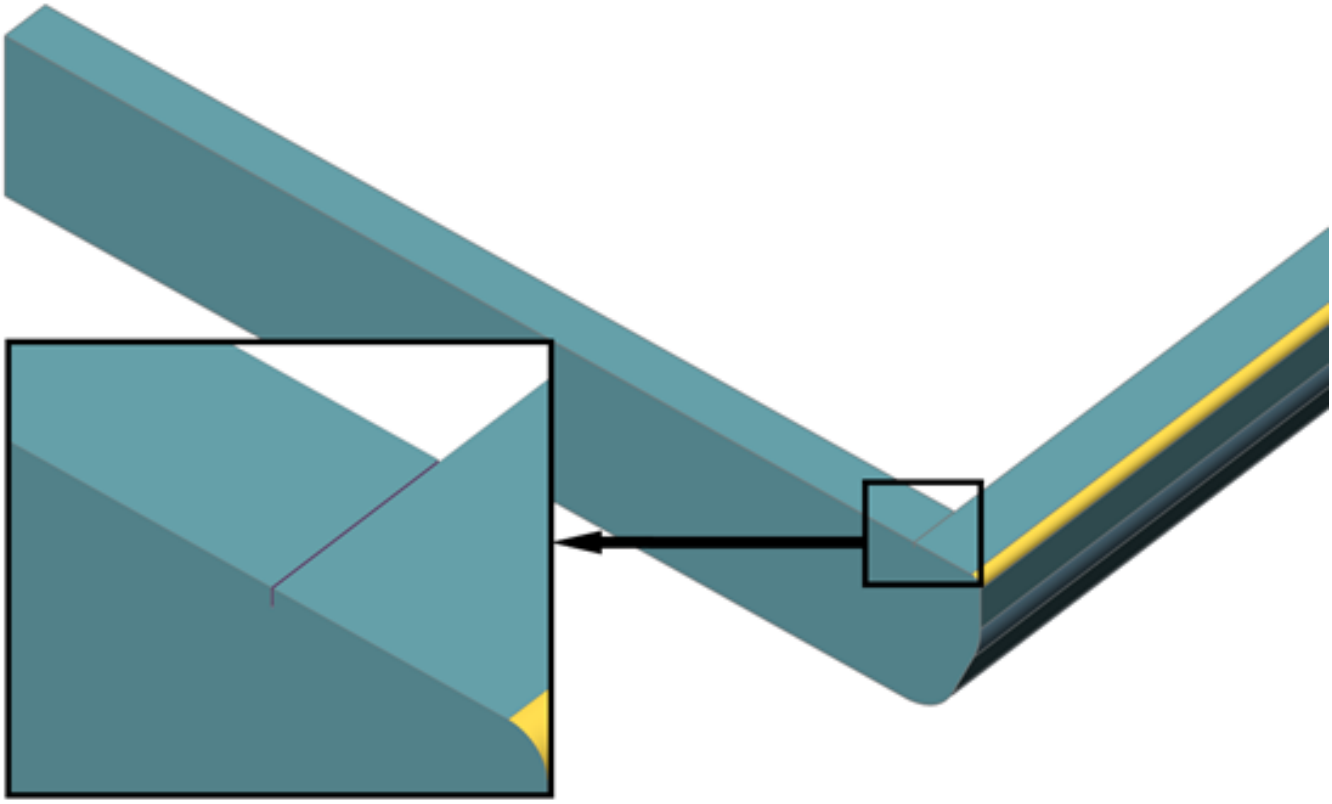


Figure 3.218

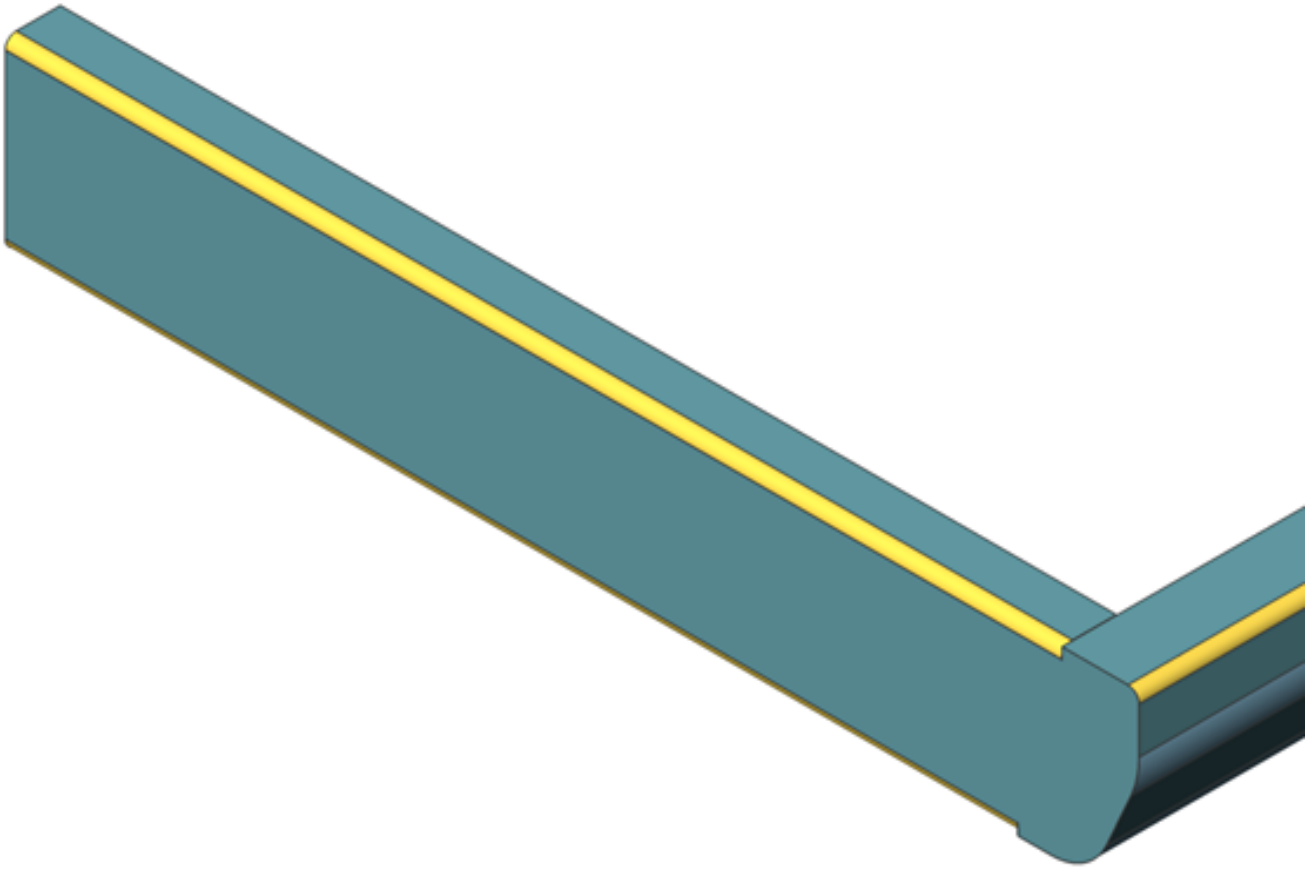


Figure 3.219

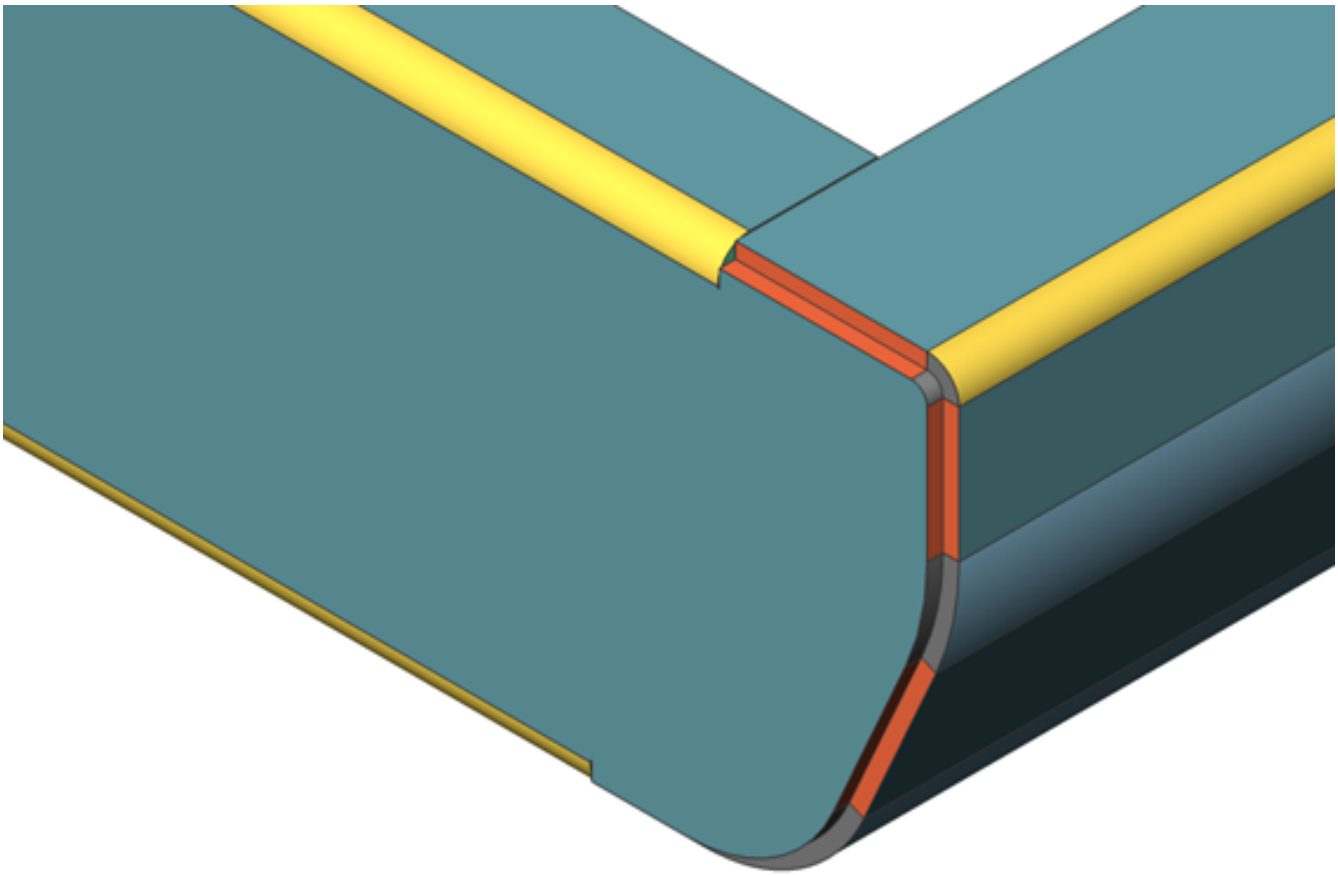
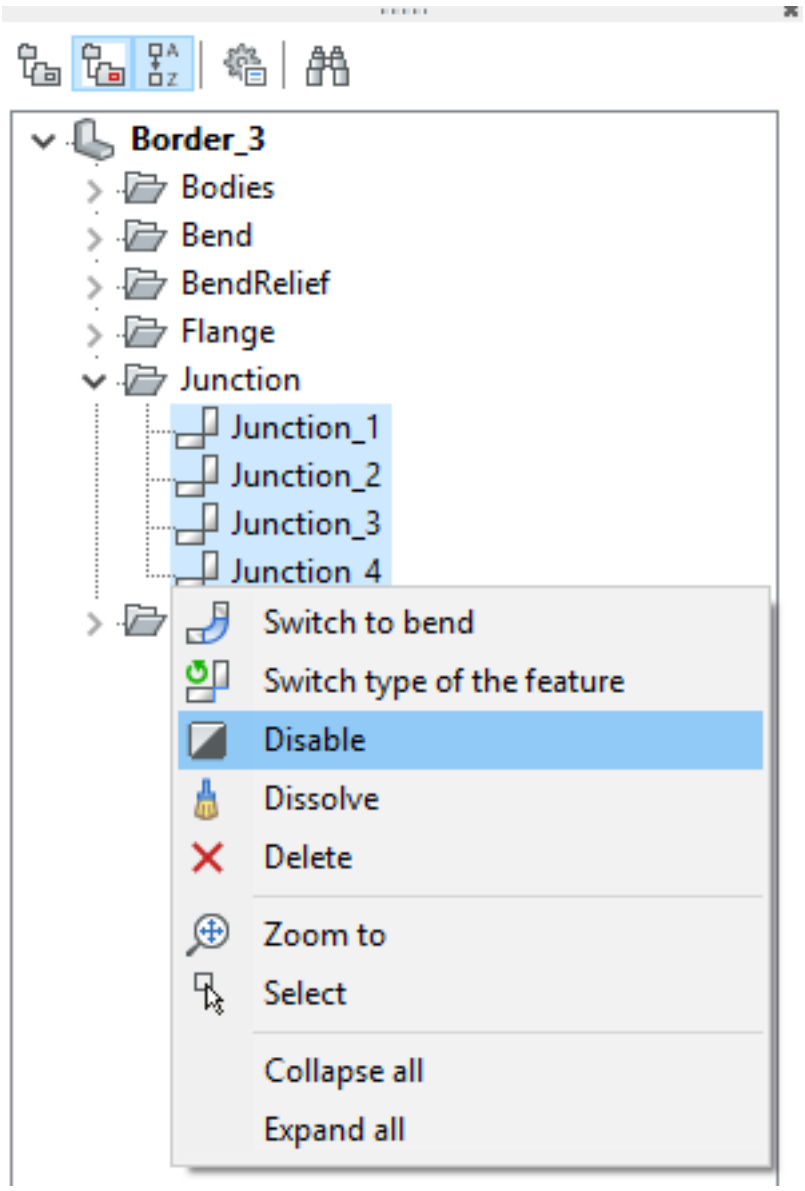


Figure 3.220



9. Select 4 junctions on Mechanical Browser (3.8.5) and click the **Disable** ( ) button on the context right-click menu (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041786013/mceclip7.png})

10. Hover over the right vertical junction face, click the **Smooth faces (6.12.26)** ( ) tool button, on the**Select** section of the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36004178583

11. Hover over the vertical junction face, click the **Push/Pull (6.12.20)** ( ) tool button, on the**Model** section of the Quad (2.2.3) and move the cursor to the **endpoint** (5mm) of the left vertical face.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041785653/mceclip4.png})

Figure 3.221

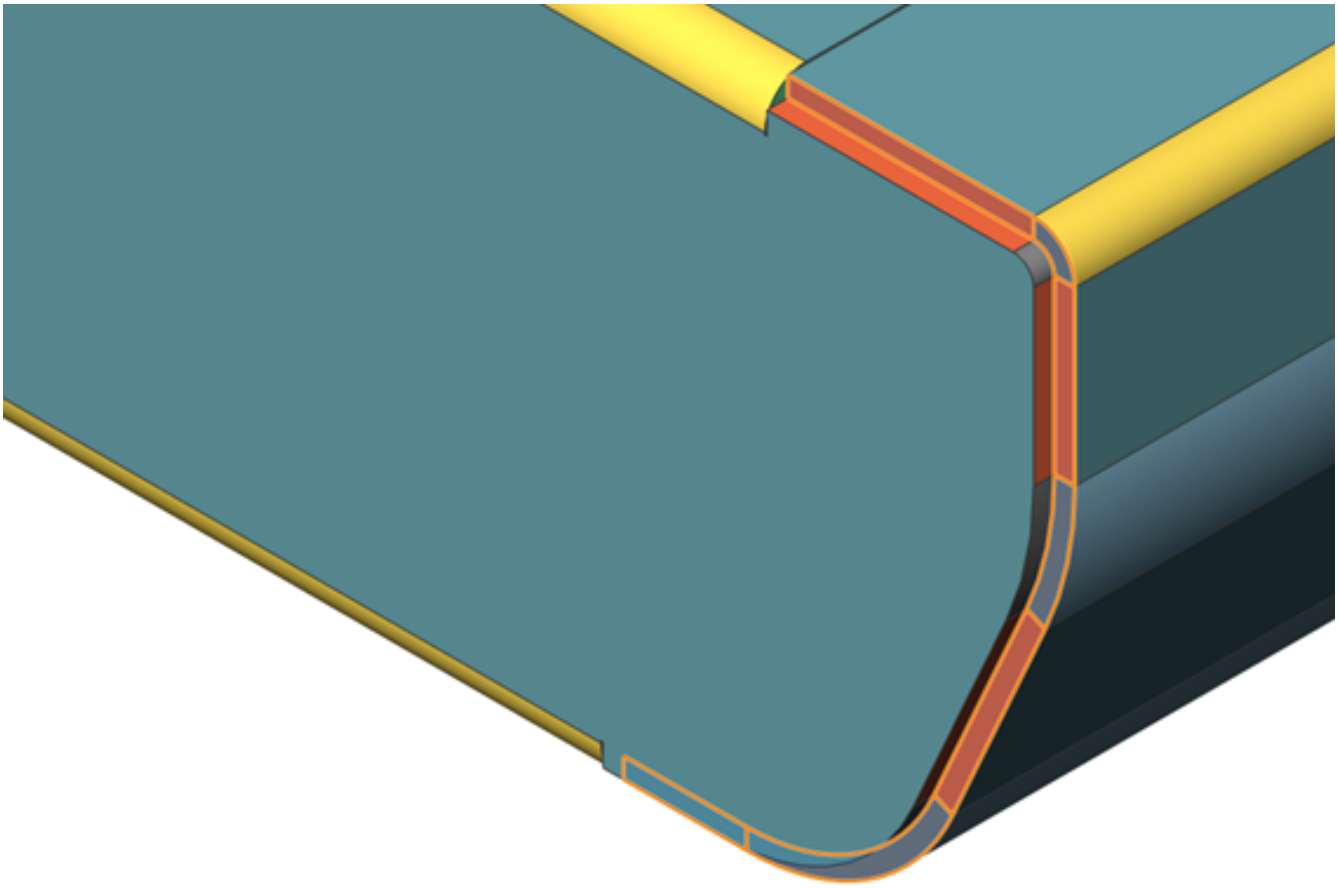
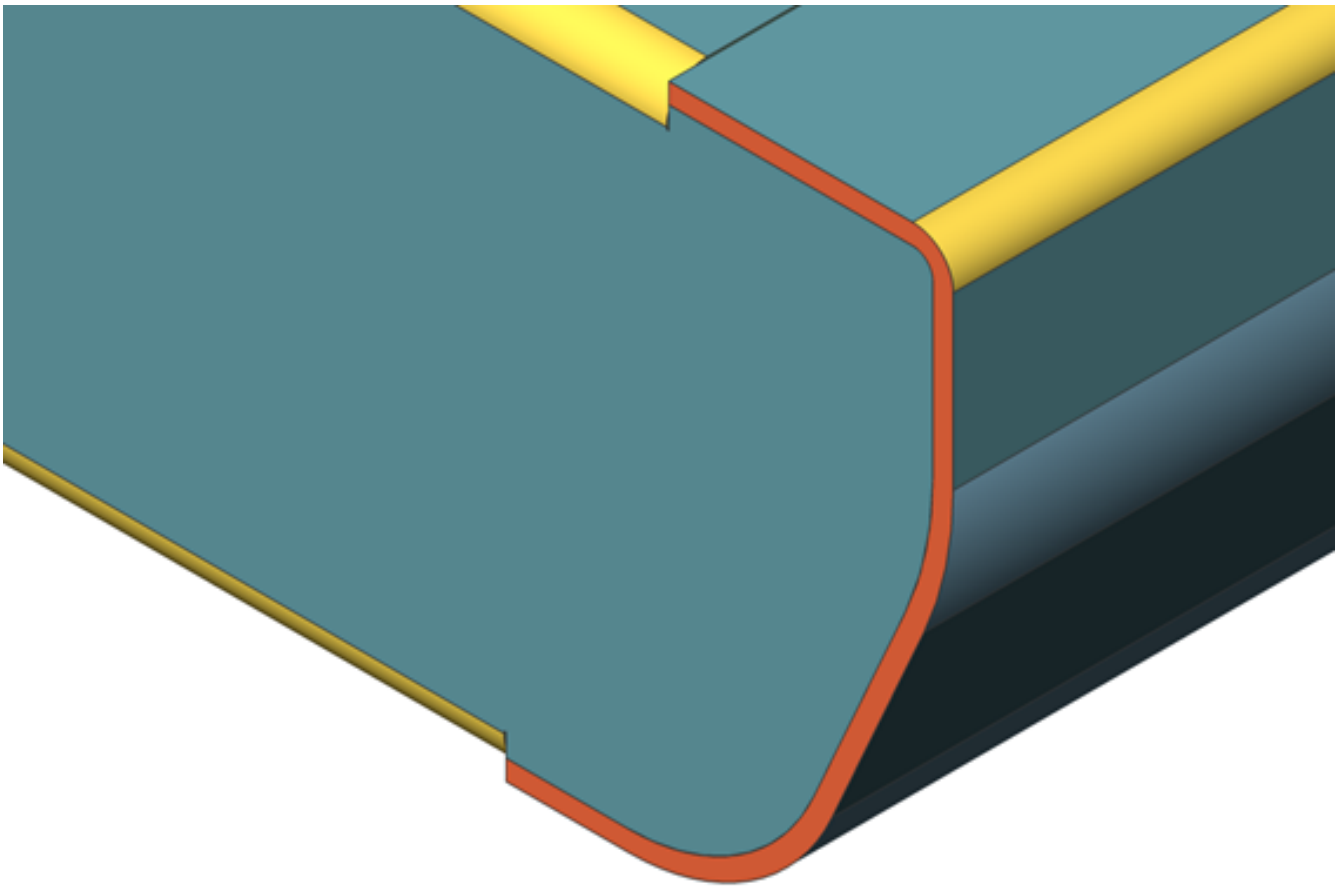


Figure 3.222

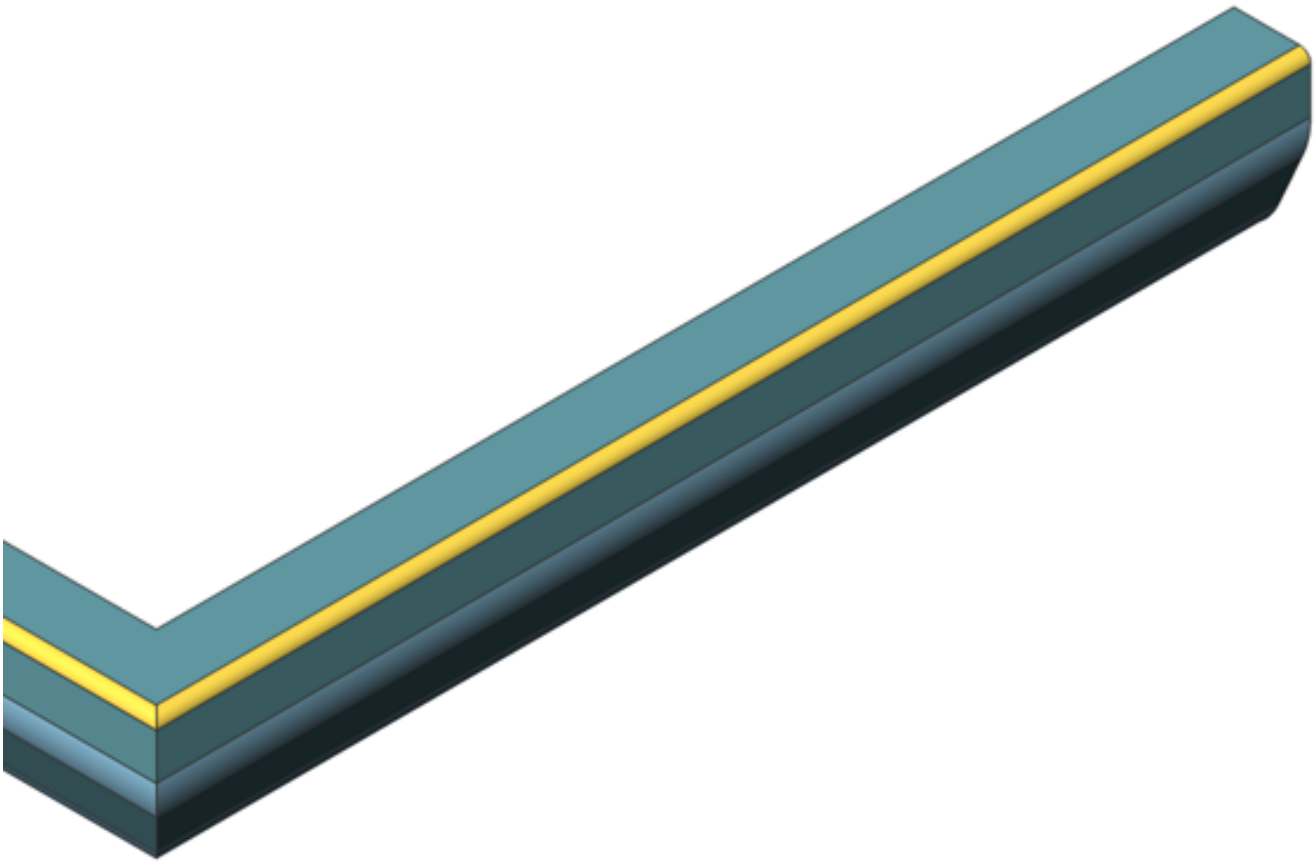


3.7.13 2. Rework the right side of the border

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041785553/mceclip3.png})

1. Click the **Dissolve (6.28.6)** () tool button on the **Modify** panel, of the **Sheet metal** tab, on the Ribbon (1.13.6). Hit **Enter**.
2. Click the **Slice (6.27.36)** () tool button on the **Edit** panel, of the **Solid** tab, on the Ribbon (1.13.6). Select the body, hit **Enter**. Specify 3 endpoints in the corner to specify a slicing plane and hit **Enter** to retain both parts. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040956754/mceclip2.png})

Figure 3.223



3. Click the**Convert to Sheet Metal (6.28.4)** ( )tool button on the **Create** panel, of the **Sheet Metal** tab, on theRibbon (1.13.6). Hit **Enter**.
4. Call the **Repair (6.28.24)** ( ) tool button on the **Heal** panel, on the **Sheet metal** tab, on the Ribbon (1.13.6).
5. Select the inner vertical face of the right sheet metal solid.
6. Repeat steps 2.3-2.5 to middle part of the border. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachr

7. Save the **Border_3.dwg** file.
8. Switch to the **Upper_Frame_Assembly.dwg**file.
9. Hover over **Border_3**, click the **Update (6.8.25)** ( ) tool button onthe **Model** section of the Quad (2.2.3).

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040956614/mceclip0.png})

Figure 3.224

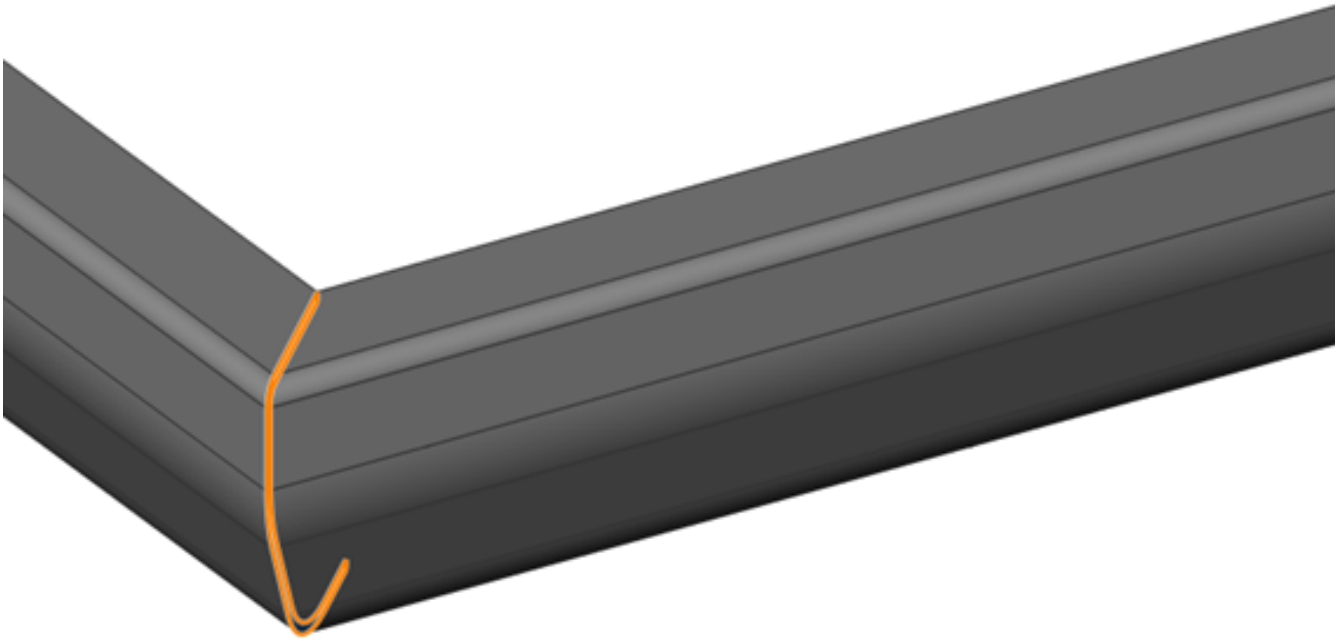


Figure 3.225

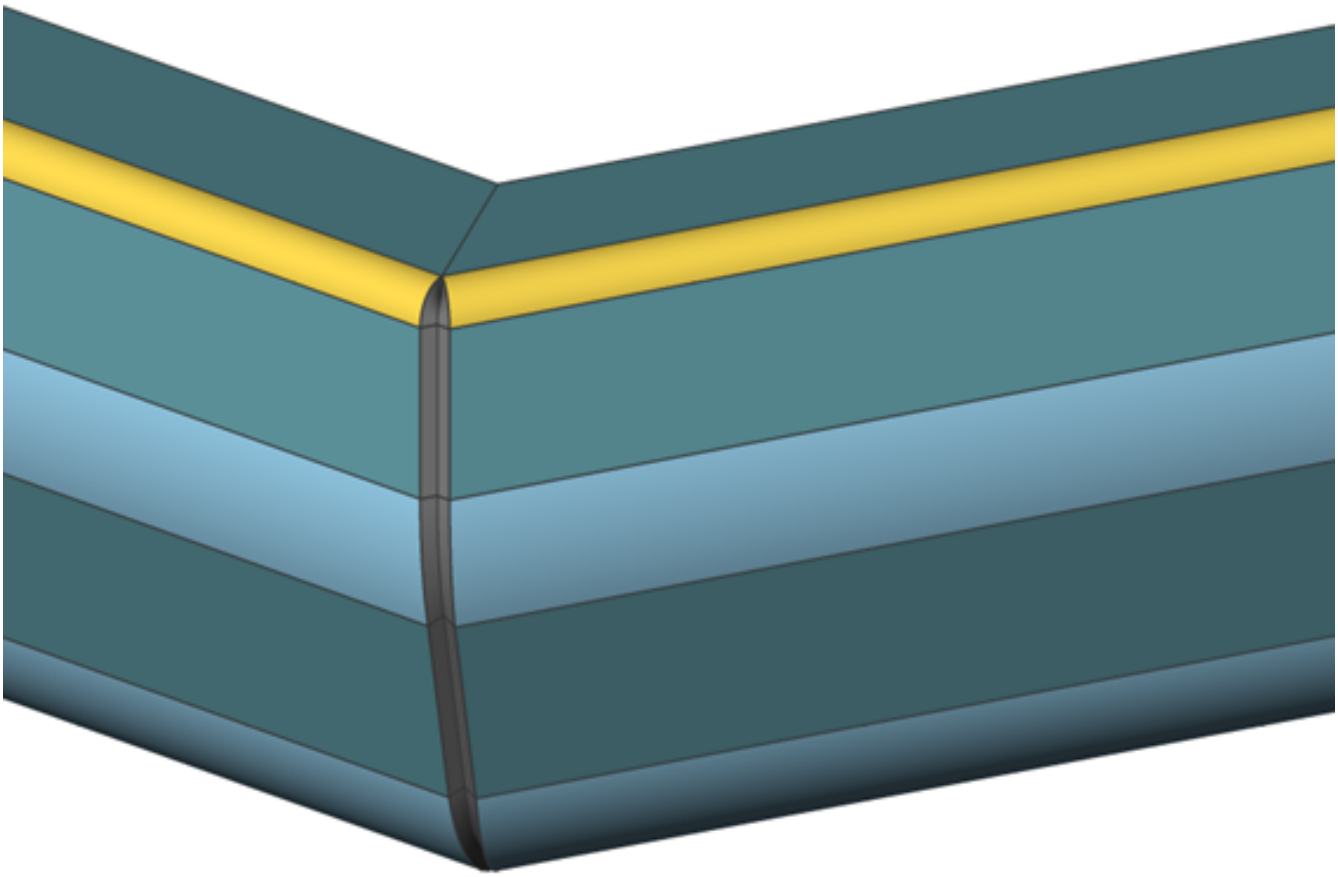


Figure 3.226

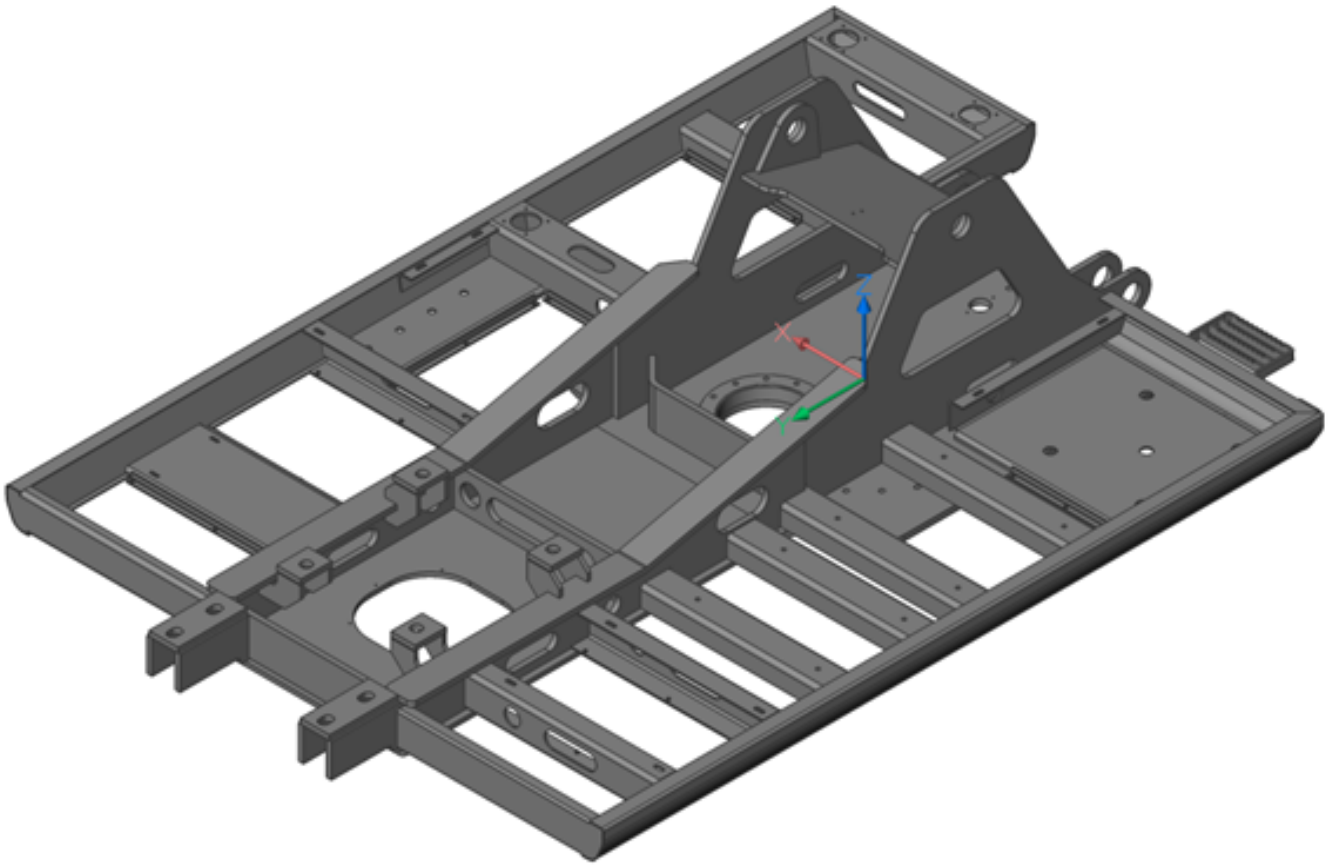
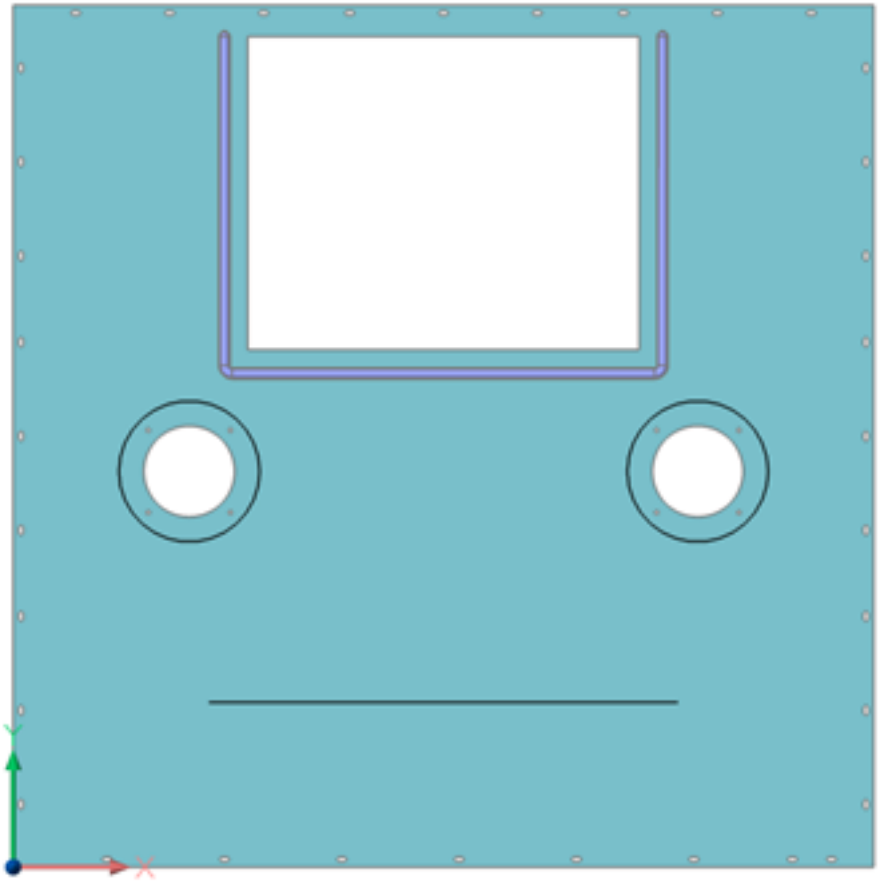


Figure 3.227



3.7.14 Sheet metal ribs

You need a**BricsCAD Platinum** and **Mechanical**licenses for this tutorial.

1. Open the **Roof.dwg**file.
2. Click the **Create Rib Features (6.28.26)** ( ) tool button, in the **Modify** section of the **Sheet Metal** tab on the **Ribbon (1.13.6)**.
3. Select the flange face.
4. Select the U-shaped profile.
5. Hit **Enter** to complete the command. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/3600
6. Click the**Create Rib Features (6.28.26)** ( )button, in the **Modify** section of the **Sheet Metal** tab on the **Ribbon (1.13.6)**.
7. Select the flange face.
8. Click the **Profile radius** option on **SmRibCreate** menu.
9. Enter **6** as the new profile radius.
10. Select the two circles and the spline using a Blue Window-Inside Selection Box (1.23.6)**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041535373/mceclip1.png})**
11. Hit **Enter** to complete the command.
12. In theMechanical Browser (3.8.5)select **rib by spline (Rib_2)**.
13. Change the **Round radius** in the Mechanical Browser (3.8.5)to **4 mm**.
14. Select the spline using a **Blue Selection window**.
15. Click the **Midpoint** of the spline and STRETCH (1.23.4) down 100. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/h
16. Hover over the top flange face.
17. Click the **Unfold body (6.28.29)** ( ) button on the**Sheet Metal** section of the **Quad (2.2.3)**.
18. Choose the position for the unfolded body.
19. Hit **Enter** to complete the command.

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041535753/mceclip3.png})

Figure 3.228

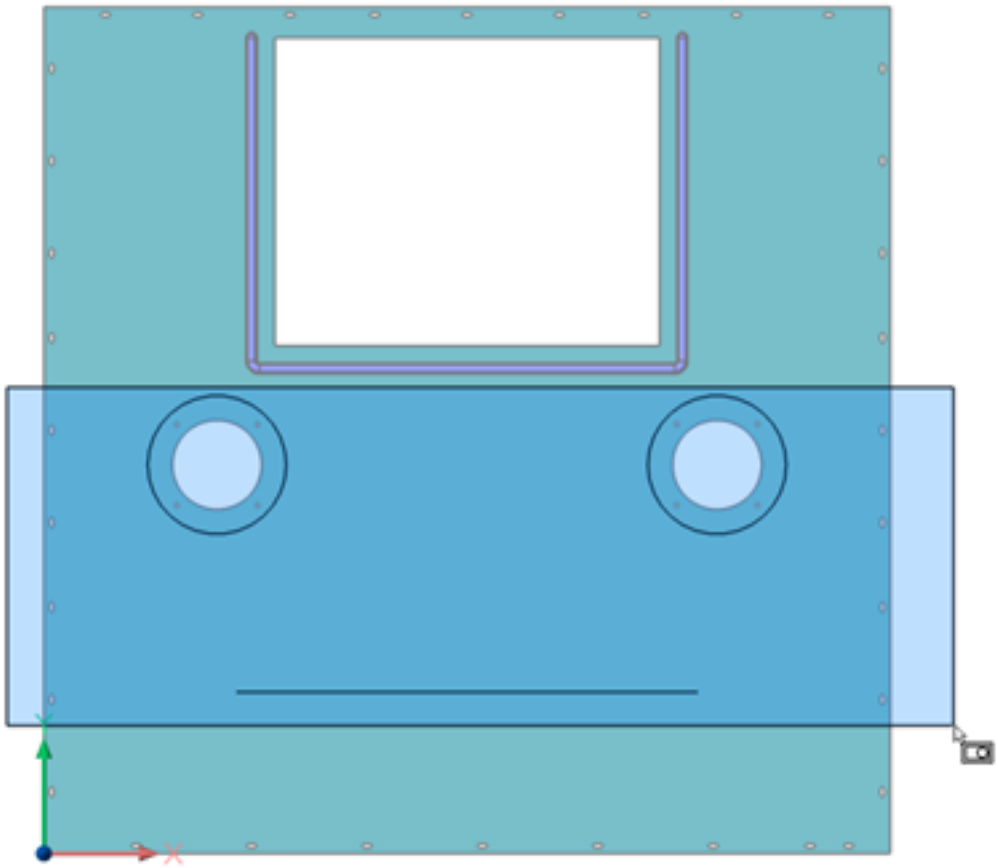


Figure 3.229

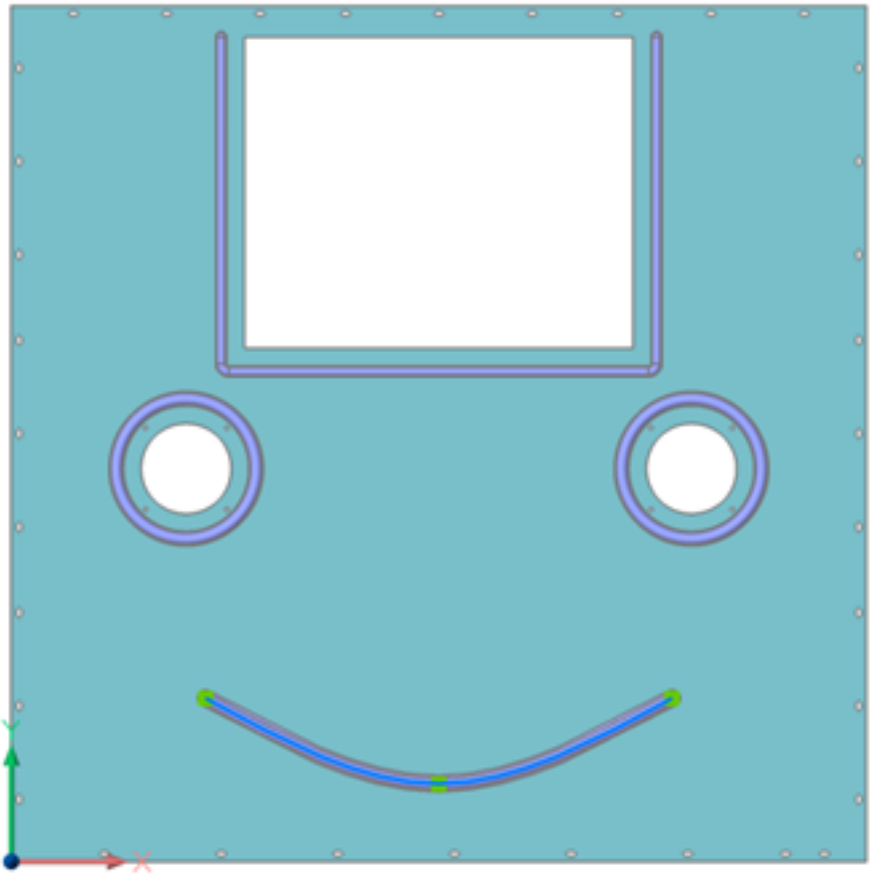
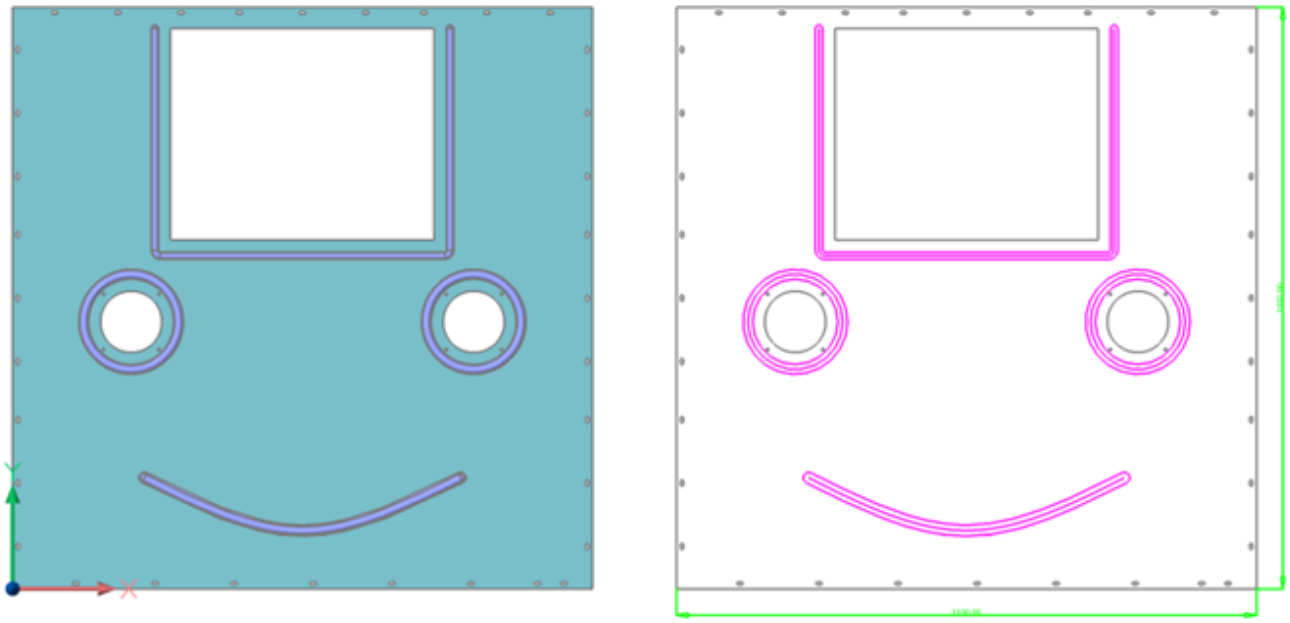


Figure 3.230



3.7.15 Sheet metal from solid

3.7.16 1. Solidcreation

You need **BricsCAD Platinum** and **Mechanical**licenses for this tutorial.

1. Open the **Geared_Traction_Machine_assembly.dwg** file: Sheet_metal_from_solid_files.zip. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041545393/mceclip14.png})
2. Make sure that the **UCS orthographic** option in the Settings (6.27.23)dialog is switched on (the UCSORTHO system variable is 1). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360039137253/mce
3. Switch off **SmSmartFeatures** variable on the Settings (6.27.23)dialog.



4. Switch to **Back** view on the LookFrom (6.20.39)widget.
5. Click the **Polyline (6.24.27)** ( ) tool button on **Solid** | **Draw** panel of the Ribbon (1.13.6). Draw a polyline that covers everything on the back view. Start from the left bottom Endpoint of the support plate of **Shaft_support** (use snap to Endpoint (6.13.7) ( )). Draw **45** back along the x-axis, **700** on the y-axis, **700** on the x-axis, **450** at an angle of **55** on the x-axis, down to the intersection with the frame (use snap to Perpendicular (6.24.22) ( )), to close the polyline. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36004071473



6. Switch to the **Right** view on the LookFrom (6.20.39)widget.
7. Hover over the new polyline, select **Move (6.12.17)** ( ) from the **Model** section of the Quad (2.2.3). Enter **0,0,0** as the base point, and **1000,0,0** as the second point. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/artic
8. Click the**Polyline (6.24.27)** ( )tool button on **Solid** | **Draw** panel of the Ribbon (1.13.6) to draw a polyline that covers everything on the right view. Start from the left bottom Endpoint of the frame (use snap toEndpoint (6.13.7) ( )). Draw **250** on the y-axis, **450** at an angle of **55** on the x-axis, **1000** at an angle of **10** on the x-axis, **700** at an angle of **80** on the x-axis, down to the intersection with the frame (use snap toPerpendicular (6.24.22) ( )), to close the polyline. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/3600



9. Switch to the **Top** view on the LookFrom (6.20.39)widget to activate **World UCS**.
10. Select both polylines. Select **Extrude (2.4.11)** ( ) from the **Model** section of the Quad (2.2.3). Create two intersecting solids from the two profiles with the **Create**option. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/a
11. Select both intersecting solids. Click the **Intersect (6.17.19)** ( ) tool button on **Solid** | **Edit** panel of the Ribbon (1.13.6)**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041545333/mceclip6.**

Figure 3.231

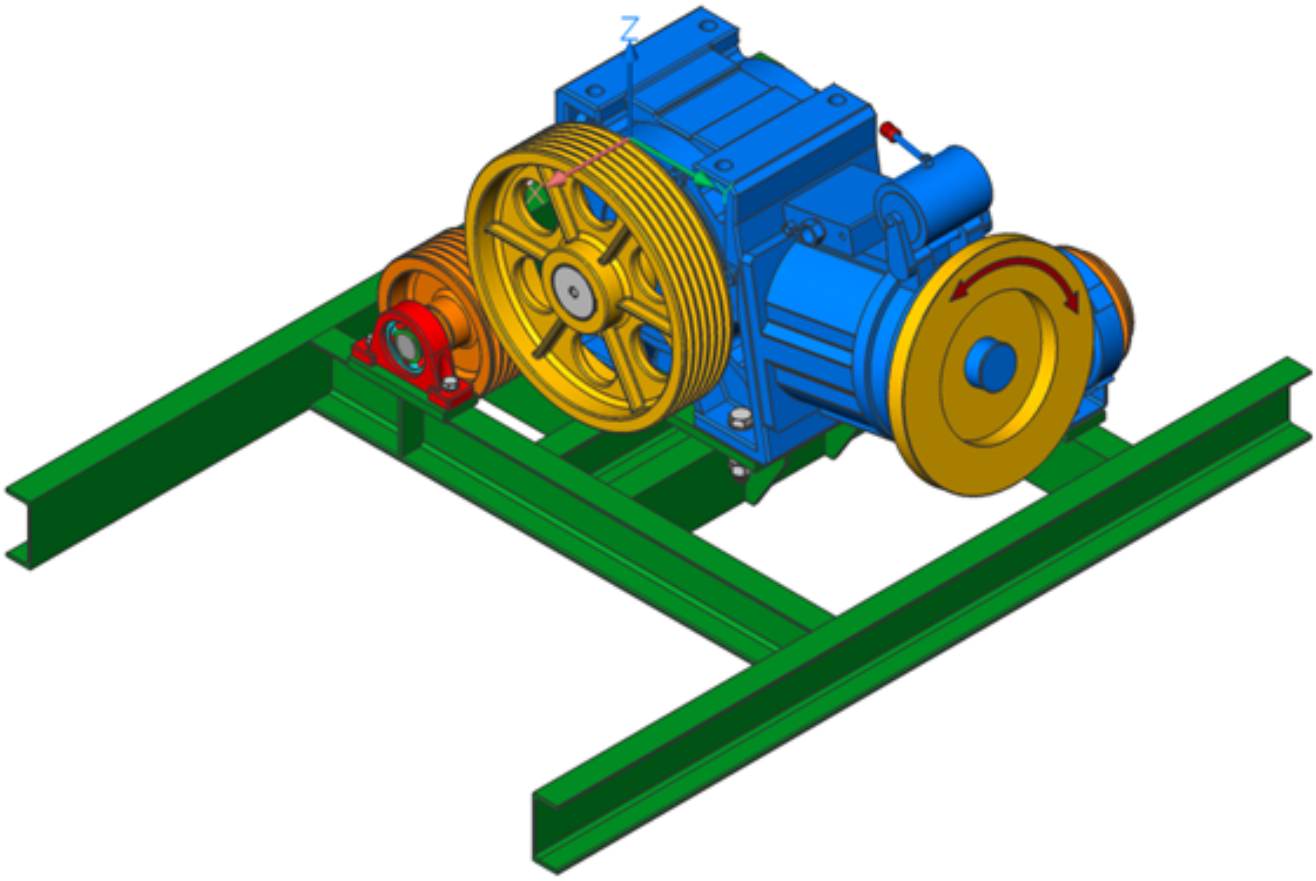


Figure 3.232

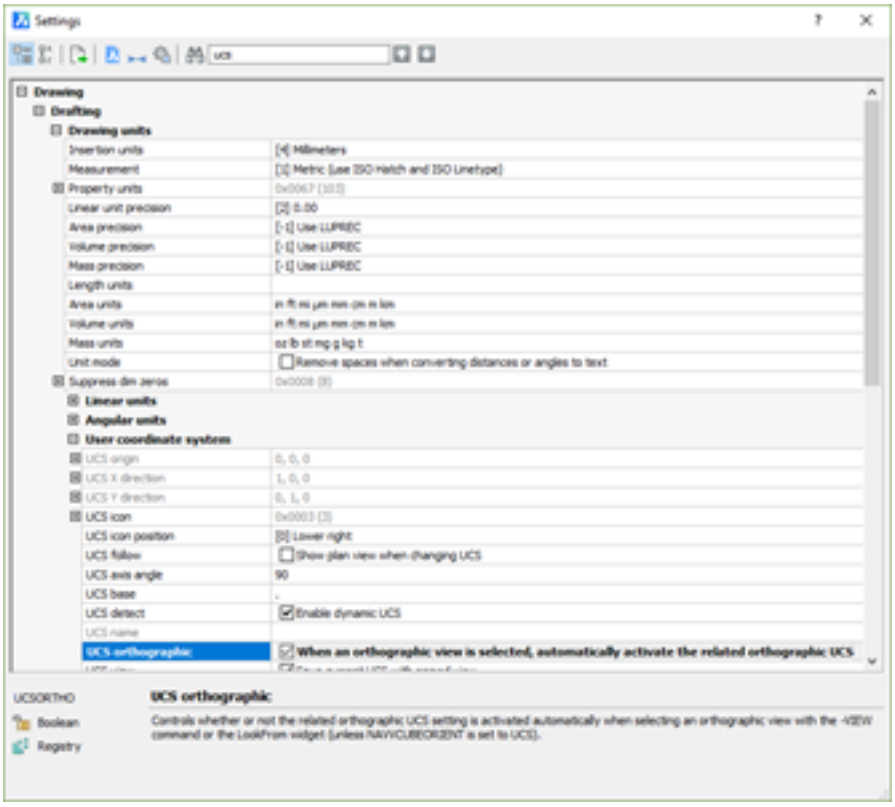


Figure 3.233

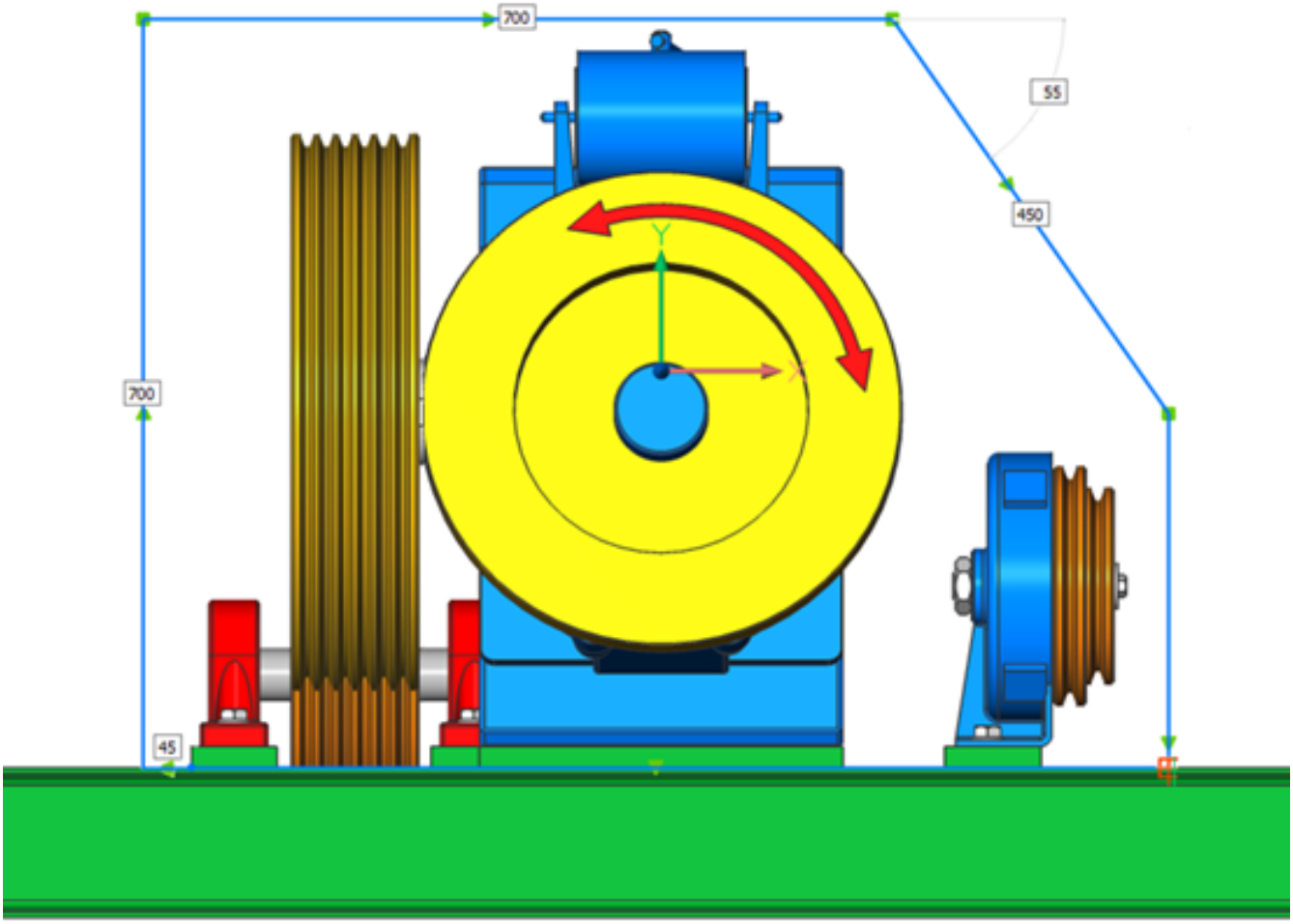


Figure 3.234

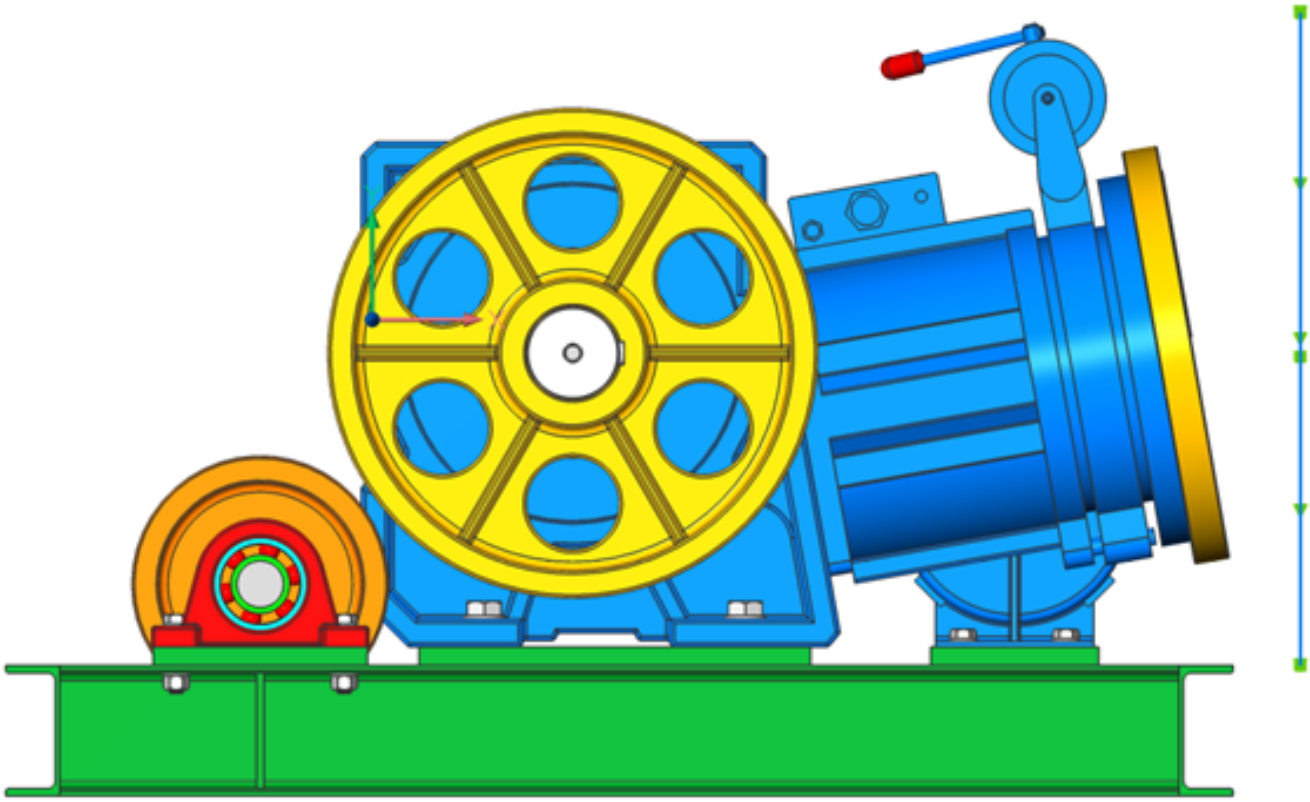


Figure 3.235

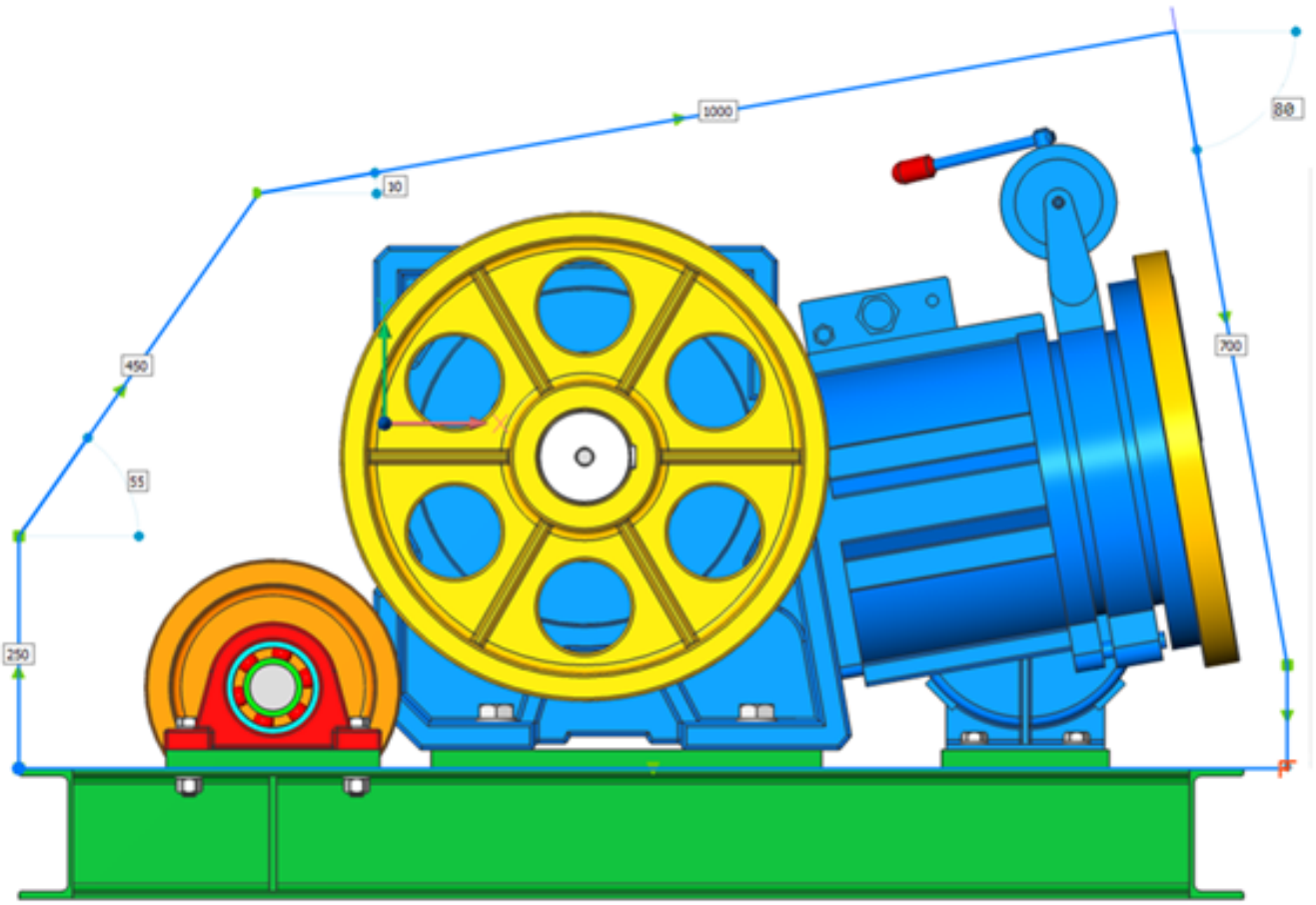


Figure 3.236

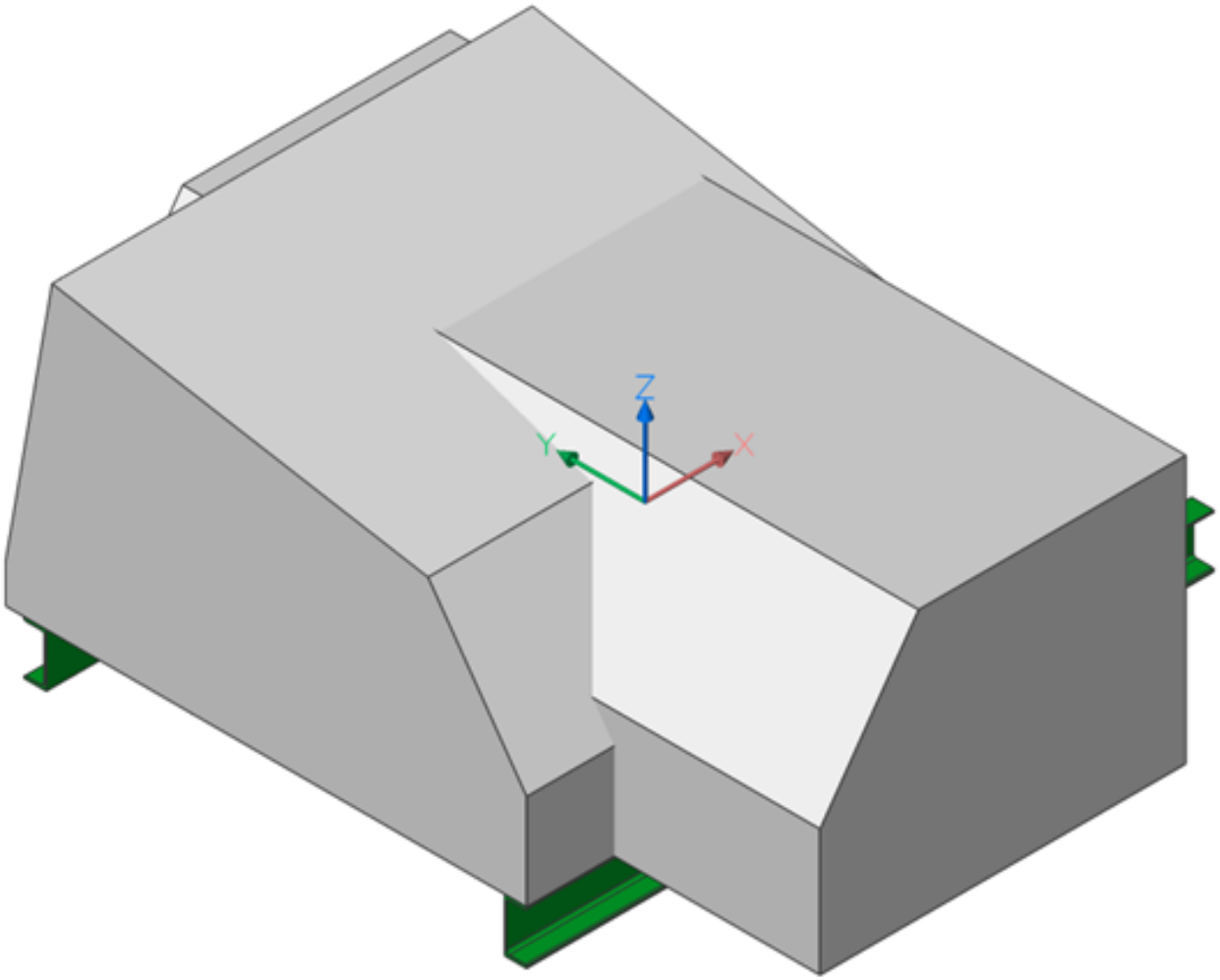
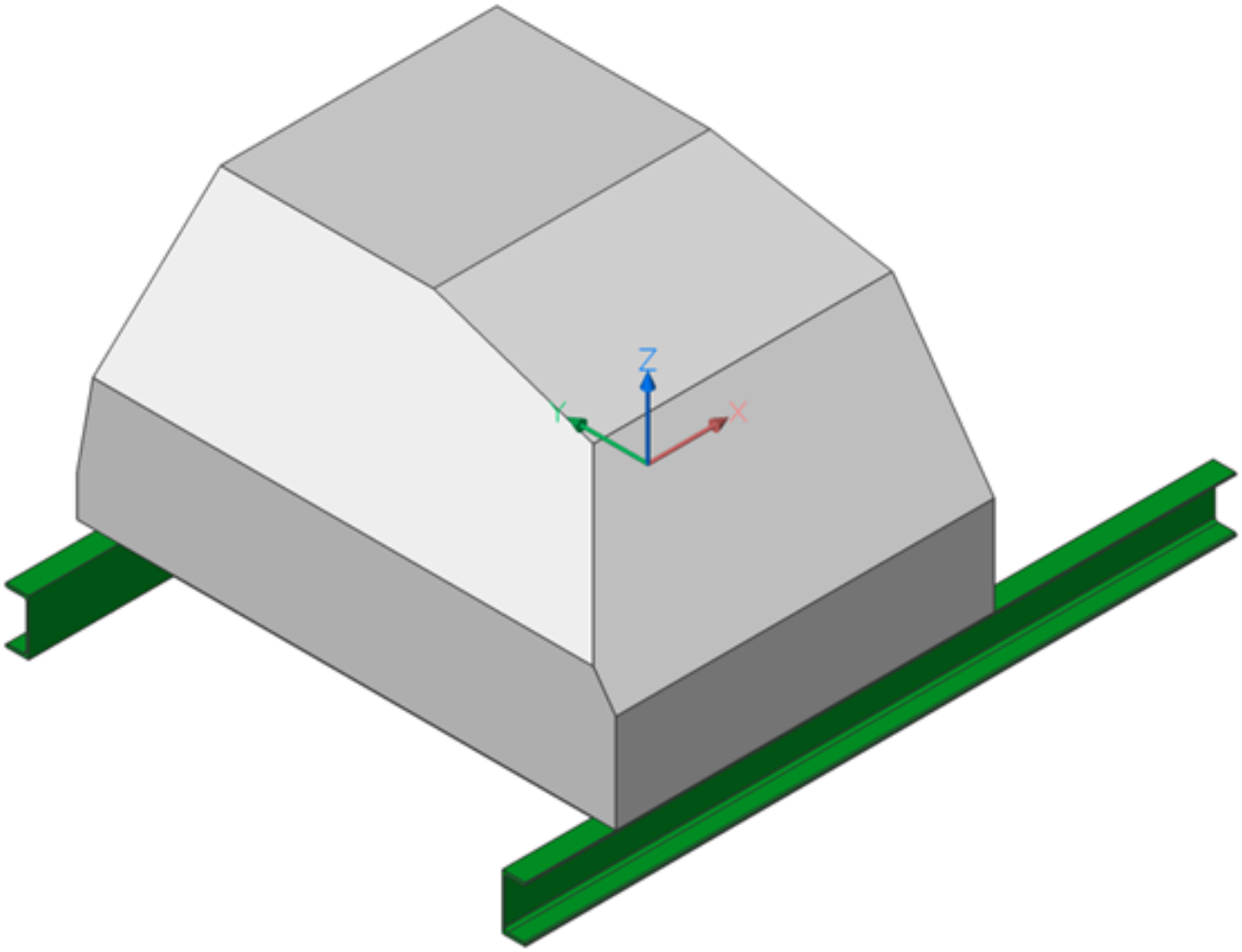


Figure 3.237



12. Hover over the new solid. Select **Isolate (6.17.21)** ( ) from the **Select** section of the Quad (2.2.3)**.**

3.7.17 2.Solid to Sheet Metal conversion

1. Click the **Shell (6.27.39)** ( ) tool button on **Solid** | **Edit** panel of the Ribbon (1.13.6). Select the solid. Select the bottom face (to remove). Enter a shell offset distance of **2**. **Enter**. **Enter**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041545413/mceclip15.png})
2. Hover over the right vertical face and select **Offset (2.4.15)** ( ) from the **Draw** section of the Quad (2.2.3). Enter a distance of **100**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041545293/m
3. Hover over the polyline. Select**Extrude (2.4.11)** ( )from the **Model** section of the Quad (2.2.3). Use the **Subtract** option to create an opening.
4. Select the horizontal face of the bottom opening (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachm

Figure 3.238

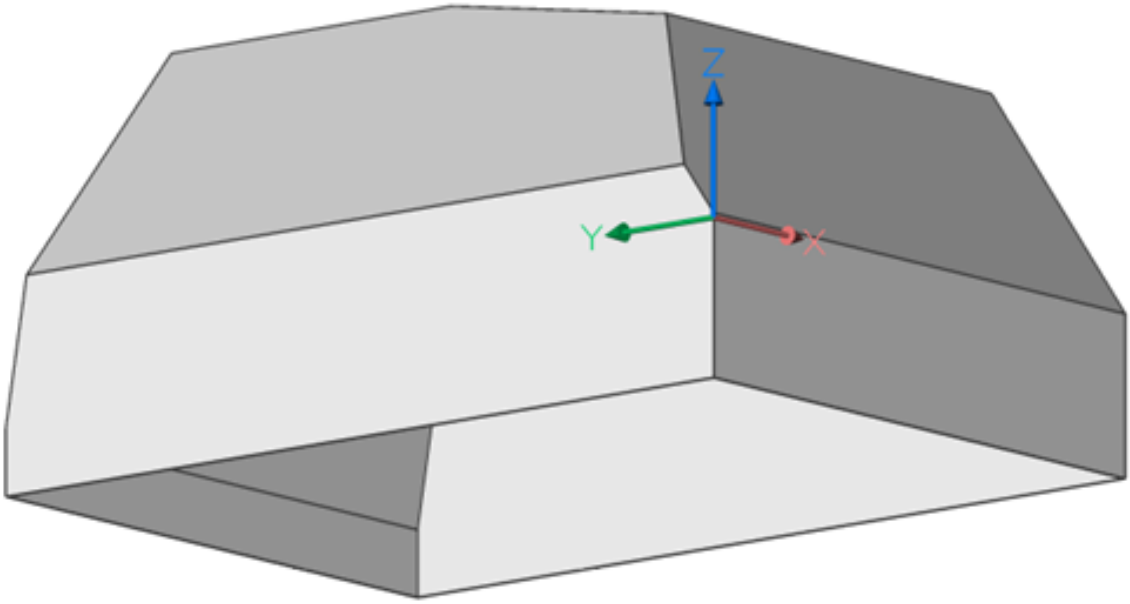
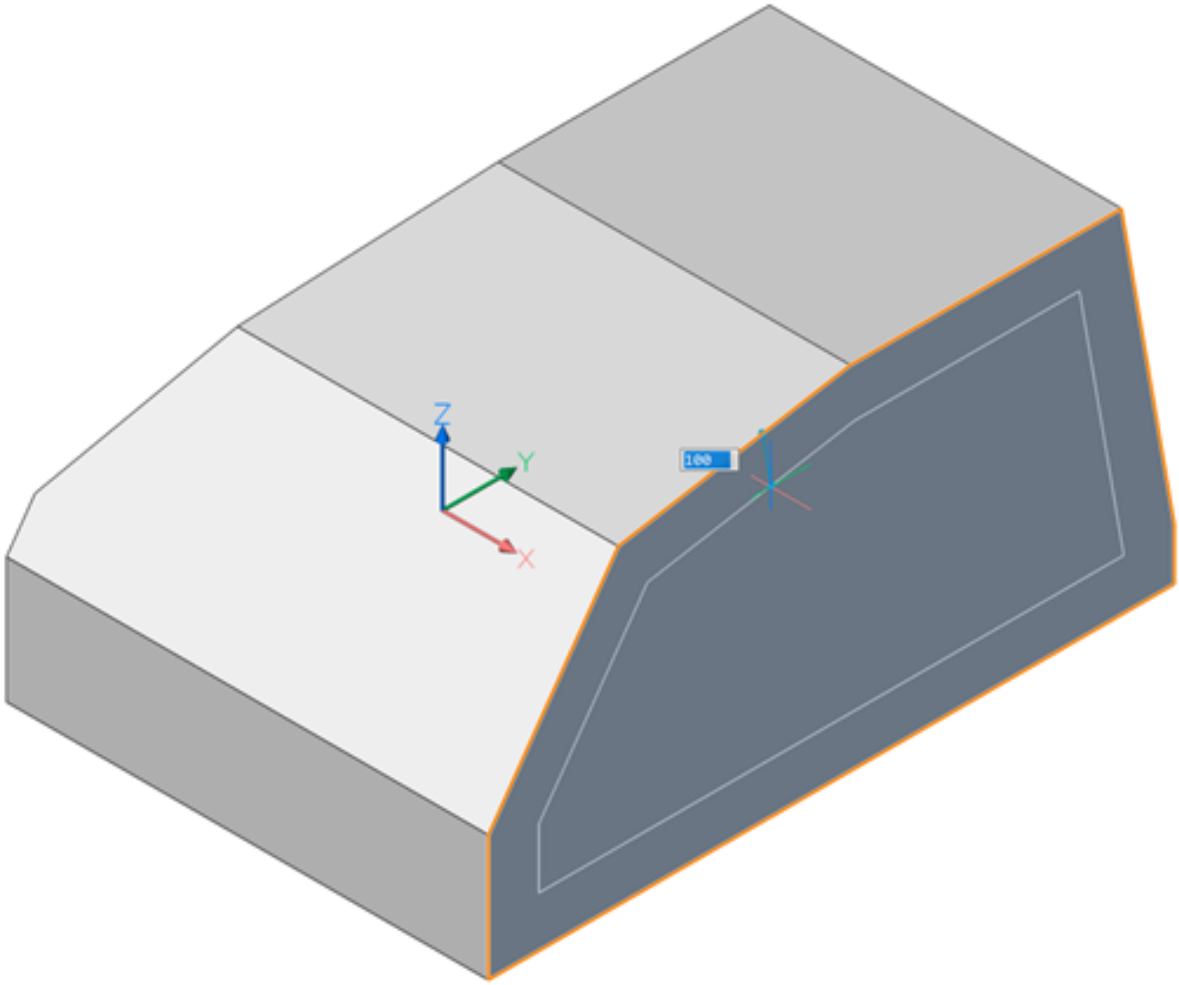


Figure 3.239



5. Select**Extrude (2.4.11)** ( )from the **Model** section of theQuad (2.2.3). Use the **Subtract** option. Move the cursor down to increase the size of the opening. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachm
6. Click the **Convert to Sheet Metal (6.28.4)** ( ) tool button on the **Sheet Metal** | **Create** panel of the Ribbon (1.13.6). Hit **Enter**.
7. Click the **Create Relief (6.28.22)** ( ) tool button on the **Sheet Metal** | **Modify** panel of the Ribbon (1.13.6). Hit **Enter**.
8. Select edges as shown below. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041545
9. Click the **Create Junction (6.28.18)**( )tool button on the**Sheet Metal** | **Modify**panel of the Ribbon (1.13.6).
10. Click the **Create (6.28.2) Bend (6.28.2)** ( ) tool button on the**Sheet Metal** | **Modify**panel of the Ribbon (1.13.6). Hit **Enter**.
11. Switch to the **Right** view on the LookFrom (6.20.39)widget.
12. Use the **Smart Split (6.28.28)** ( ) tool from the**Sheet Metal** | **Modify**panel of the Ribbon (1.13.6)to create miters on all the corners of the opening: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36

Figure 3.240

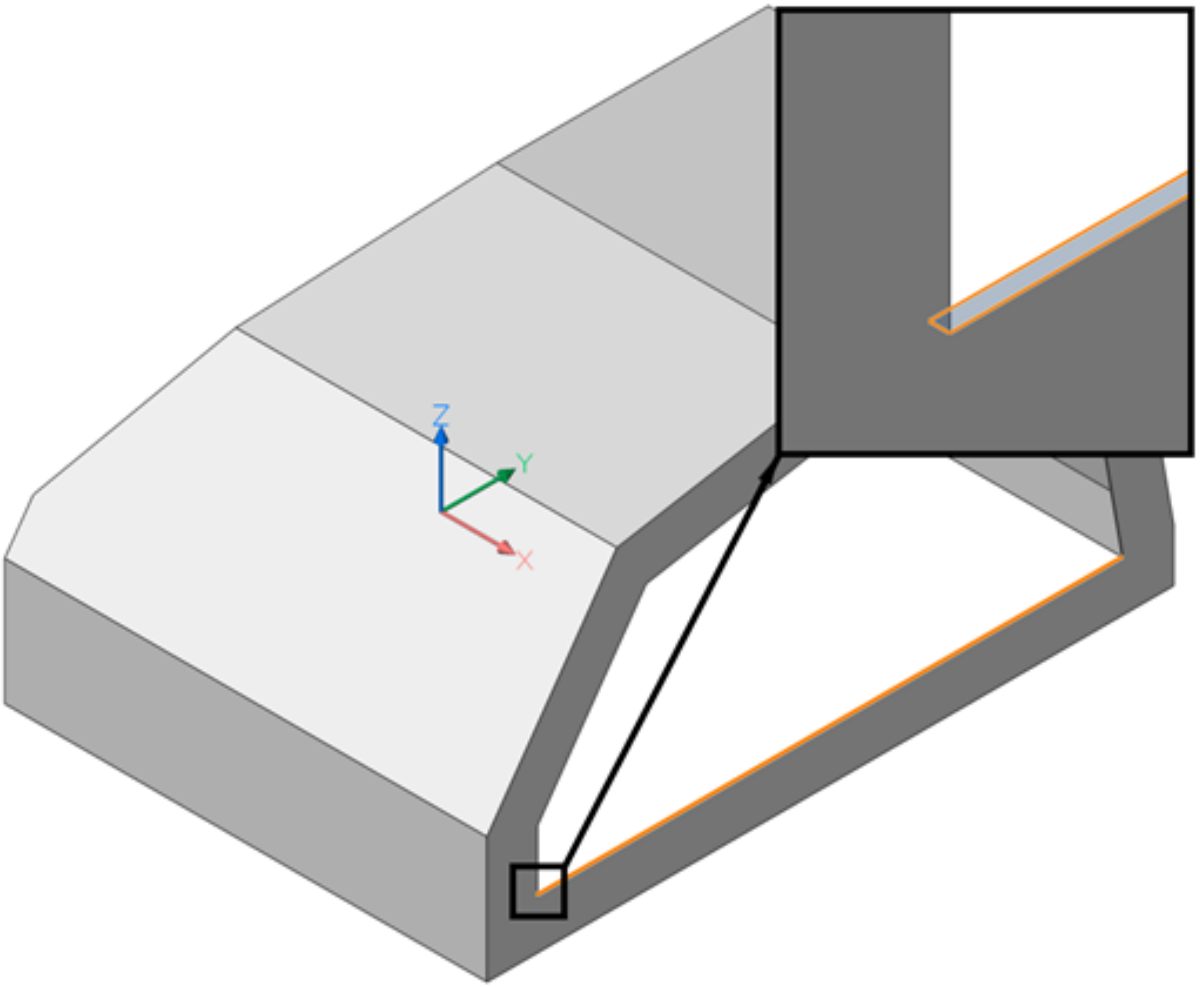


Figure 3.241

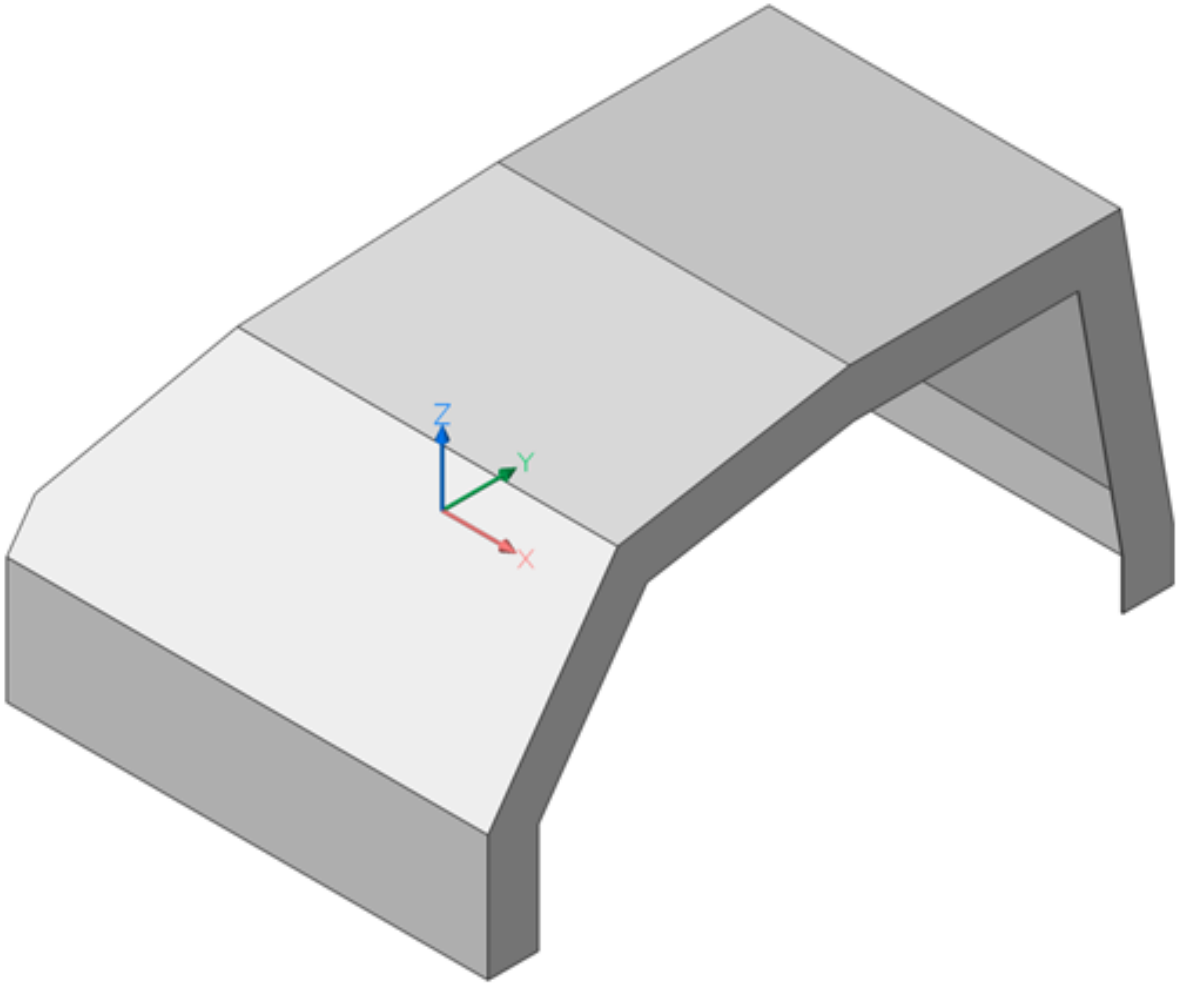


Figure 3.242

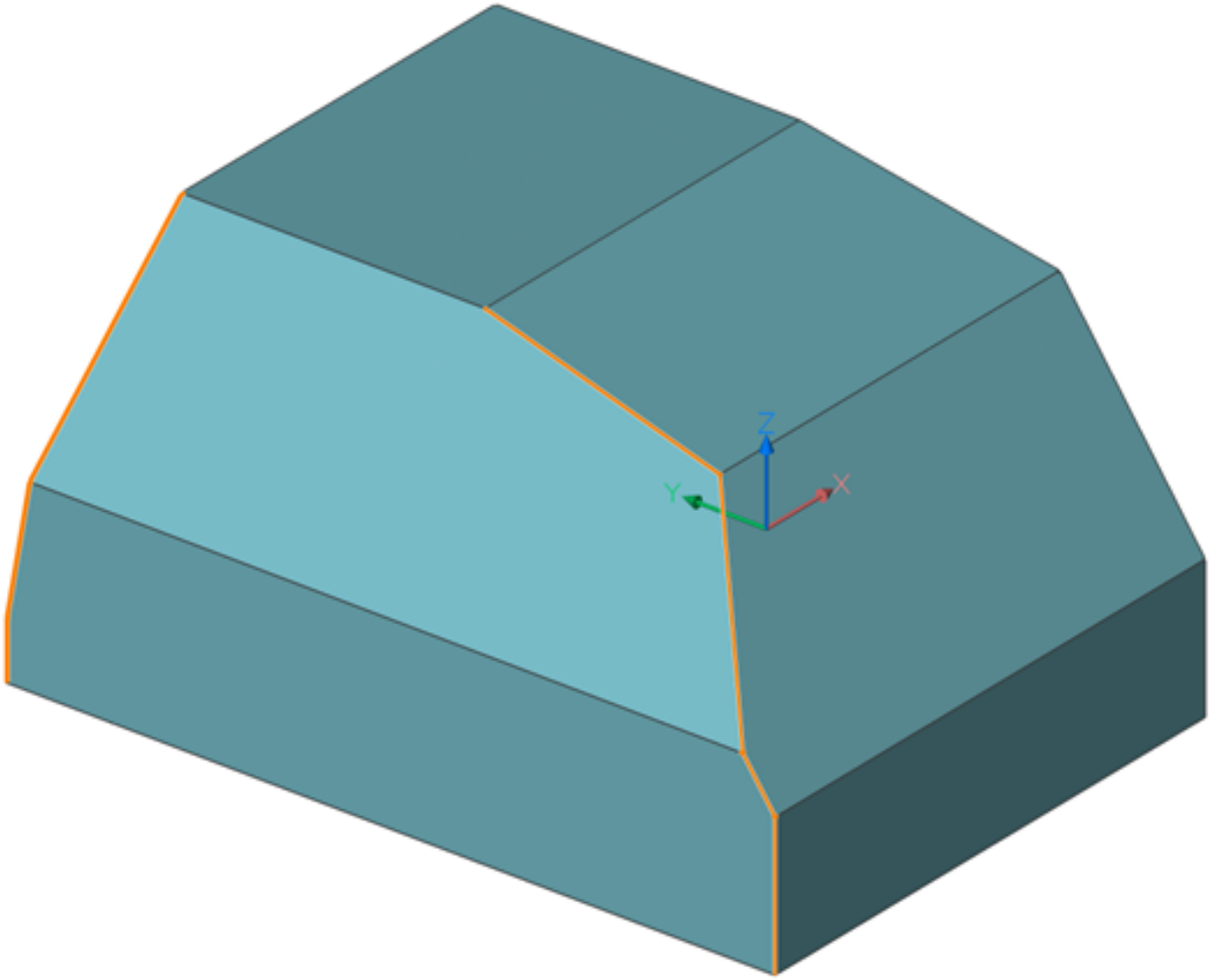
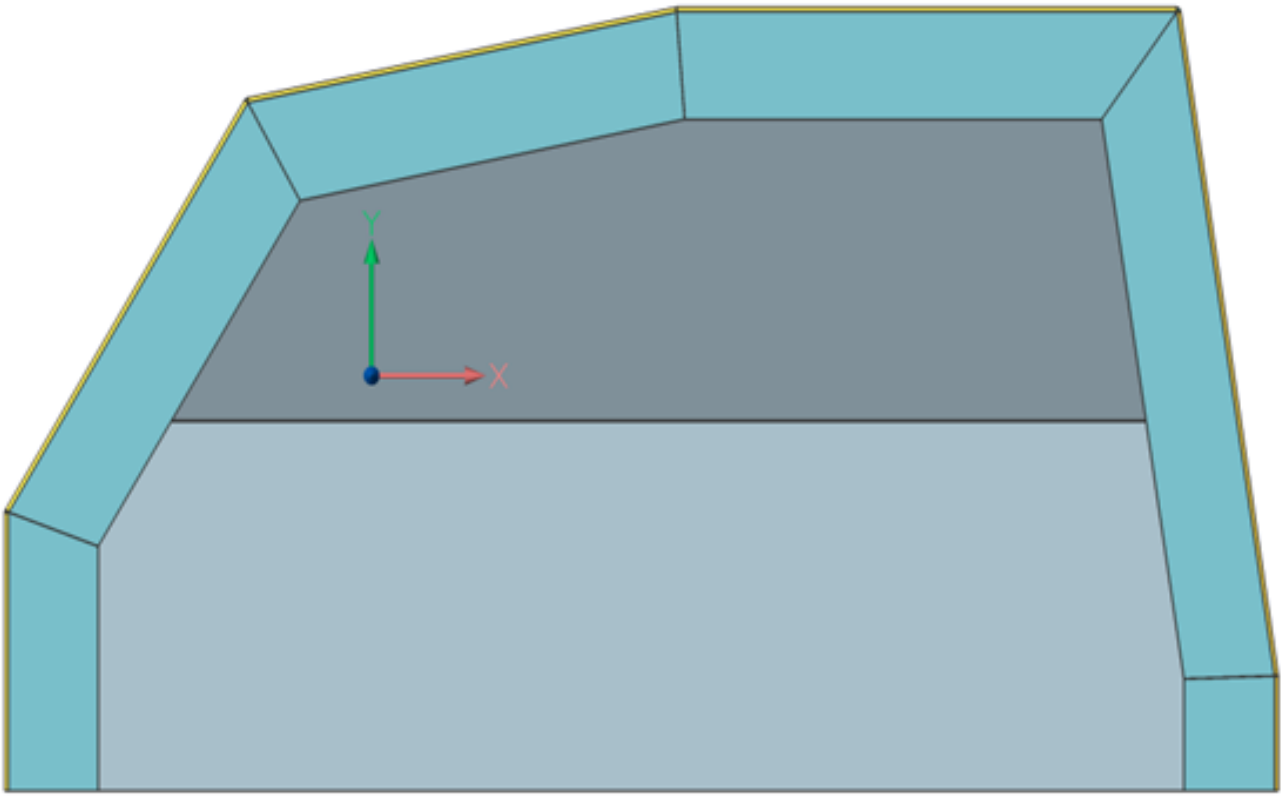


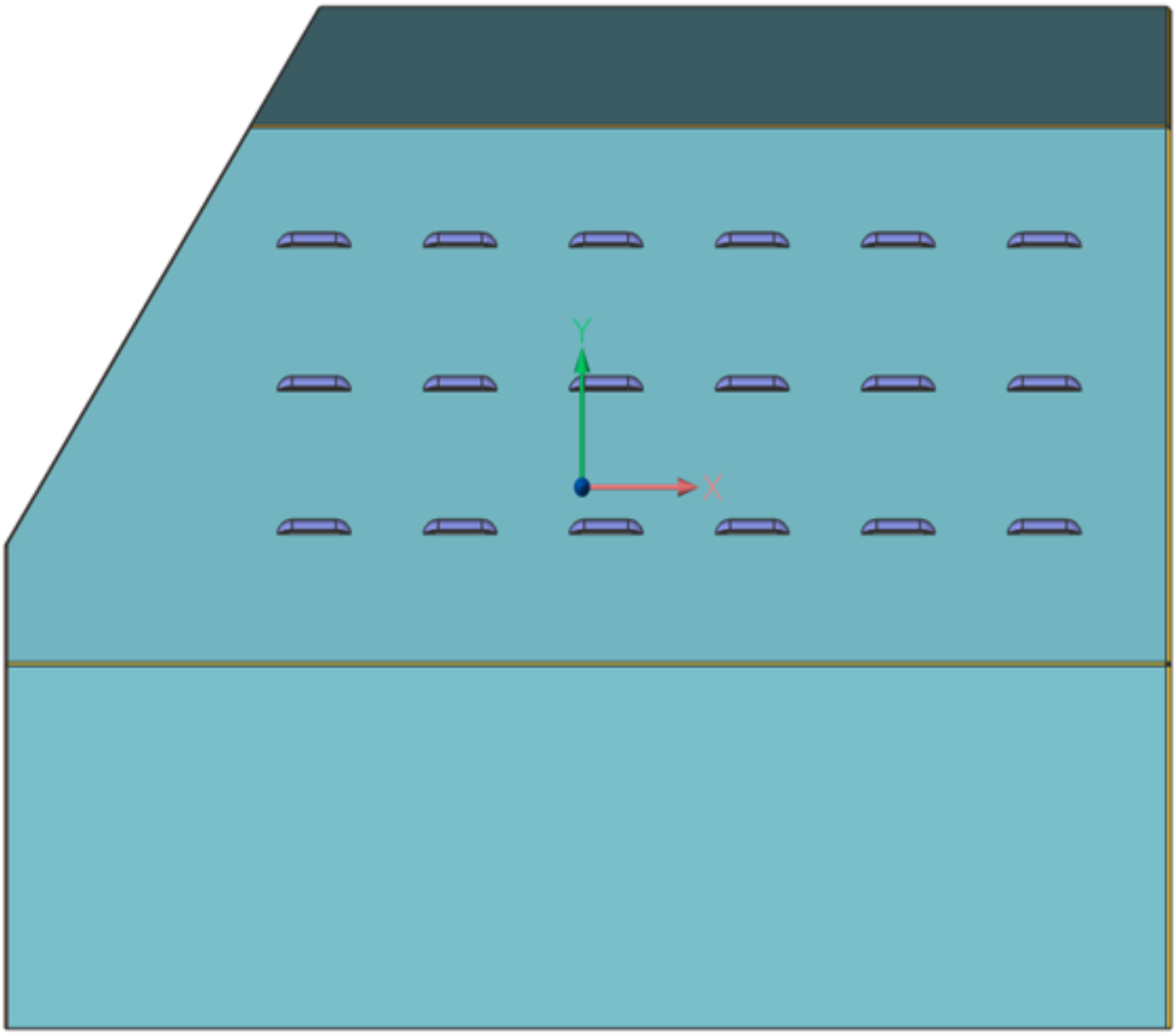
Figure 3.243



3.7.18 3.Insert an array of Form Features

1. Switch to **Front** view on the LookFrom (6.20.39)widget.
2. On the Components panel (6.9.61) select **Louver** from **Sheet Metal | Form Features** folder.
3. On **BmInsert** Prompt menu choose the **Multiple | Array**option.
4. Enter **100,100** from the top right corner as a Basepoint, **120** as the distance between columns; **120** as the distance between rows; placement **18.** Hit **Enter** to complete the command. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041545813/mceclip8.png})

Figure 3.244



3.7.19 3. Creating flanges

1. Switch to the **Top** view on the LookFrom (6.20.39)widget to activate **World UCS**.
2. Hover over the bottom inner edge along the x-axis.
3. Select **Create Flange Edge (6.28.13)** () from the **Sheet Metal** section of the Quad (2.2.3).
4. Enter **90** as the bend angle and **100** as the length of the flange. Hit **Enter** to complete the command. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040715334/mceclip10.png})
5. Repeat steps 3.1-3.4 to create a symmetrical bottom flange: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/ar

Figure 3.245

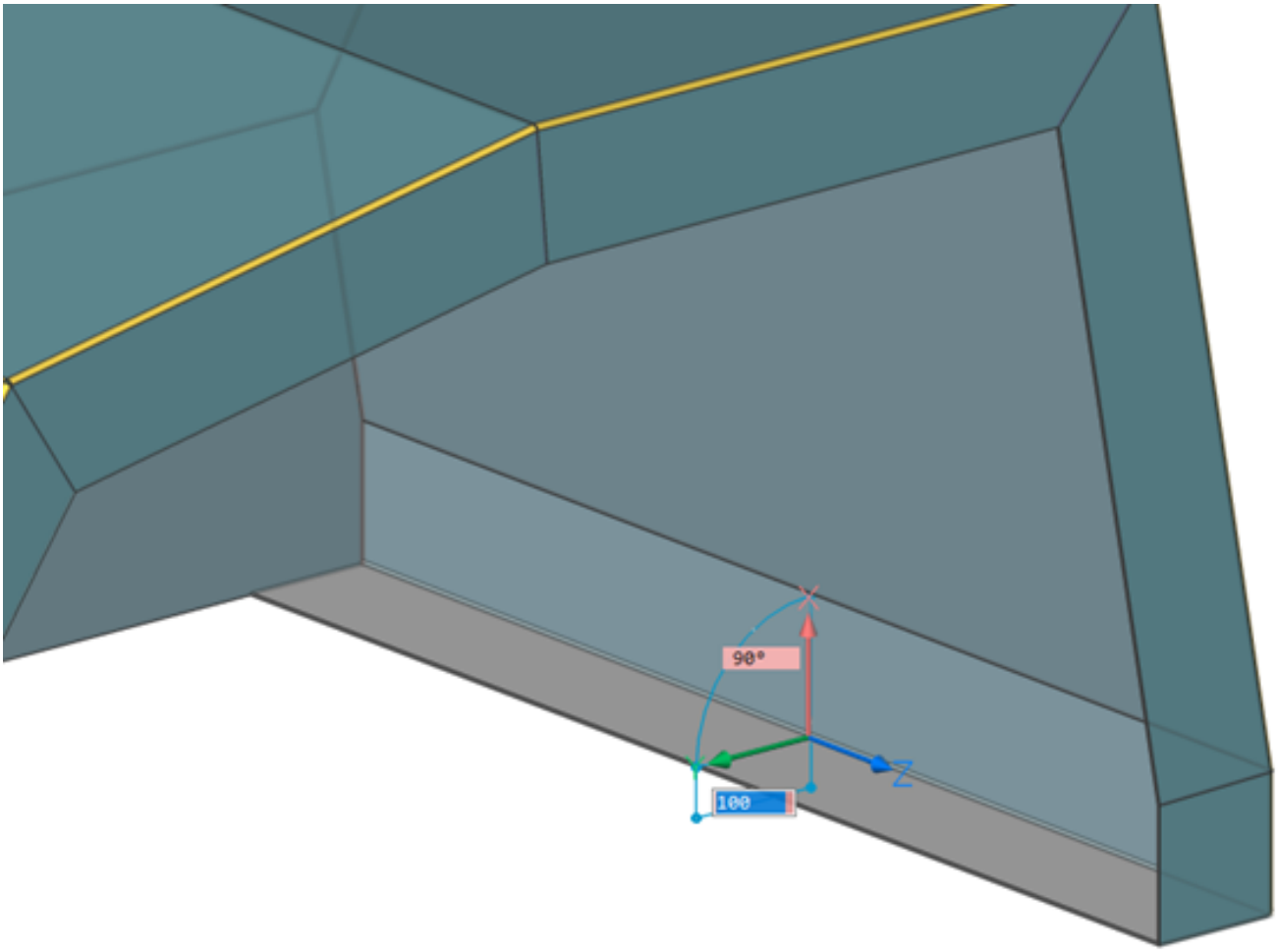
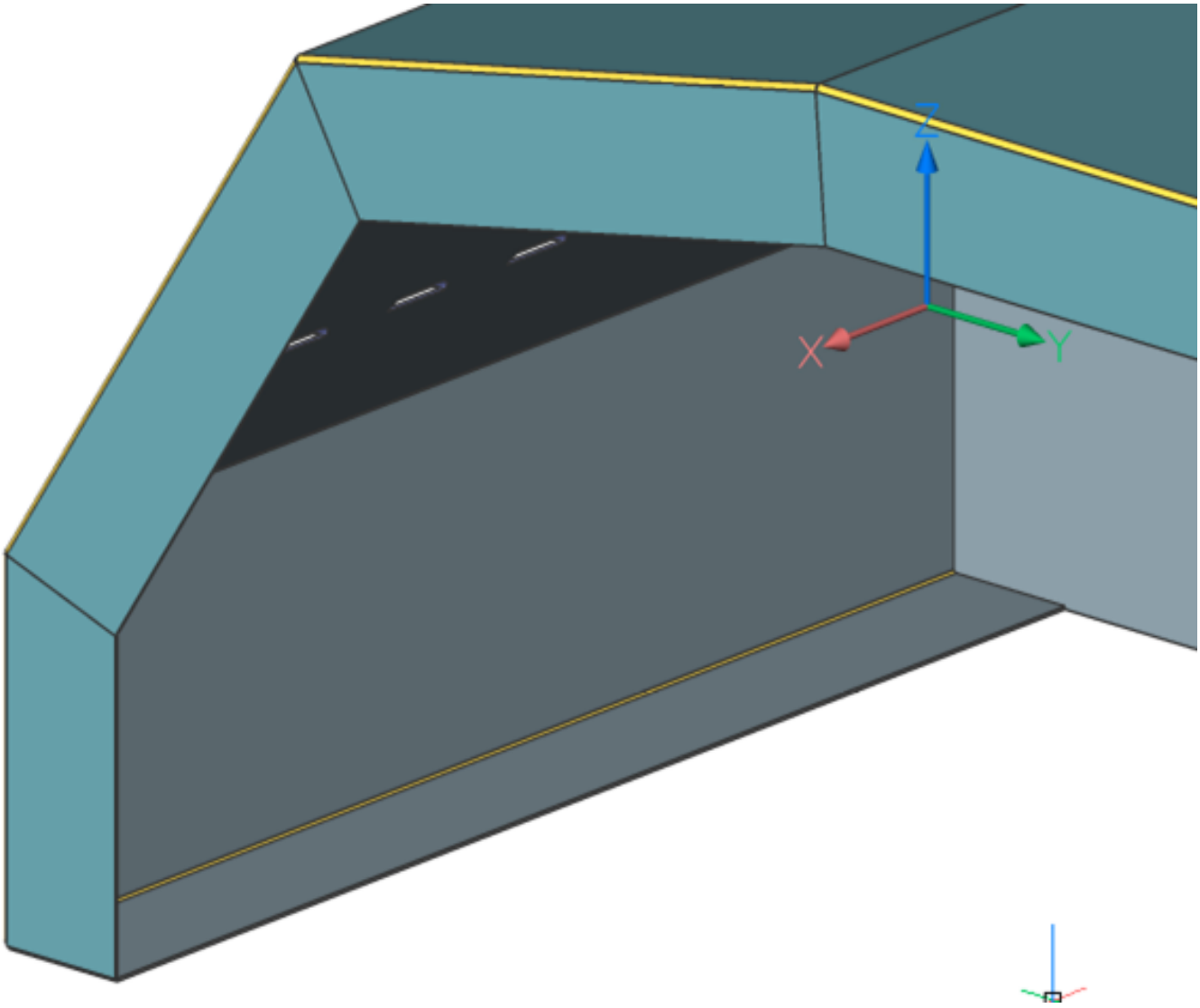


Figure 3.246



3.7.20 4. Finished sheet metal model

1. With nothing selected, right-click to call theQuad (2.2.3). Select **Show Entities (6.30.7)** () from the **General** section of the Quad (2.2.3).
2. Select the Sheet metal entity.
3. In the Properties Panel (1.12.4) click the **Transparency** field. Change the value to **90. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040715374/mceclip12.png})**
4. Hover over the top flange face. Select **Unfold body (6.28.29)** () from the **Sheet Metal** section of theQuad (2.2.3).
5. Choose the position for the unfolded body. Hit **Enter** to complete the command. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041545873/mceclip13.png})

Figure 3.247

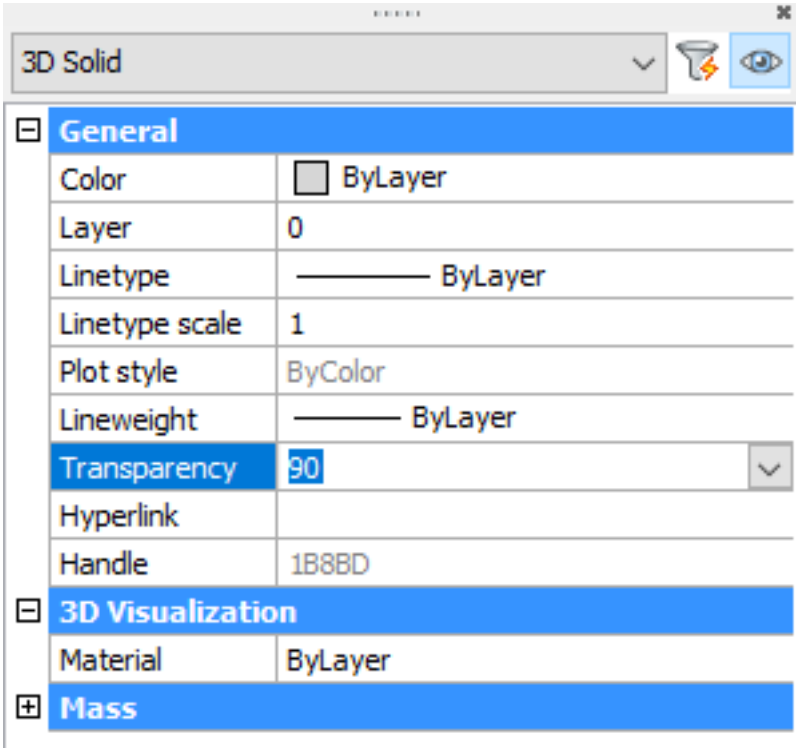


Figure 3.248

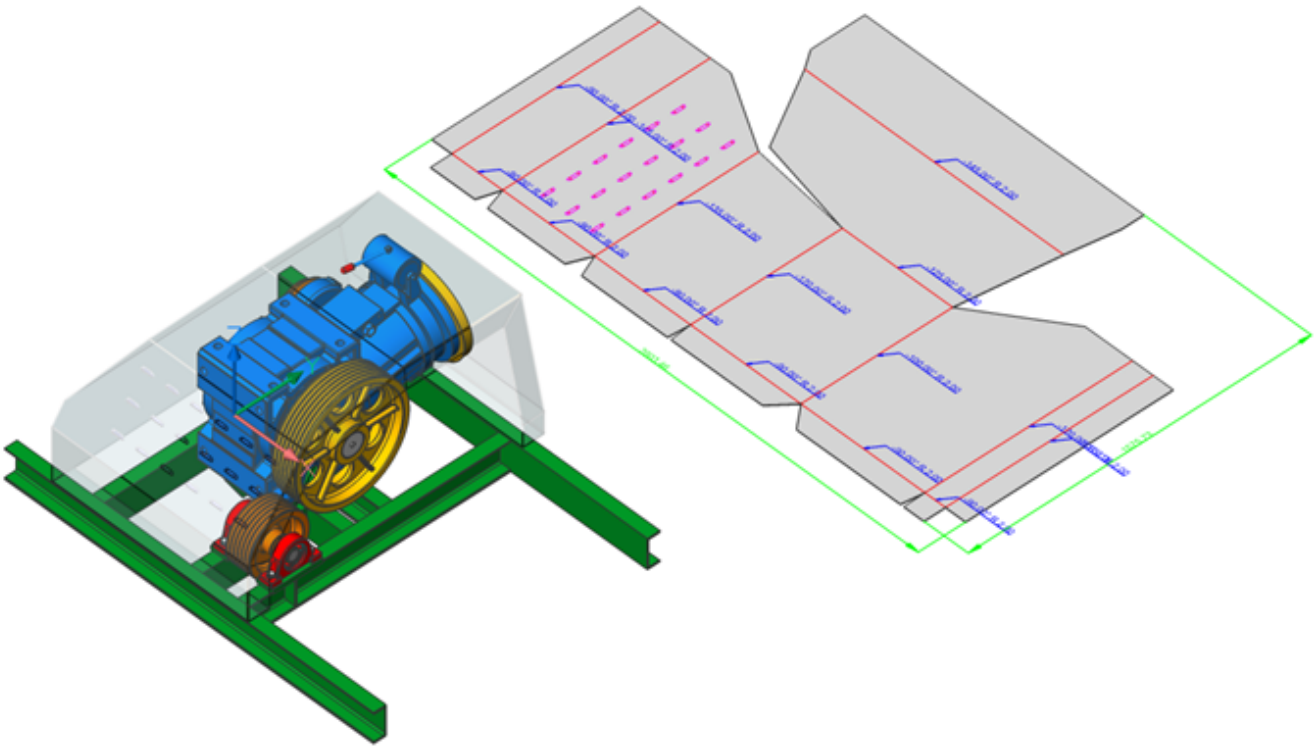
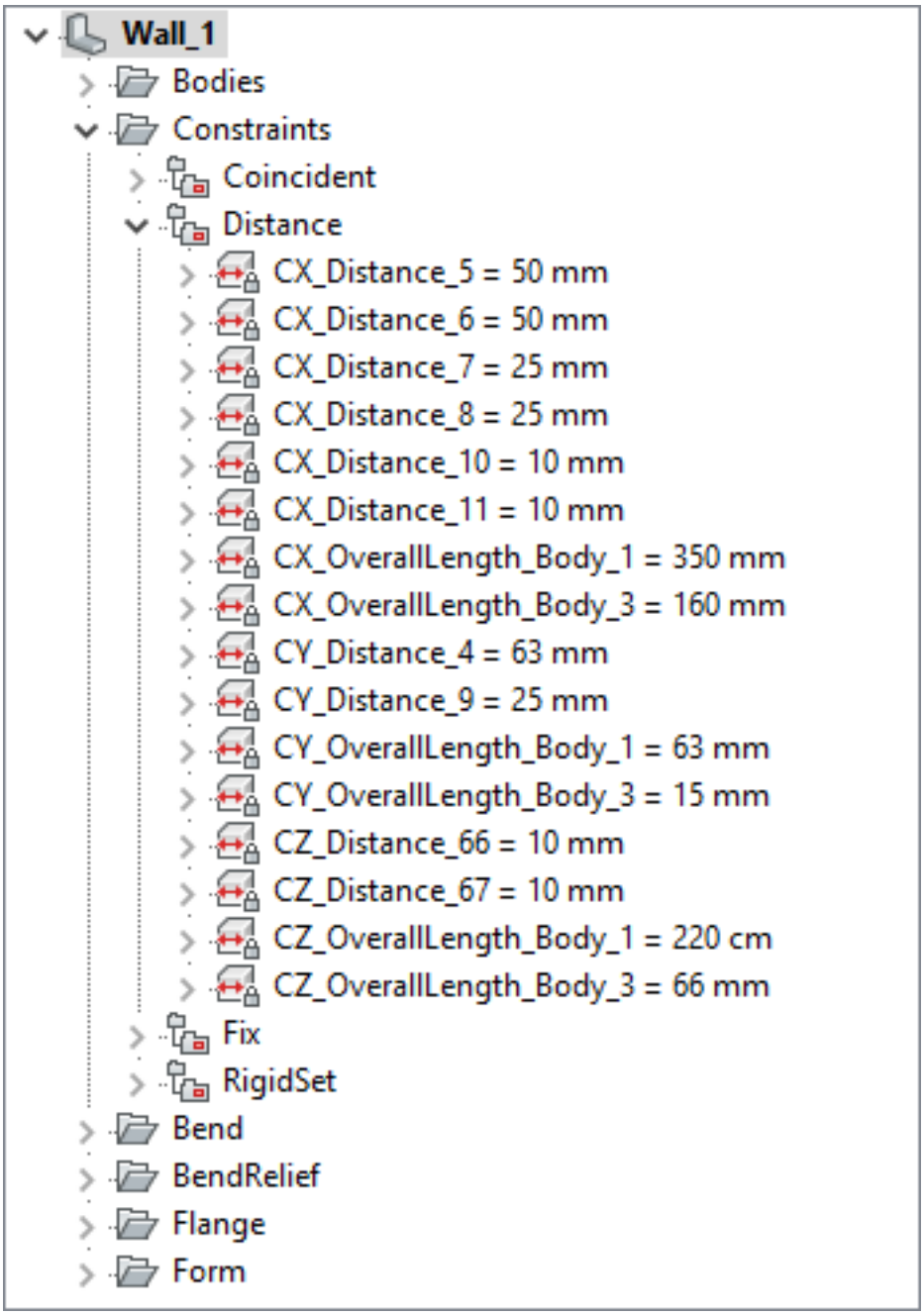


Figure 3.249



3.7.21 Sheet metal parametrize

You need **BricsCAD Platinum** and **Mechanical** licenses for this tutorial.

1. Open the **Cabin_assembly.dwg**file from SmParametrize_files.zip.
2. Select **Wall_1:1** in the Mechanical Browser (3.8.5).
3. Click **Open (6.23.8)** ( ) on the right-click menu.
4. Click the **Parametrize (6.28.21)** ( ) button in the **Sheet Metal| 3D Constraints** panel on the **Ribbon (1.13.6)**.
5. Hit **Enter**. The following 71 constraints are created: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattac

6. Save the file as **Wall_1_parameter.dwg.**
7. Close the **Wall_1_parameter.dwg** file.
8. Switch to the **Cabin_assembly.dwg**file.
9. Select **Wall_1** in the Mechanical Browser (3.8.5).
10. Click **Replace all insert... (6.8.21)** ( ) on the right-click menu.
11. Select the **Wall_1_parameter.dwg**file in the **Select file for component** menu, click **Open**.

12. Select **Wall_R** in the Mechanical Browser (3.8.5).
13. Click**Open (6.23.8)**( ) on the right-click menu.
14. Click the**Parametrize (6.28.21)**( ) tool button in the **Sheet Metal | 3D Constraints**panel of the **Ribbon (1.13.6)**.
15. Select the wall.
16. Click **Enter**. The following 39 constraints are created: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleatt
17. Save the file as **Wall_R_parameter.dwg.**
18. Close the **Wall_R_parameter.dwg**file.
19. Switch to the **Cabin_assembly.dwg**file.
20. Select **Wall_R** on the Mechanical Browser (3.8.5).
21. Click **Replace** ( ) on the right-click menu.
22. Select the **Wall_R_parameter.dwg** file in the **Select file for component** menu, click **Open**.
23. Select the following parameters in the Mechanical Browser (3.8.5):
    - **Wall_1:1|CZ_OverallLength_Body_1**
    - **Wall_1:2|CZ_OverallLength_Body_1**
    - **Wall_1:3|CZ_OverallLength_Body_1**
    - **Wall_1:4|CZ_OverallLength_Body_1**
    - **Wall_1:5|CZ_OverallLength_Body_1**
    - **Wall_1:6|CZ_OverallLength_Body_1**
    - **Wall_1:7|CZ_OverallLength_Body_1**
    - **Wall_R:1|CZ_OverallLength_Body_1**
24. Click **Link to parameter** on the right click menu. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachm
25. Change the expression of the **CZ_OverallLength_Body_1** parameter to **2400**. (\vref {fig:C:/Dev/Task-2448/images/helpbr

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040939314/mceclip4.png})

Figure 3.250

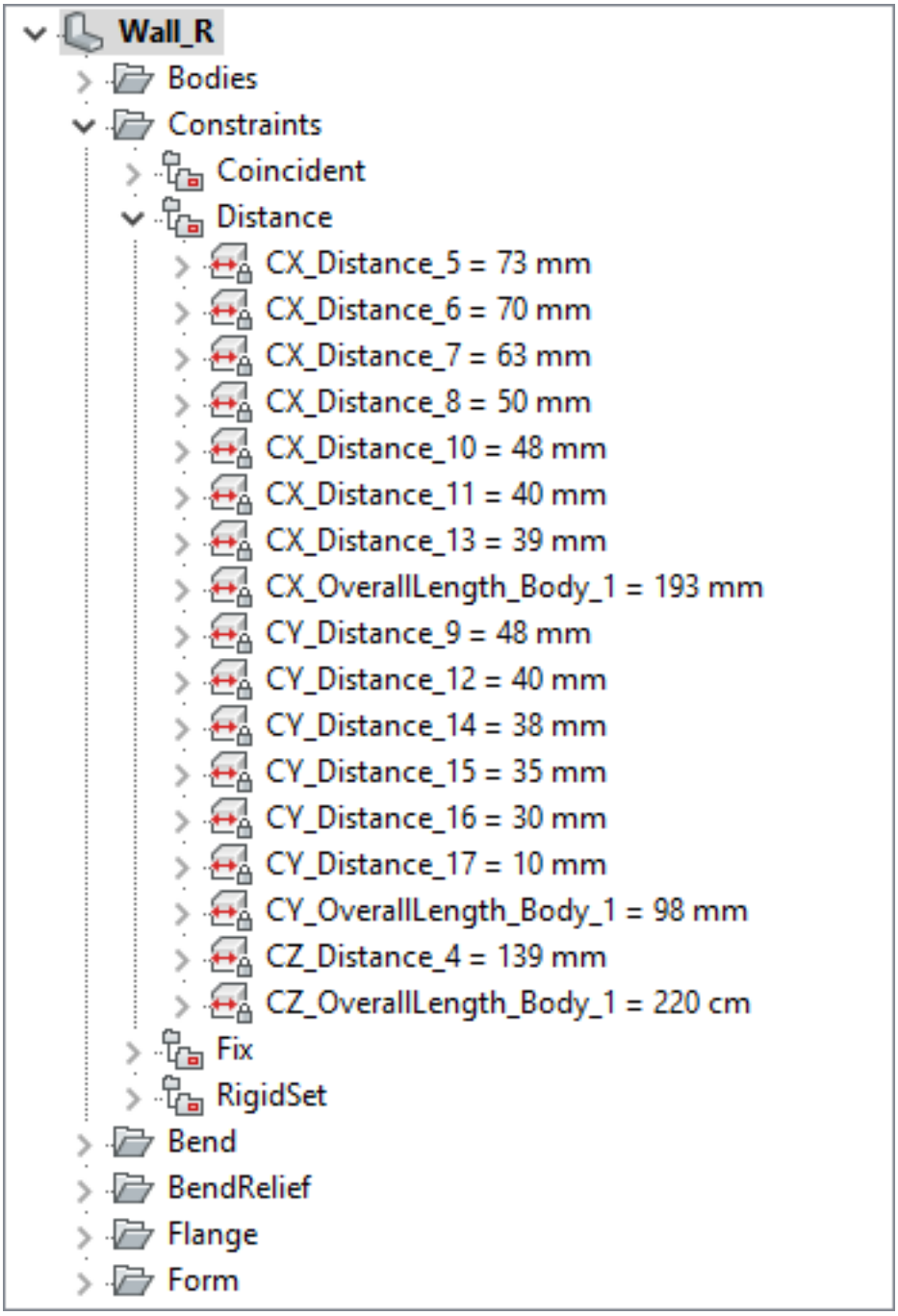


Figure 3.251

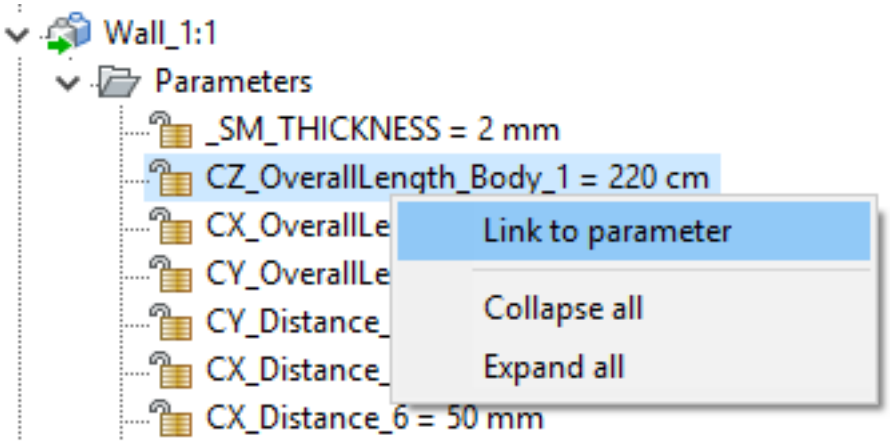


Figure 3.252

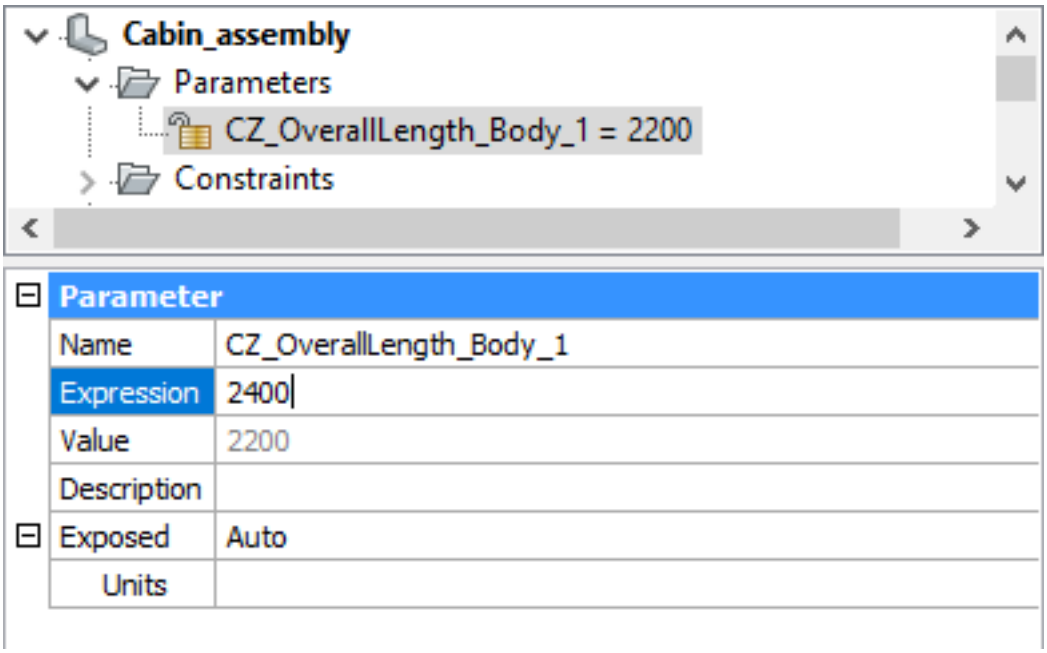
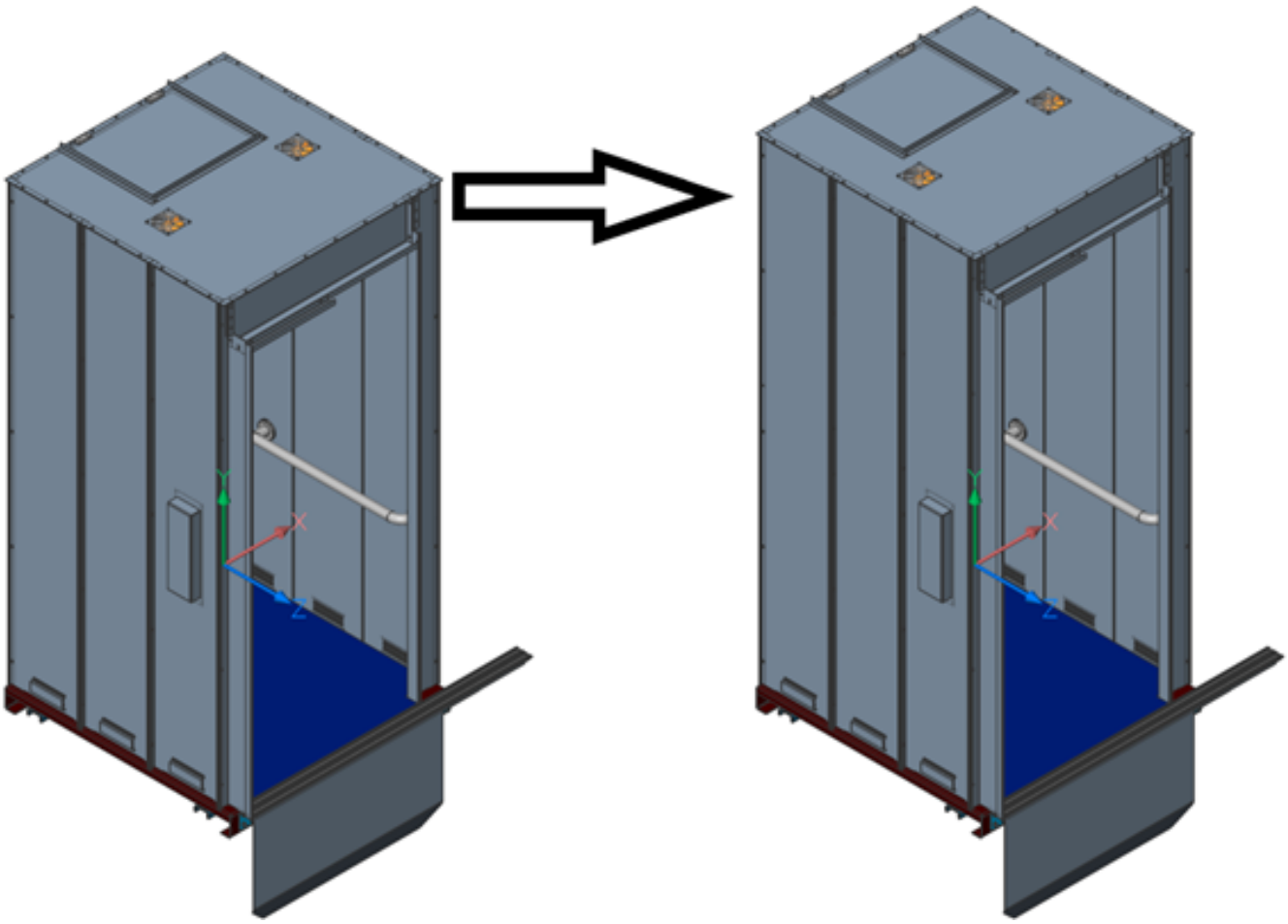


Figure 3.253



3.7.22 Parametrizing sheet metal parts

https://www.youtube-nocookie.com/embed/uK90ff5Ha64 Sheet Metal for BricsCAD adds new design intelligence by automatically placing3D constraints withinyour model as you work. The result? Better dimensional control over your designs with the automated creationof these constraints. Back to Tutorial Overview

3.7.23 Making a parametric workbench - Part 1

https://www.youtube-nocookie.com/embed/Lw98FrIc6D0 In this tutorial series we're going to demonstrate how you can create a parametric workbench in sheet metal. Back to Tutorial Overview

3.7.24 Making a parametric workbench - Part 2

https://www.youtube-nocookie.com/embed/DRcDVhHsxpg Starting with the completed model of the sheet metal bench from Part 1 we now add the parameters and constraints, which will allow us to drive our model. Back to Tutorial Overview

3.7.25 Making a parametric workbench - Part 3

https://www.youtube-nocookie.com/embed/El4bOrCaM7w In the third part of this tutorial series we demonstrate how an Architectural Designer can make use of our bench to complete their design. Back to Tutorial Overview

3.8 Sheet Metal

3.8.1 Sheet Metal Commands and Toolbars

Sheet Metal Toolbar Icon	Flyout Toolbar Icon	Command	Description
		SmFlangeBase (6.28.10)	Creates a base (initial) flange of a sheet metal part from a closed planar profile.
		SmExtrude (6.28.9)	Creates a sheet metal part by extruding a polyline in the direction of the normal of the curve plane. Thickness is assigned with respect to the external dimensions of the polyline.
		SmLoft (6.28.20)	Creates sheet metal part with lofted bends and flanges from two non-coplanar curves.
		SmConvert (6.28.4)	Automatically recognizes flanges, bends and lofted bends in a 3D solid.
		SmFlangeEdge (6.28.13)	Creates one or more flanges on a sheet metal part by pulling one or more edges of an existing flange.
		SmFlip (6.28.15)	Changes the reference side for selected flanges with an optional shift of the flange geometry by thickness.
		BmInsert (6.8.12)	Inserts form features.
		SmRibCreate (6.28.26)	Creates an associative rib form feature on a flange face from a planar 2D curve.
		SmForm (6.28.16)	Creates form feature from given set of faces.
		SmForm (6.28.16)	Explodes form feature in flanges and bends if possible.
		SmReplace (6.28.25)	Replaces form features (including recognized ones) in sheet metal parts with form features from a built-in or user library.
		SmBendCreate (6.28.2)	Converts hard edges (sharp edges between flange faces) and junctions into bends.
		SmBendSwitch (6.28.3)	Converts bends to lofted bends
		SmFlangeBend (6.28.11)	Bends an existing flange along a line, obeying k-factor for given bend radius.

		SmJunctionCreate (6.28.18)	Converts hard edges into junctions.
		SmJunctionSwitch (6.28.19)	Changes a symmetrical junction feature to one with overlapping faces.
		SmFlangeConnect (6.28.12)	Closes gaps between two arbitrarily oriented flanges.
		SmReliefCreate (6.28.22)	Makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.
		SmReliefSwitch (6.28.23)	Converts a corner relief to a circular relief.
		SmReliefSwitch (6.28.23)	Converts corner reliefs to a rectangular relief.
		SmReliefSwitch (6.28.23)	Converts corner reliefs to a VType relief.
		SmReliefSwitch (6.28.23)	Converts corner reliefs to a Smooth relief.
		SmReliefSwitch (6.28.23)	Converts corner reliefs to a Round relief.
		SmReliefSwitch (6.28.23)	Converts corner reliefs to a Rip relief.
		SmSplit (6.28.28)	Splits a flange along a line drawn on its face.
		SmSplit (6.28.28)	Splits a flange by a single vertex in the corner.
		SmImprint (6.28.17)	Splits thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends.
		SmDissolve (6.28.6)	Removes sheet metal data from the selected faces or entities.
		SmDelete (6.28.5)	Removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.
		SmFlangeRotate (6.28.14)	Rotates a flange.
		SmSelect (6.28.27) Same Form Features	Selects all identical form features in the sheet metal model
		SmSelect (6.28.27) Hard Edges	Selects all hard edges on sheet metal parts.

		SmRepair (6.28.24)	Fixes problems specific to Sheet Metal parts, both global and local workflows are supported. Optionally heals lofted bend features: merges adjacent bends and provides tangential connection with flanges.
		SmParametrize (6.28.21)	Creates a consist set of 3D constraints for the selected sheet metal part.
		SmUnfold (6.28.29)	Unfolds the sheet metal body.
		SmExportOSM (6.28.8)	Exports a sheet metal solid to the .osm (Open Sheet Metal) file format.
		SmExport2d (6.28.7)	Exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.
		SmAssemblyExport (6.28.1)	Performs a batch export of assembly components to unfolding, creating a .dxf per each 3D solid recognized as sheet metal part.
		FEATURECOLORS (1.13.11)	Toggles the **FEATURECOLORS** system variable.

The Sheet Metal tools are available:

- In the Quad (2.2.3) cursor menu (depending on the feature under the cursor): (\vref {fig:C:/Dev/Task-2448/images/staticb
- On the **Sheet Metal** toolbars:
  - **Sheet Metal** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/SheetMetal-1Toolbar.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/SheetMeta2Toolbar.png})
  - **SM Bend** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/SMBendToolbar
  - **SM Flange** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/SMFlangeTool
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  - **SM Repair (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/SMRepairToo**
  - **SM Split** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/SMSplitToolbar.p

Figure 3.254

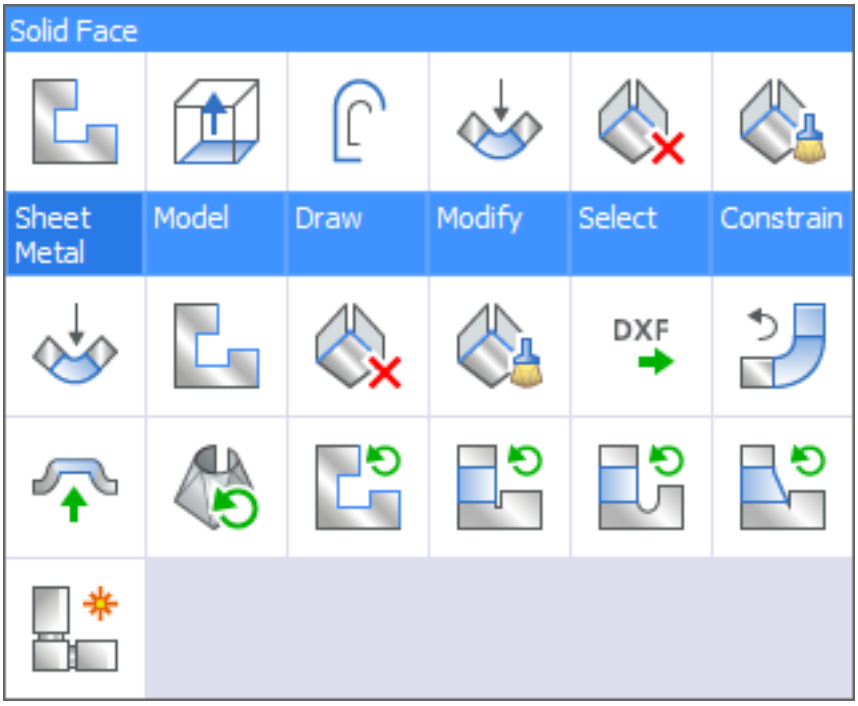


Figure 3.255



Figure 3.256

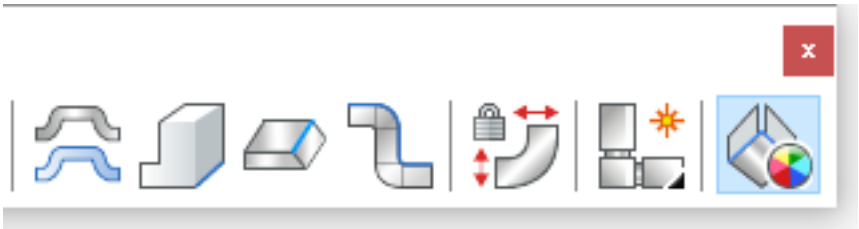


Figure 3.257

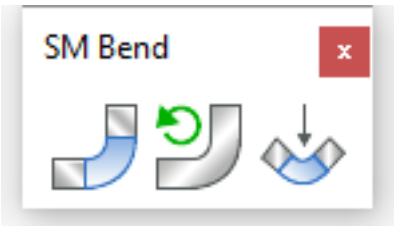


Figure 3.258

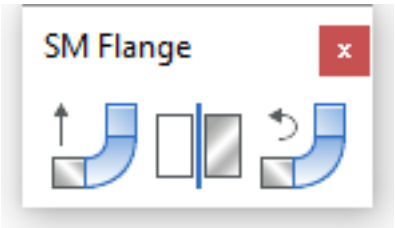


Figure 3.259

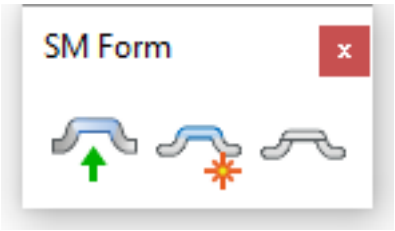


Figure 3.260

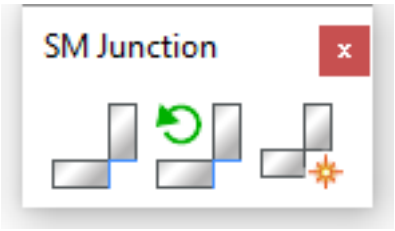
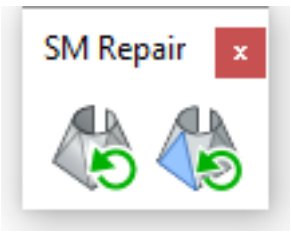


Figure 3.261



Figure 3.262



  - **SM Unfold** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/SMUnfoldTool

- In the **Sheet Metal** menu (**Mechanical** Workspace only): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU

- In the **Sheet Metal** tab of the **Sheet Metal** workspace Ribbon. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalCommands

See alsoDirect Modeling (1.18.2)tools.

Figure 3.263

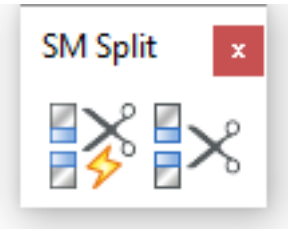


Figure 3.264

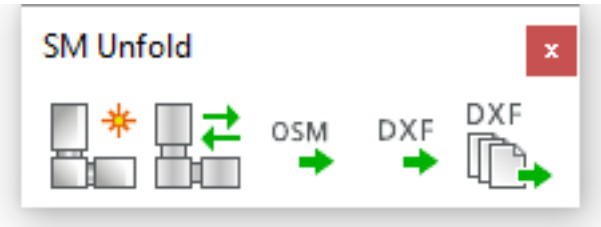


Figure 3.265

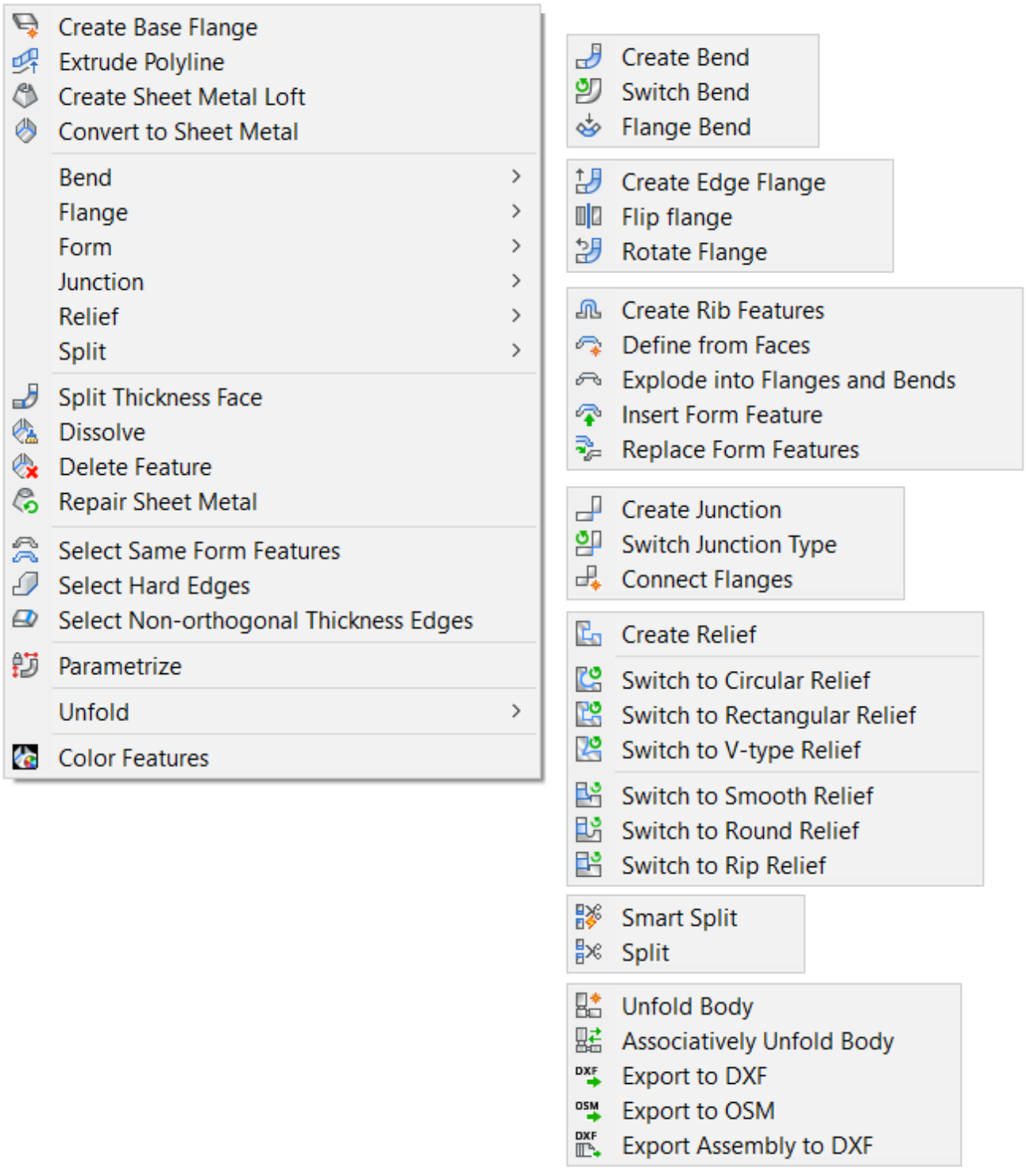


Figure 3.266

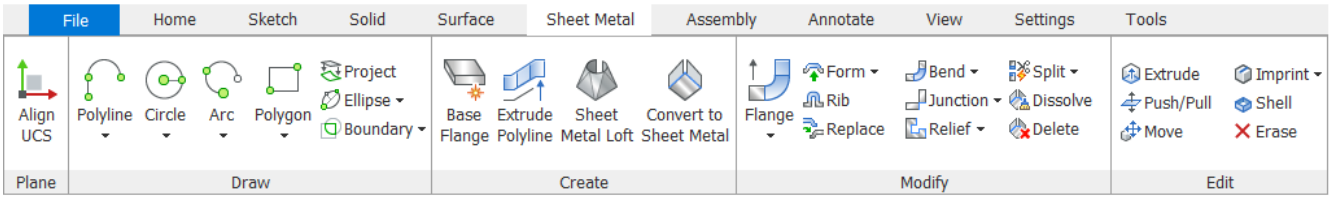
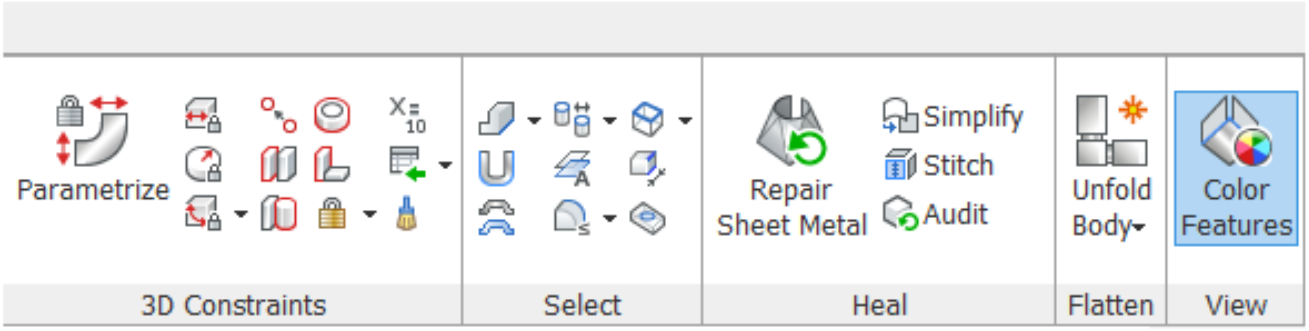


Figure 3.267



3.8.2 Sheet Metal Design Methods

Sheet metal design allows you to model sheet metal parts and generate their unfolded representations with manufacturing information. You can create complex sheet metal parts with BricsCAD easily and rapidly because the design process is different from a manufacturing process. Do not think in terms of a planar sheet that should be cut and bent, but model your part directly as you create solid bodies with Direct Modeling tools.

Tutorial: Introducing BricsCAD Sheet Metal Click here (3.7.10) to watch

Tutorial: Designing from scratch in BricsCAD Sheet Metal Click here (3.7.8) to watch You can create sheet metal parts in BricsCAD in four different ways (or in any combination of them):

1. Create a base flange from a closed planar profile, then add flanges:
   - Pull its edges to create additional edge flanges with bends.
   - Create partial flanges with automatic creation of the corresponding bend reliefs.
   - Pull several adjacent edges to create several flanges at once with automatic creation of corner reliefs and junctions between them.
   - Bend a flange to create a new flange with a bend from its portion.
2. Create a combination of flanges and lofted bends connecting two 3D profiles Using Sheet Metal Loft (6.28.20) tool.
3. Create a combination of flanges and bends from a planar profile Using Sheet Metal Extrude (6.28.9)tool.
4. Create a shelled body from a 3D solid Using **Shell**option of the SolidEdit (6.27.39) command with thedesired thickness, then convert it to a sheet metal part withSmConvert (6.28.4) command, and create bends and junctions from its hard edges.
   - Make holes by drawing closed profiles on faces of the flanges and pushing them through the flanges.
   - Adjust material thickness and bend radius using pre-defined parameters.
   - Apply Direct Modeling operations and 3D constraints (1.19.4) to further adjust your design while always maintaining the design intent in terms of sheet metal features.
   - Automatically generate unfolded representation of your part and send it to a CAM system by exporting the corresponding 2D drawing with bend annotations in a .dwg or .dxf file.

You can further detail your sheet metal part using the following operations:

- Make holes by drawing closed profiles on faces of the flanges and pushing them through the flanges.
- InsertForm Features (3.8.18)from a library. A library of parametric form features is installed with BricsCAD Sheet Metal and you can create your own form features.
- Create rib form features from a 2D curve lying on the flange by the SmRibCreate (6.28.26) command.
- Adjust material thickness and bend radius using pre-defined parameters.
- Apply Direct Modeling operations and 3D constraints (1.19.4) to further adjust your design while always maintaining the design intent in terms of sheet metal features.

When your sheet metal part is ready, you can automatically unfold it and send the result to a manufacturing engineer by exporting the corresponding 2D drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/s
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/

Tutorial: Batch Processing in BricsCAD Sheet Metal Click here (3.7.5) to watch

Figure 3.268

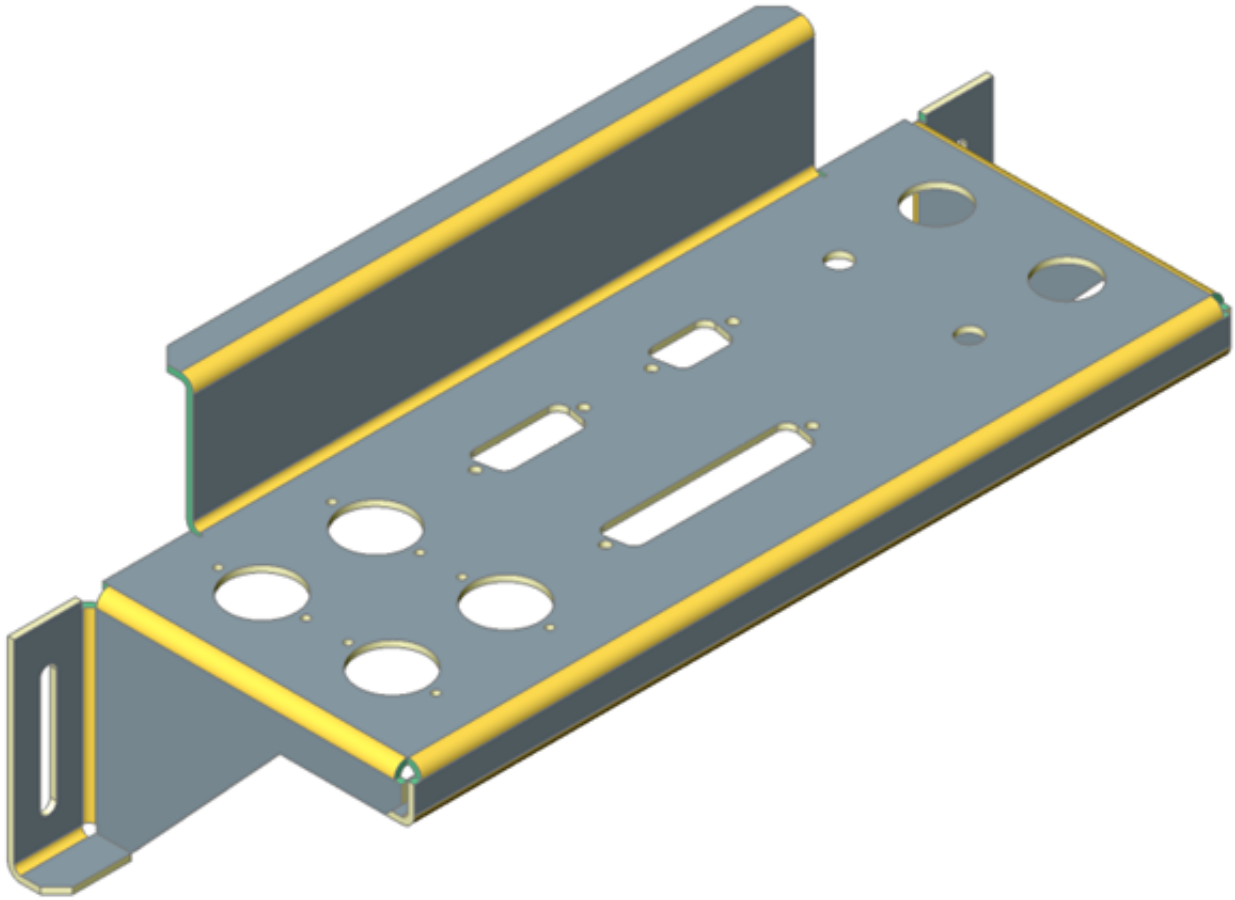


Figure 3.269

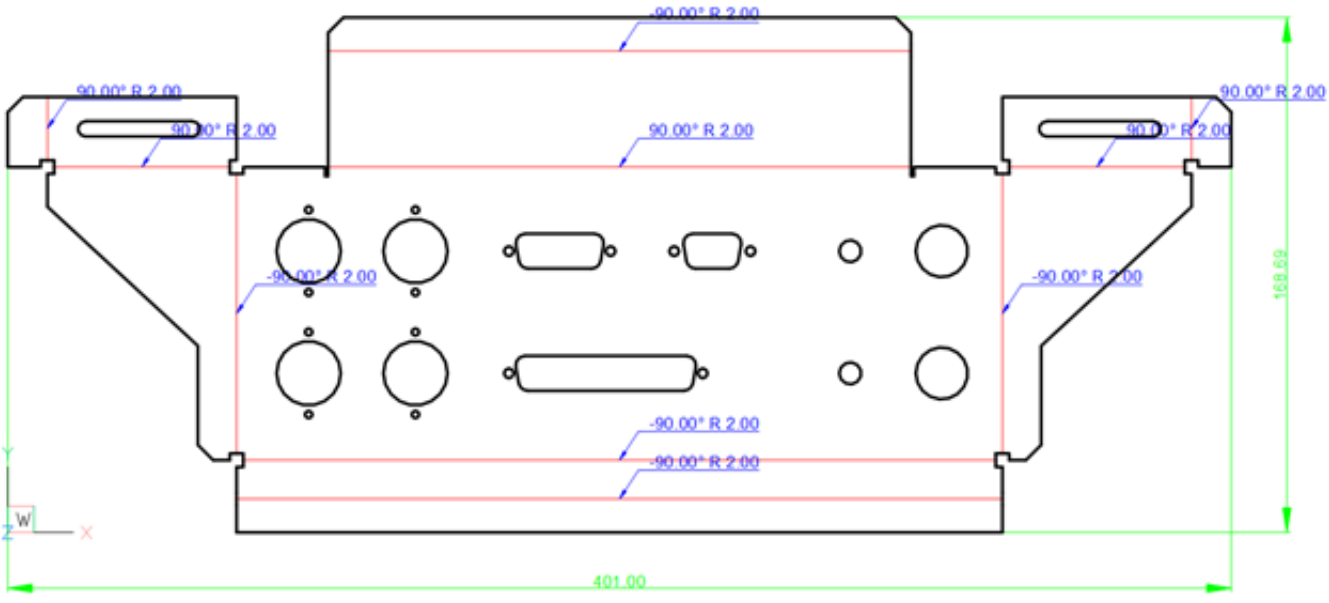
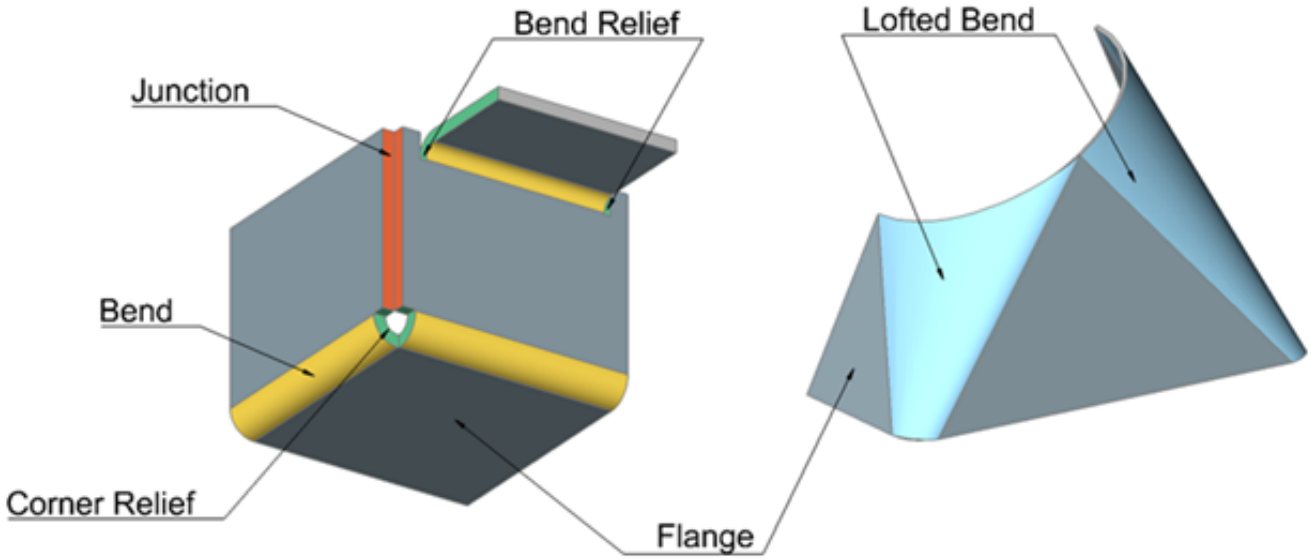


Figure 3.270



3.8.3 Sheet Metal Features

In BricsCAD features are smart regions (groups of faces) of your 3D part. Each feature maintains specific spatial and parametric relationship between its faces and some adjacent faces. The features allow you to embed design intent into your model. The features are created automatically depending on a particular geometric operation you apply. Images below show the sheet metal features in BricsCAD: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enI
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enINTL/V18/UsrGui/source/10DrawingEntities/03DrawingSolids

Flange The main feature of any sheet metal part is a flange, which consists of two parallel planar faces located in front of each other (with some solid volume between them) on the distance, which is significantly smaller than the linear size of the faces.This distance is equal to the material thickness. When you modify the model, this distance relation is always maintained automatically. Other faces, which are adjacent to flange faces and do not belong to bends are called thickness faces. They are always perpendicular to flange faces. Flange features are created bySmFlangeBase (6.28.10),SmExtrude (6.28.9),SmLoft (6.28.20),SmFlangeEdge (6.28.13),SmFlangeBend (6.28.11)andSmConvert (6.28.4)commands. If the FEATURECOLORS (1.13.11)system variable is ON, the display color distinguishes between the reference face of a flange and its opposite side. Parametric modifications of a sheet metal part, such as thickness change, do not move reference faces, if possible. TheSmFlip (6.28.15)command swaps sides of a selected flange so that reference faces are on the other geometric side of the flange; optionally the command shifts the flange over the thickness of the sheet metal part. The images below illustrate the color legend for the flange faces of the same part.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enINTL/V18/UsrGui/source/10DrawingEntities/03Drawing	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enINTL/V18/UsrGui/source/
Reference Faces	Other Faces

Bend Two flanges are connected by a bend. A bend consists of two coaxial cylindrical faces, which are always tangent to the adjacent planar faces of the flanges. Bend features are created bySmFlangeBase (6.28.10),SmExtrude (6.28.9),SmFlangeEdge (6.28.13),SmFlangeBend (6.28.11)andSmConvert (6.28.4)commands.

Wrong Bend Wrong bend features are recognized by the SmConvert (6.28.4)command and reflects assumptions on the user's design intent. Like regular bends, wrong bends connect flanges, but their faces can be non-coaxial or they can have non-tangent connections with adjacent flange faces, which prevent using the geometry as regular bend features. Before unfolding the part all wrong bends must either be repaired bySmRepair (6.28.24)or removed by SmDelete (6.28.5) and then recreated bySmBendCreate (6.28.2).

Lofted Bend A lofted bend feature consists of two ruled surfaces located in front of each other (on the offset equal to the material thickness) with some solid volume between them. A ruled surface can be described as the set of points swept by a moving straight line. Examples are cylinders, cones, helicoids. Lofted bends can be created by thickening of ruled surfaces. A lofted bend can connect one or two flanges or be stand-alone, without flanges at all. Some combinations of adjcency between regular bends and lofted bend are also allowed. Differences between

Figure 3.271

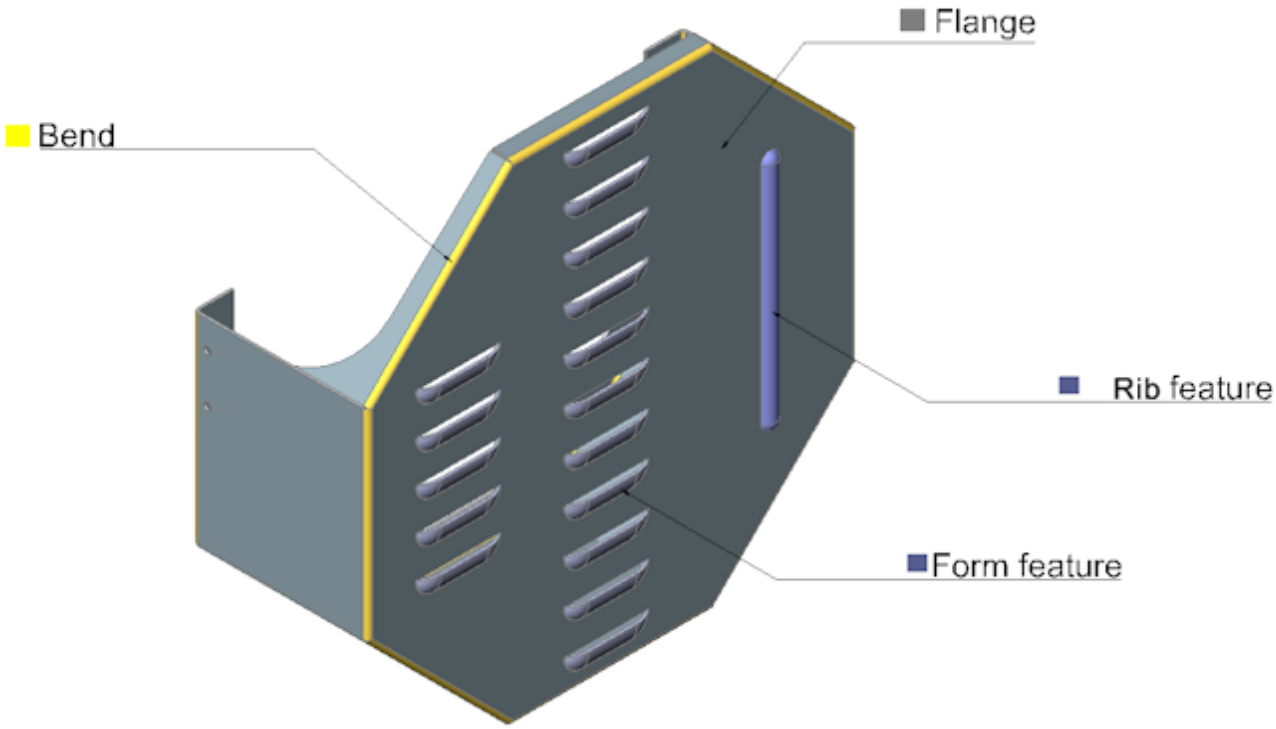


Figure 3.272

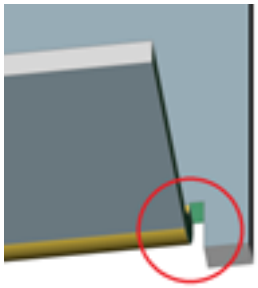


Figure 3.273

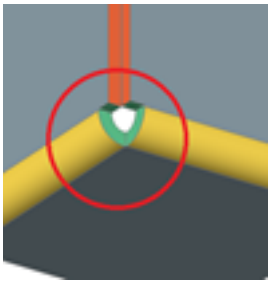
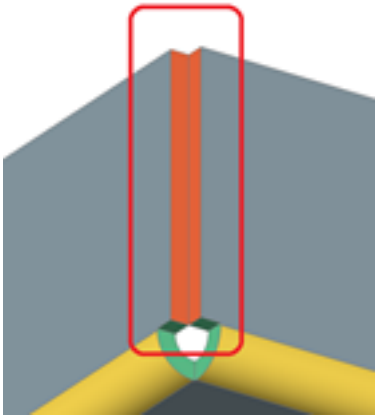


Figure 3.274



Bend Relief Group of faces representing a technological cut between two flanges of different width connected with a bend. The bend relief feature maintains the distance between two opposite faces of the cut. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enINTL/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART
Bend relief features are created by SmFlangeEdge (6.28.13), SmBendCreate (6.28.2)andSmReliefCreate (6.28.22)commands.

Corner Relief Group of faces representing a cut in the corner where three or more flanges meet together. This cut is needed to prevent unwanted material deformation during bending of the sheet metal part. Corner relief feature maintains the form and size of this cut. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enINTL/V18/UsrGui/
Corner relief features are created by SmFlangeEdge (6.28.13), SmBendCreate (6.28.2)andSmReliefCreate (6.28.22)commands.

Junction Junction feature consists of two thickness faces of adjacent flanges, which are not connected via bend. A junction can be symmetric or overlapping. Junction feature maintain the minimal distance between the flanges (called **junction gap**). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enINTL/V18/UsrGui/source/10Dra
Junction features are created bySmFlangeEdge,SmJunctionCreate (6.28.18)andSmFlangeConnect (6.28.12)commands.

Miter Feature Miter feature represent a split between coplanar flanges which is created bySmFlangeEdgeand **SmSplit** commands. Miter feature maintains the miter gap. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enIN

Form and Rib Features Form and rib features represent a rich variety of shapes which are produced by punching and wheeling. Two representations of the form features are important: one in 3D and a 2D symbol in the unfolded geometry. In case of imported geometry the symbol is obtained by the projection of the 3D faces of the form feature In case the form feature was inserted/replaced from Form Feature Library, the unfolding symbol is taken from the library file. Rib features represent imprint on flange made by a wheel following an arbitrary trajectory. Form and rib features are created byBmInsert (6.8.12),SmRibCreate (6.28.26)and SmConvert (6.28.4)commands.

Figure 3.275

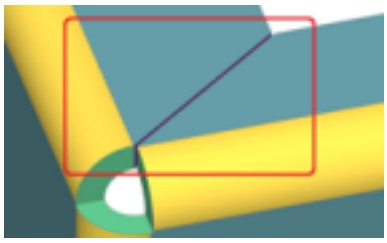


Figure 3.276

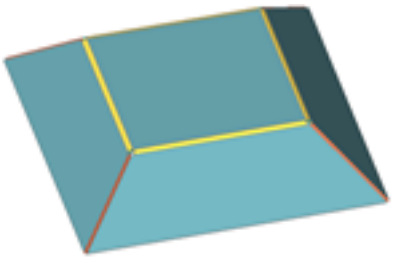


Figure 3.277

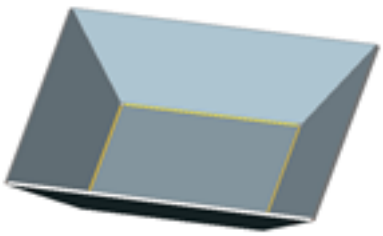
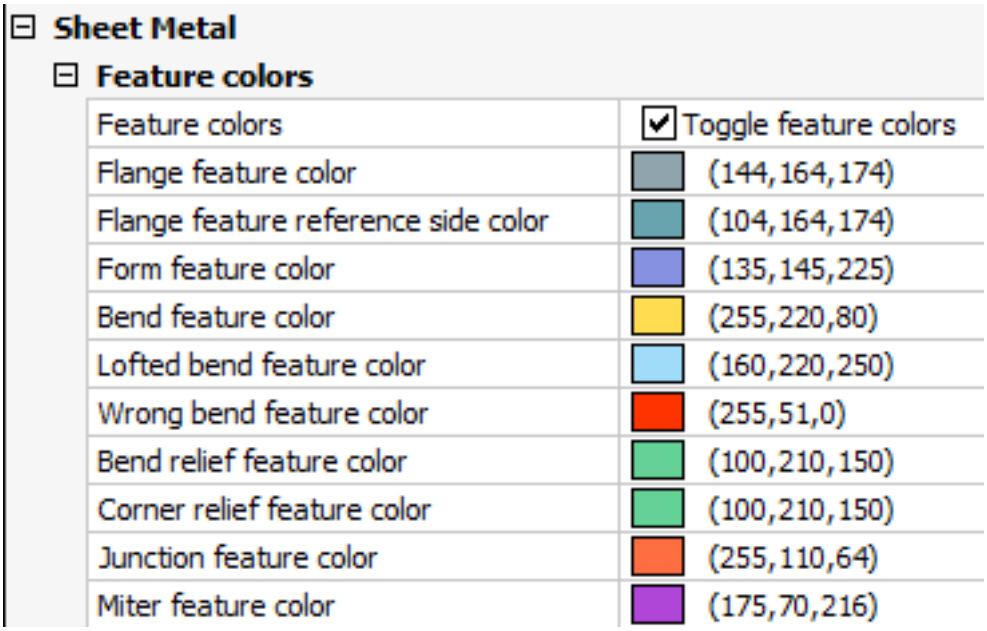


Figure 3.278



3.8.4 Feature Coloring

For better experience and easy distinguishing between the features, different colors are applied to the various types of features in the geometry. These colors are for visualization only and will never show in generated documents, paper space views, block inserts, etc. The display of feature colors is controlled through the FEATURECOLORS (1.13.11) system variable and a series of system variables, one for each feature color. Although the default feature colors are selected carefully to combine with each other, you can set them differently in the **Sheet Metal** settings group on the Settings dialog (1.13.11). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source
To toggle the display of feature colors:

- Click the **Color Features** tool button ( ) on the **Sheet Metal** toolbar.
- Click the **Color Features** tool button ( ) on the **Sheet Metal**ribbon bar.
- Choose **Color Features** in the **Sheet Metal**menu.

Tutorial: Feature coloring in BricsCAD Sheet Metal Click here (3.7.1) to watch

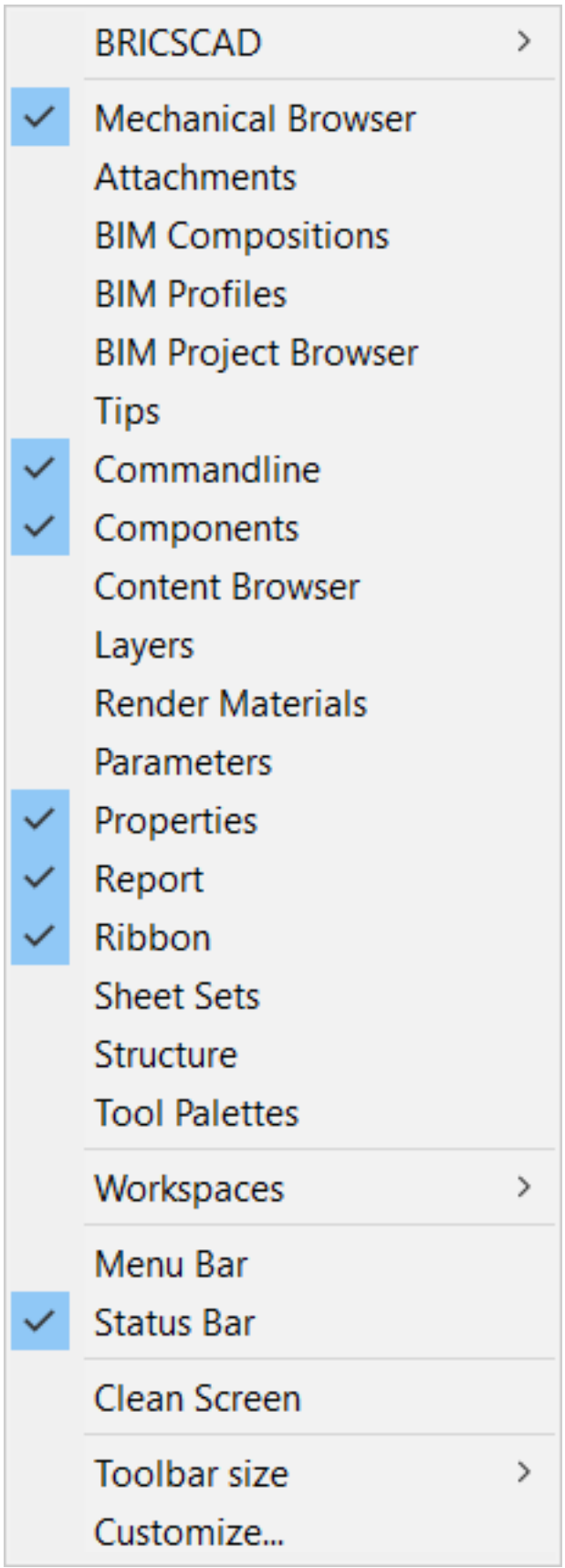
3.8.5 Mechanical Browser for Sheet Metal

The **Mechanical Browser (3.2.2)** opens automatically when switching to the **Mechanical** workspace. To open the **Mechanical Browser** manually do the following:

1. Move the cursor over a toolbar, then right click. A context menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbrics

2. Choose Mechanical Browser in the context menu.

Figure 3.279


BRICSCAD
Mechanical Browser
Attachments
BIM Compositions
BIM Profiles
BIM Project Browser
Tips
Commandline
Components
Content Browser
Layers
Render Materials
Parameters
Properties
Report
Ribbon
Sheet Sets
Structure
Tool Palettes
Workspaces
Menu Bar
Status Bar
Clean Screen
Toolbar size
Customize...

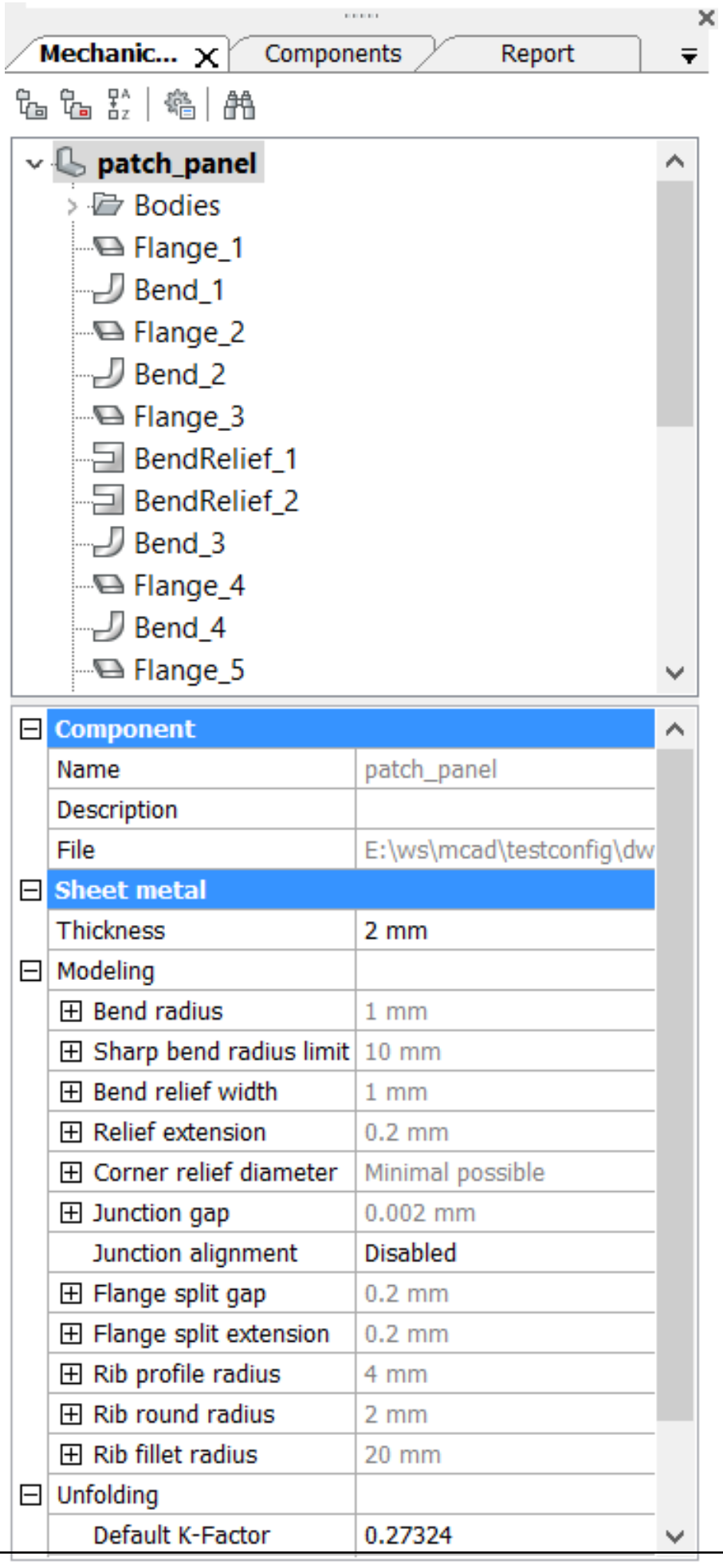
Sheet Metal Context

If you work with a sheet metal part (if you created any sheet metal feature (3.8.3)), properties of the root node in **Mechanical Browser**contain ”Sheet Metal” section (also called **Sheet Metal Context associated with the document**). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/MechanicalBrowserMechan Sheet Metal properties (3.8.6) are represented as grouped fields. The **Thickness** field controls the material thickness of sheet metal solids which has to be equal for all solids of the document. The **Modeling** group plays two roles:

- It provides default settings for Sheet Metal Commands, (3.8.1)such as SmFlangeBase (6.28.10), SmExtrude (6.28.9), SmFlangeEdge (6.28.13), SmFlangeBend (6.28.11), SmLoft (6.28.20),and SmConvert (6.28.4) in a newly created model. The initial parameters are saved in the drawing, which allows creating a set of .dwt template drawings for each type of sheet metal models created in a company.
- New features created by the commands usually derive their properties (3.8.6) from Sheet Metal Context. For example, the default value of**Radius Type** property for the bend feature is ”Use global value”. Thus when you change **Bend radius** in Sheet Metal Context, all the bend features with the derived property will change its radius. So the second application of Sheet Metal Context properties is that they allow globally change properties of sheet metal part.

The **Unfolding** group of properties in Sheet Metal Context lists some properties of SmUnfold (6.28.29),SmExport2d (6.28.7) andSmAssemblyExport (6.28.1) commands. Sheet Metal Context is created when first sheet metal part appears in the document and is initialized from settings listed inInitial Parameters (3.8.9). Typical workflows for creating sheet metal part are listed inSheet Metal Design Methods (3.8.2). Sheet Metal Context is removed when all the features in the document are dissolved by SmDissolve (6.28.6) command. Note that if you just select all sheet metal parts in the document andErase (6.13.8) them, Sheet Metal Context is kept, despite there are no sheet metal solids and no sheet metal features.

Figure 3.280

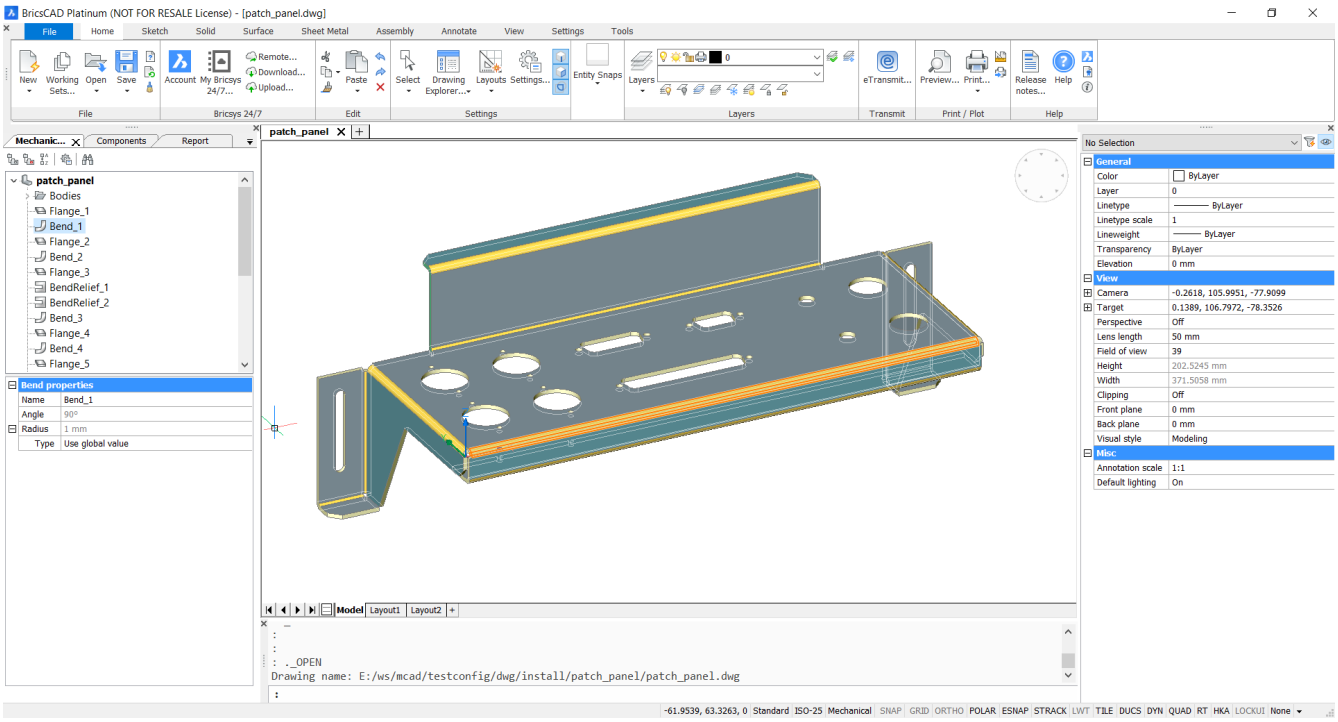

Mechanic...
Components
Report
patch_panel
Bodies
Flange_1
Bend_1
Flange_2
Bend_2
Flange_3
BendRelief_1
BendRelief_2
Bend_3
Flange_4
Bend_4
Flange_5
Component
Name
patch_panel
Description
File
E:\ws\mcad\testconfig\dw
Sheet metal
Thickness
2 mm
Modeling
Bend radius
1 mm
Sharp bend radius limit
10 mm
Bend relief width
1 mm
Relief extension
0.2 mm
Corner relief diameter
Minimal possible
Junction gap
0.002 mm
Junction alignment
Disabled
Flange split gap
0.2 mm
Flange split extension
0.2 mm
Rib profile radius
4 mm
Rib round radius
2 mm
Rib fillet radius
20 mm
Unfolding
Default K-Factor
0.27324

Working with Features

All features of a sheet metal part are listed in the **Mechanical Browser**.When you select a feature in the **Mechanical Browser**, its faces are highlighted in the model area: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/M
Note that **Sheet Metal** group of properties has changed to **Bend properties**. Thus clicking on feature in Mechanical Browser allows:

- locate feature on the part by highlighting its geometry
- see properties of given feature

Figure 3.281

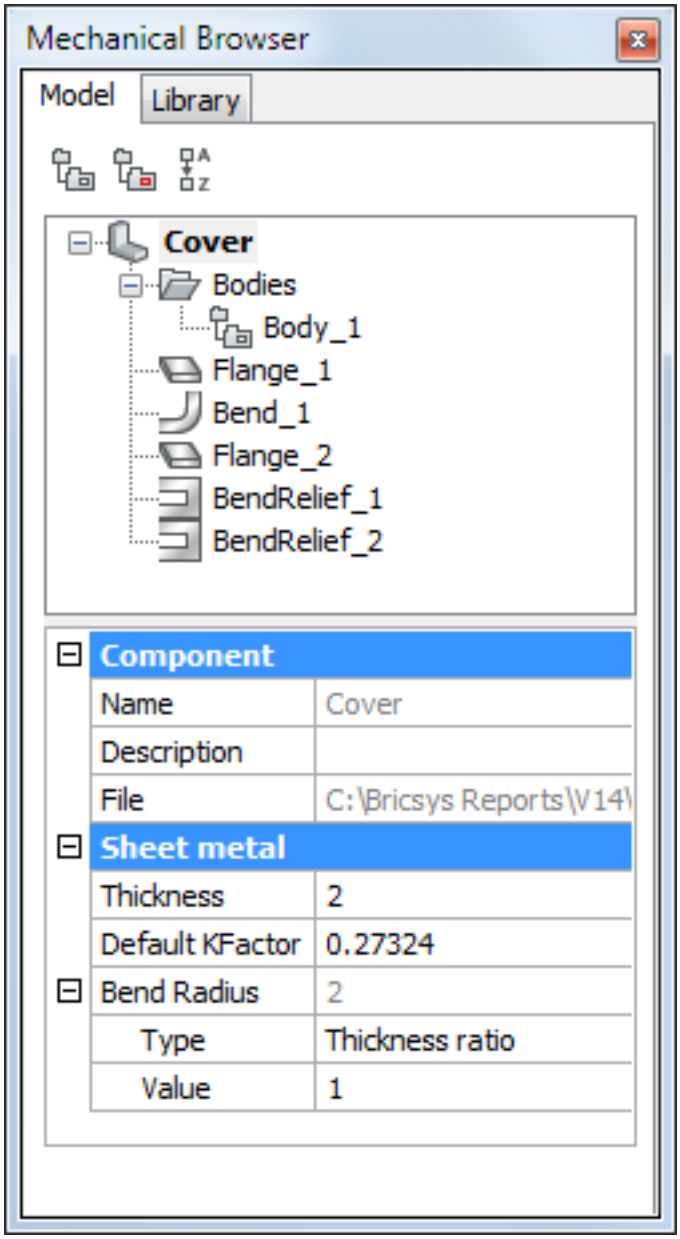


Changing the Thickness

To change the thickness of a sheet metal part:

1. Select the root node in the Mechanical Browser (3.2.2).
2. Type a value in the **Thickness** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/

Figure 3.282


Mechanical Browser
Model
Library
Cover
Bodies
Body_1
Flange_1
Bend_1
Flange_2
BendRelief_1
BendRelief_2
Component
Name
Cover
Description
File
C:\Bricsys Reports\V14\
Sheet metal
Thickness
2
Default KFactor
0.27324
Bend Radius
2
Type
Thickness ratio
Value
1

Feature Operations

Mechanical Browser provides fast access to operations over given sheet metal feature, list depend upon the feature type. List of operations depends upon selected features and contains operations available for the selection. Selecting two bends results in operations for bend feature: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/M
Selecting flange and bend limits list to operations available for both features: (\vref {fig:C:/Dev/Task-2448/images/staticbricsy

Figure 3.283

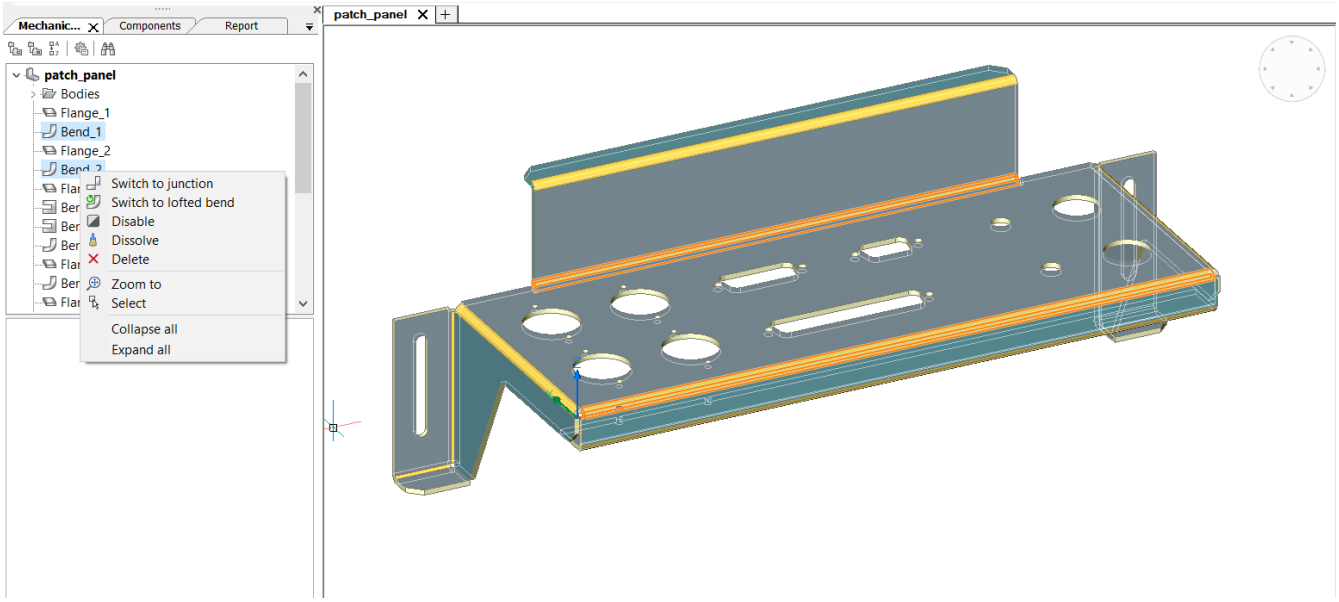
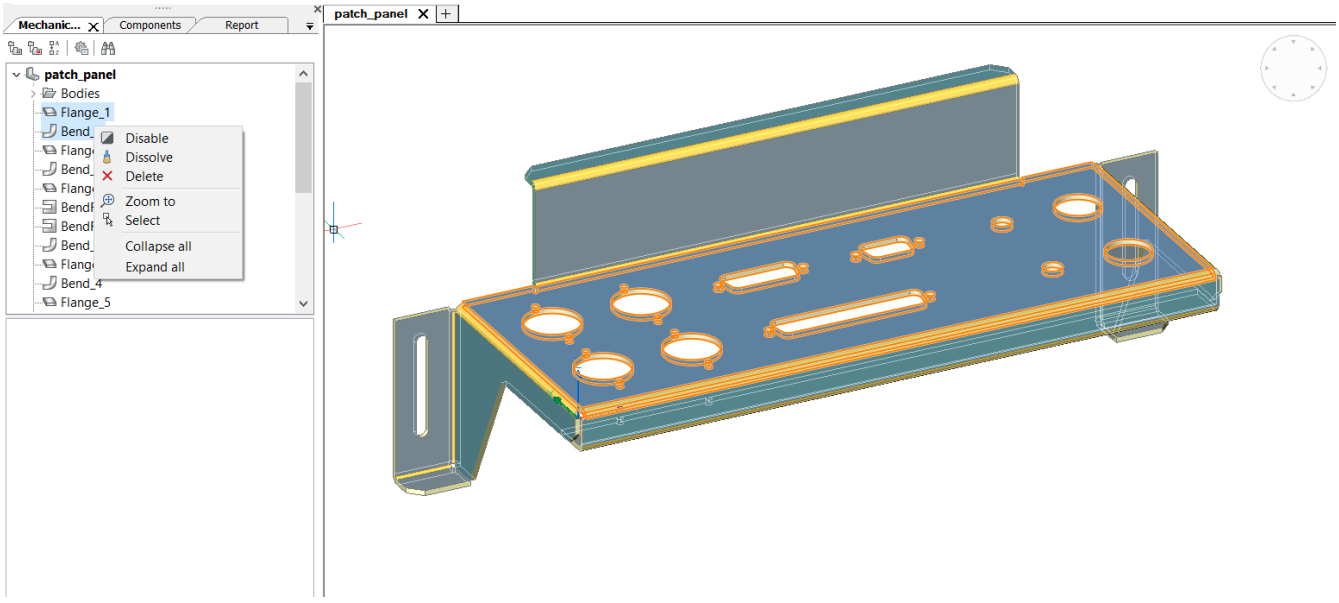


Figure 3.284



Understanding the Difference Between Disable, Dissolve and Delete

When you right click a feature in the **Mechanical Browser**, then select **Dissolve** from the context menu the selected feature is removed from the part, but it will keep its geometry. However, design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed If your intent is temporary forget about given feature, you can choose **Disable** option. The feature will remain in the **Mechanical Browser**, but the part will behave like the feature was dissolved. It is beneficial in contrast with dissolve, that feature still updates on geometry changes and you can turn it on when you want, avoiding recreating it or recognizing by SmConvert (6.28.4). Some of Sheet Metal features support **Delete** operation in the **Mechanical Browser**which is the analogue ofSmDelete (6.28.5) command. In this case the feature is removed from the browserand geometry is changed depending upon the type of the feature.

3.8.6 Sheet Metal Global and Feature Properties

Sheet metal features and context menus sit in the Mechanical Browser (6.8.29). SeeMechanical Browser for Sheet Metal (3.8.5)to understand the general principles. This article describes the particular properties and context menus for each node type in the Mechanical Browser. When working with sheet metal parts, clicking the root node in **Mechanical Browser**, displays the properties of entire drawing. Sometimes these properties are referred to as the Sheet Metal Context associated with the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Me
Sheet metal global properties are placed in the bottom part of the Mechanical Browser. **Global Properties** Thickness: global sheet metal material thickness. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechani
Bend radius: default radius of bend features; it can be changed per bend as well.

- Type: either thickness ratio or absolute value.
- Value: either ratio or value.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalBendRadi
Sharp bend radius limit : maximum bend radius to be considered a sharp (regular) bend. Used inSmConvert (6.28.4)andSmReliefCreate (6.28.22)commands to distinguish between regular and lofted bends.

- Type: either thickness ratio or absolute value.
- Value: either ratio or value.

Bend relief width: width of the cut near a single bend.

- Type: either thickness ratio or absolute value.
- Value: either ratio or value.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalBendRelie
Relief extension: size of the bend or corner relief extension.

- Type: either thickness ratio or absolute value.
- Value: either ratio or value.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalReliefExte
Corner relief diameter: default diameter of a circular corner relief.

- Type: minimal possible or absolute value.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalCornerRel
{fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalCornerReliefDia
Junction gap : gap between two adjacent flanges.

- Type: either thickness ratio or absolute value.
- Value: either ratio or value.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalJunctionG
{fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalJunctionGap2.p
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalJunctionG
{fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalJunctionGap4.p
Junction alignment : controls the alignment of junction faces with the corresponding corner relief faces.

- Type: either thickness ratio or absolute value.
- Value: either ratio or value.
- Junction Alignment: can be enabled(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical
  or disabled(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetal

Flange split gap: gap between thickness faces of a flange or the adjacent bend when splitting, see SmSplit (6.28.28).

- Type: either thickness ratio or absolute value.
- Value: either ratio or value.

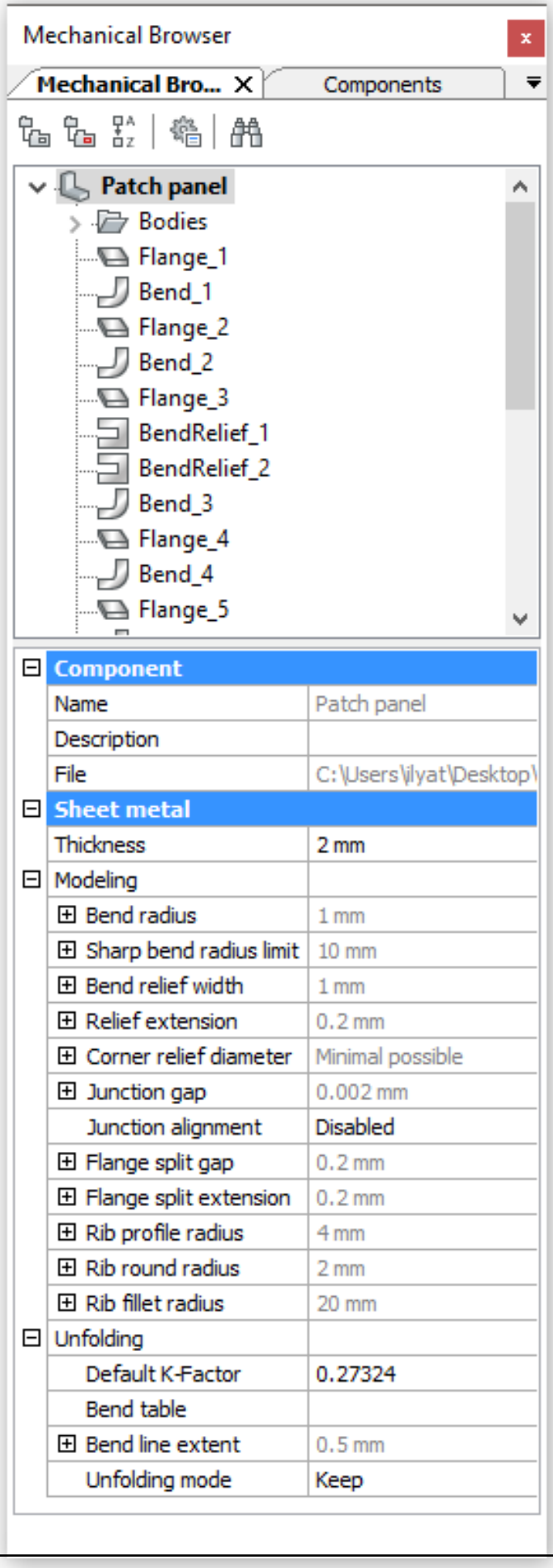
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalFlangeSpl
Flange split extension: controls the depth cut of an adjacent flange in case of a flange split.

- Type: either thickness ratio or absolute value.
- Value: either ratio or value.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalFlangeSpl
Rib profile radius: radius of a tool, which creates the rib.

- Type: either thickness ratio or absolute value.

Figure 3.285


Mechanical Browser
Mechanical Bro...
Components
Patch panel
Bodies
Flange_1
Bend_1
Flange_2
Bend_2
Flange_3
BendRelief_1
BendRelief_2
Bend_3
Flange_4
Bend_4
Flange_5
Component
Name
Patch panel
Description
File
C:\Users\ilyat\Desktop\
Sheet metal
Thickness
2 mm
Modeling
Bend radius
1 mm
Sharp bend radius limit
10 mm
Bend relief width
1 mm
Relief extension
0.2 mm
Corner relief diameter
Minimal possible
Junction gap
0.002 mm
Junction alignment
Disabled
Flange split gap
0.2 mm
Flange split extension
0.2 mm
Rib profile radius
4 mm
Rib round radius
2 mm
Rib fillet radius
20 mm
Unfolding
Default K-Factor
0.27324
Bend table
Bend line extent
0.5 mm
Unfolding mode
Keep

- Value: either ratio or value.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalRibProfile
Rib round radius: radius of the fillet between the flange and a rib.

- Type: either thickness ratio or absolute value.
- Value: either ratio or value.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalRibRound
Rib fillet radius: radius of the rib control curve filleting, seeWorking with Form and Rib Features (3.8.18).

- Type: either profile radius ratio or absolute value.
- Value: numeric value.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SheetMetalGlobalRibFilletRa
Unfolding: parameters for the unfolded representation. Default K-Factor: K-Factor for the bend radius equal to the thickness, seeK-Factor (3.8.7). Bend table: displays the Open dialog for bend tables in CSV file format. Bend line extent: controls how far bend a line extends beyond the unfolding boundary (zero and negative values are allowed).

- Type: either thickness ratio or absolute value.
- Value: either ratio or value.

Unfolding mode: controls the appearance of form features in the unfolded representation.

- Keep: keep the 3D representation of the form feature.
- Remove: completeley remove form features from the unfolded representation.
- Project: project all edges of form features as 2D entities.
- Contour: project only the contour of form features as2D entities.
- Symbol: place 2D symbols at the insertion points inunfolded representation.

The Sheet metal feature context menu displays when right clicking on a feature while feature properties displayed automatically when selecting a feature in the **Mechanical Browser.**

Feature Context Menus	Common tools: • Disable: temporarily disables a feature and skips its constraining. • Dissolve: removes a feature but keep its geometry. Design intent associated with the geometry of a dissolved feature will be removed. • Delete: same as theSmDelete (6.28.5) command. Removes features from the Mechanical browser and deletes the corresponding geometry inthe model. • Zoom to: zooms to the selected feature. • Collapse all: collapses the main component and all components and subcomponents. • Expand all: expands the main component and all components and subcomponents.
Flange feature	Right click a flange feature: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Flip: swaps the sides of a selected flange so that the reference face is on the opposite geometric side of the flange. This operation shifts the flange by its thickness, in order preserve the position of reference face.

Bend feature	Right click on a bend feature:  (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Switch to junction: switches to a junction feature type with a gap between adjacent flanges. • Switch to lofted bend: switches to a lofted bend feature.
Lofted Bend feature	Right click on a lofted bend feature:  (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Switch to junction: switches to a junction feature type with a gap between adjacent flanges. • Switch to bend: switches to bend feature.
Wrong Bend feature	Right click on a wrong bend feature:  (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Repair feature: repairs a wrong bend feature by creating a bend feature with proper geometry, if possible.
Bend Relief feature	Right click on bend relief feature:  (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Switch to smooth, round or rip: switches a bend relief to the selected feature type.
Corner relief feature	Right click on corner relief feature:  (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Switch to V-type or circular: switches a corner relief feature to the selected feature type.
Junction feature	Right click on junction feature:  (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Switch to bend: switches to a bend feature. • Switch type of the feature: switches the alignment mode between symmetrical and aligned with one of the flanges.
Miter feature	Right click on miter feature:  (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee

Form feature	Right click on form feature:  (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Select same feature: selects all form features of the same type in the model. • Explode to flanges and bends: explodes a form feature into a set of flanges and bends, if possible. • Repair feature: recalculates the insertion point of a form feature. • Replace...: displays the Open dialog box in order to choose a replacement form feature.
Feature Properties	Common property: • Name: the name of the selected sheet metal feature. Optionally type a name for the feature. The default name of a feature is the feature type plus an increment, e.g. Flange_1, Flange_2, ...
Flange feature	Click a flange feature: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Thickness: current thickness. The value is read-only; it equals the sheet metal component thickness property.
Bend feature	Click a bend feature: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Angle: current angle of the bend feature; can be changed by theSmFlangeRotatecommand. • Radius: current bend radius. • Radius type: – Thickness ratio: ratio of thickness multiplied by a coefficient – Absolute value: the global value (main sheet metal component properties).
Bend relief	Click a bend relief feature: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Relief type: current relief type; either rectangular, smooth, round or rip • Width: width of the cut near the bend; either global, thickness ratio or absolute.
Corner relief	Click a corner relief feature: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Relief type: current relief type; either rectangular 2 bends, rectangular, v-type or circular • Width: width of the cut near corner.; either global, thickness ratio or absolute.

Figure 3.286

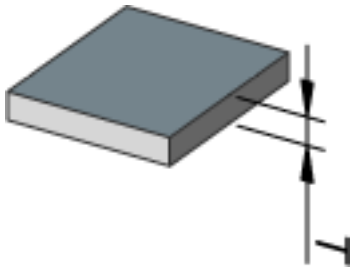
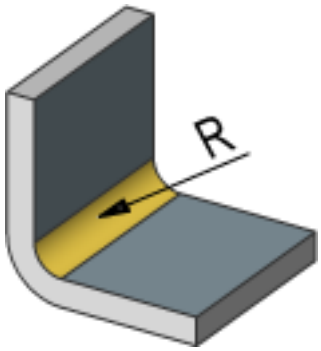


Figure 3.287



Junction feature	Click a junction feature: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Junction gap: gap between two adjacent flanges. • Junction gap type: either global, thickness ratio or absolute. • Connection type: controls the relative position ofadjacent flanges; either symmetric or adjacent to one of the flanges. • Junction alignment: controls the position of the faces of a junction feature; either global, enabled (aligned with the corresponding corner relief feature faces) or disabled
Miter feature	Click a miter feature: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Miter gap: gap between the side faces of the miter. • Type: either thickness ratio or absolute.
Form feature	Click a form feature: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee • Component name: name of form feature component definition. • Parameters: contains the parameters of a parametric sheet metal form feature.

Figure 3.288

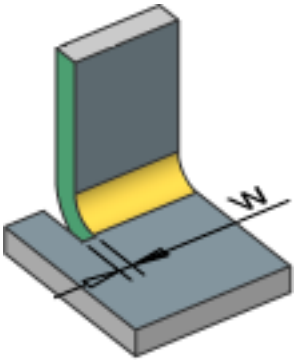


Figure 3.289

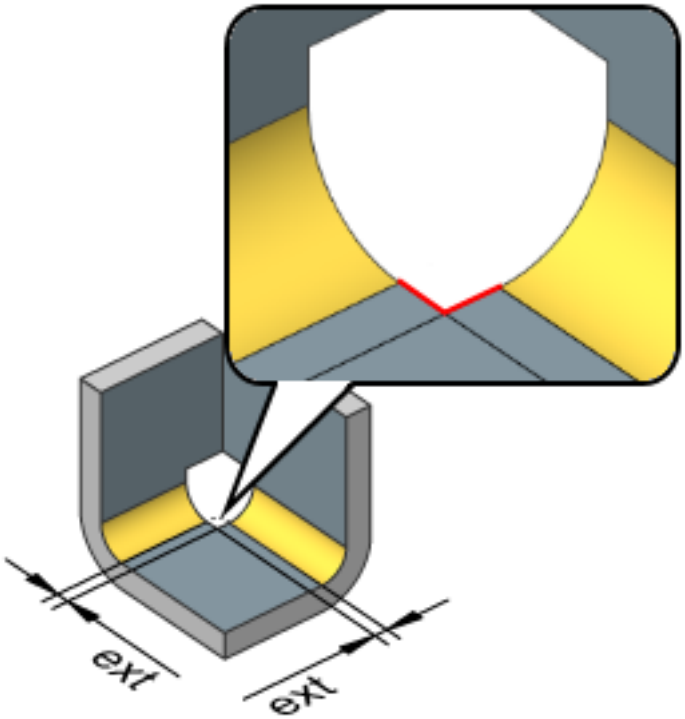


Figure 3.290

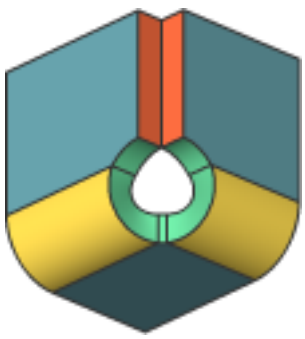


Figure 3.291

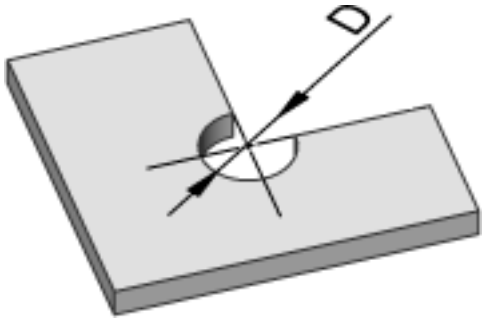


Figure 3.292



Figure 3.293

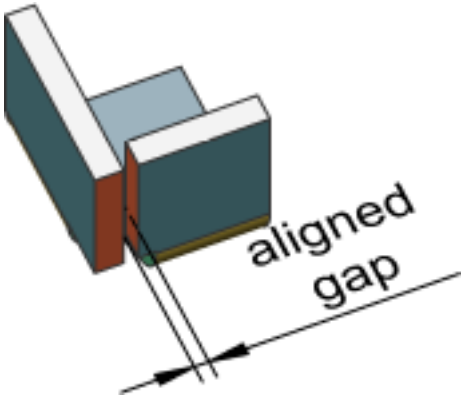


Figure 3.294

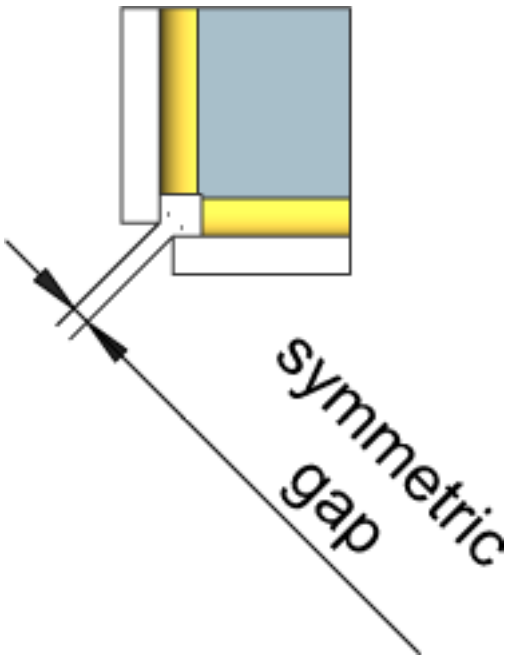


Figure 3.295

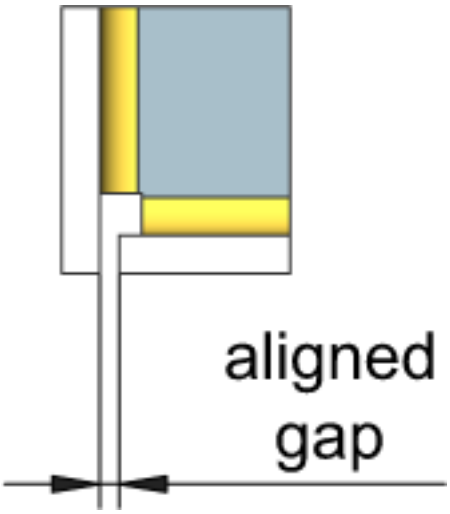


Figure 3.296

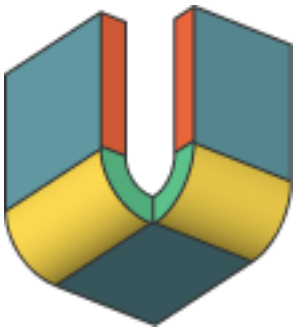


Figure 3.297

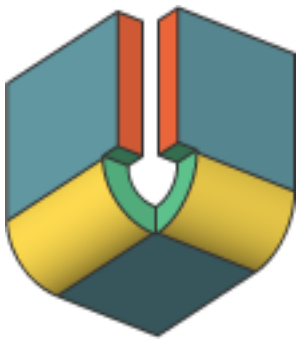


Figure 3.298

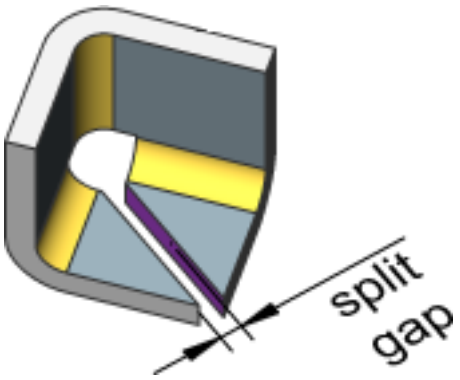


Figure 3.299

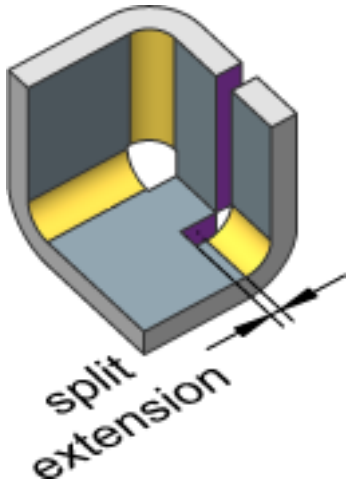


Figure 3.300

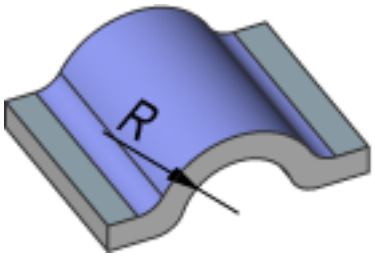


Figure 3.301

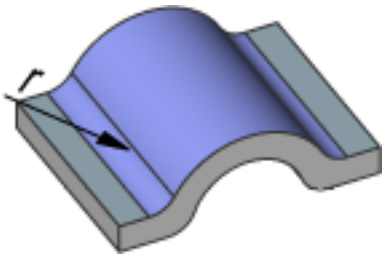


Figure 3.302

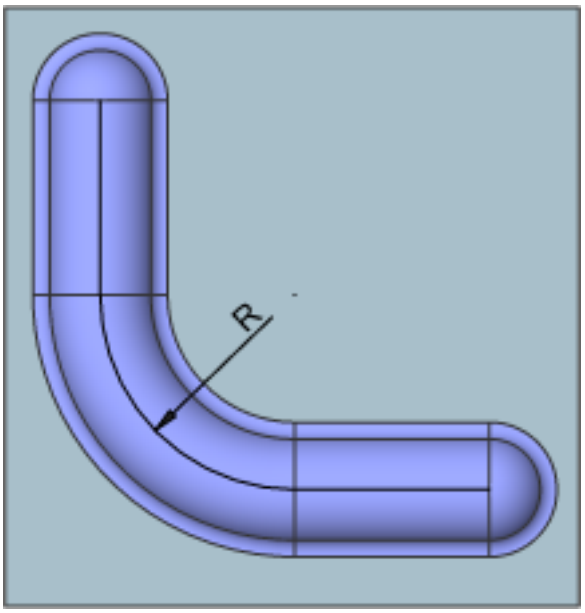


Figure 3.303

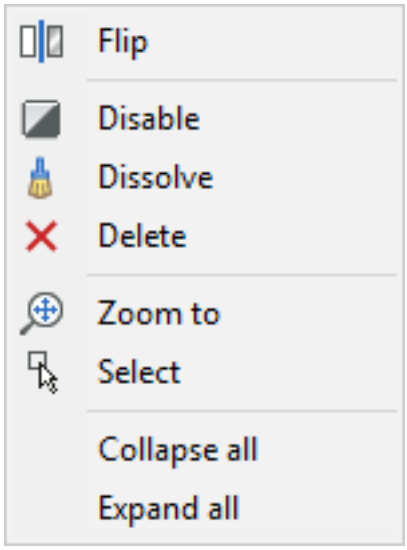


Figure 3.304

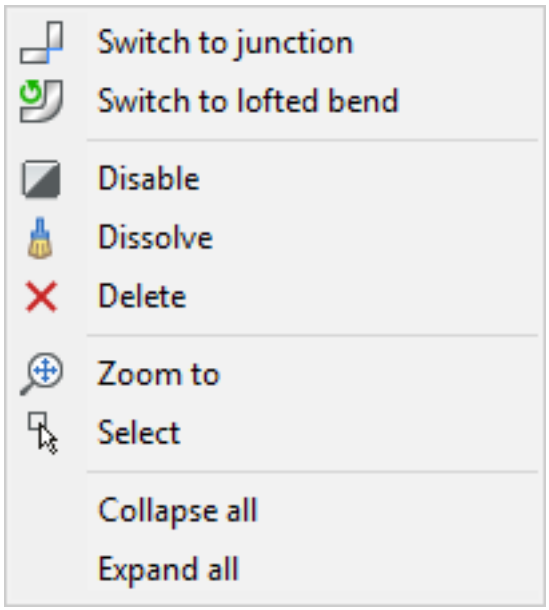


Figure 3.305

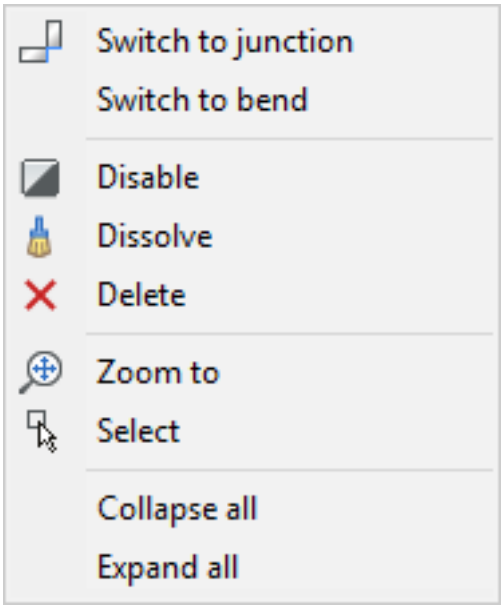


Figure 3.306

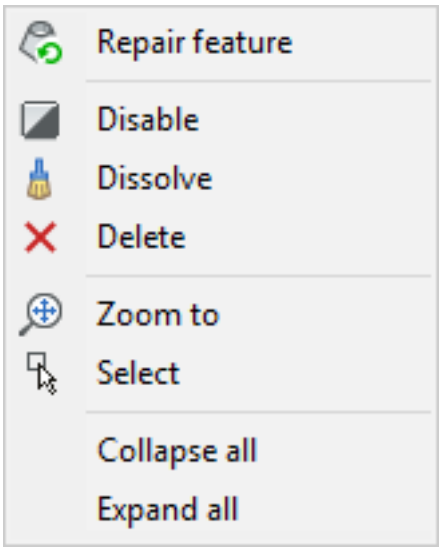


Figure 3.307

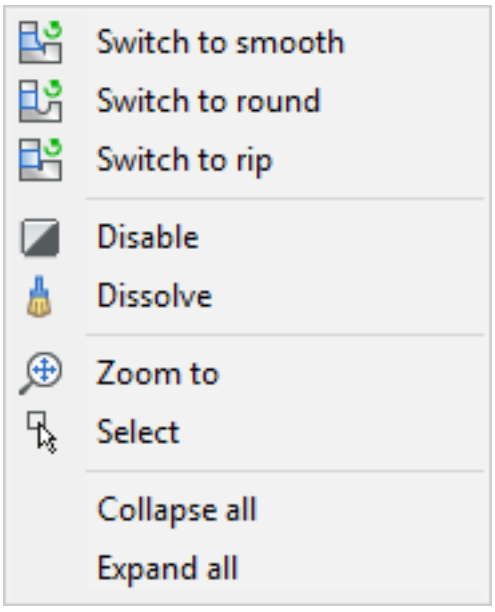


Figure 3.308

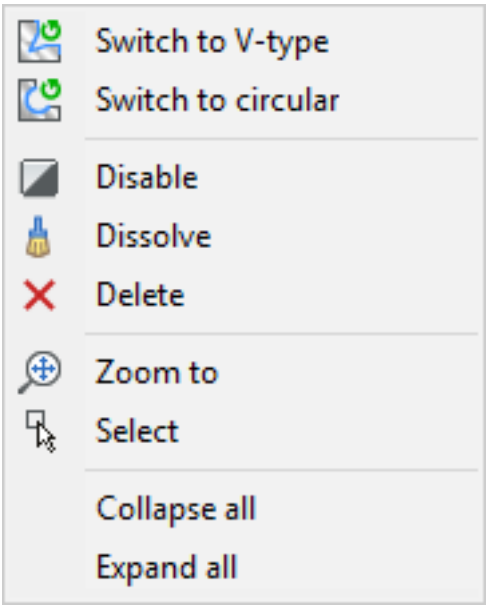


Figure 3.309

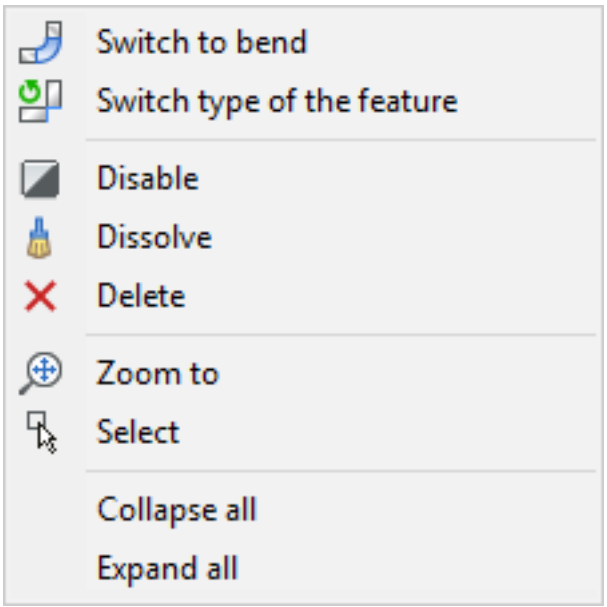


Figure 3.310

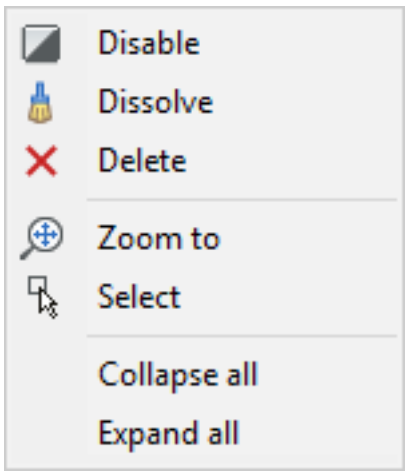


Figure 3.311

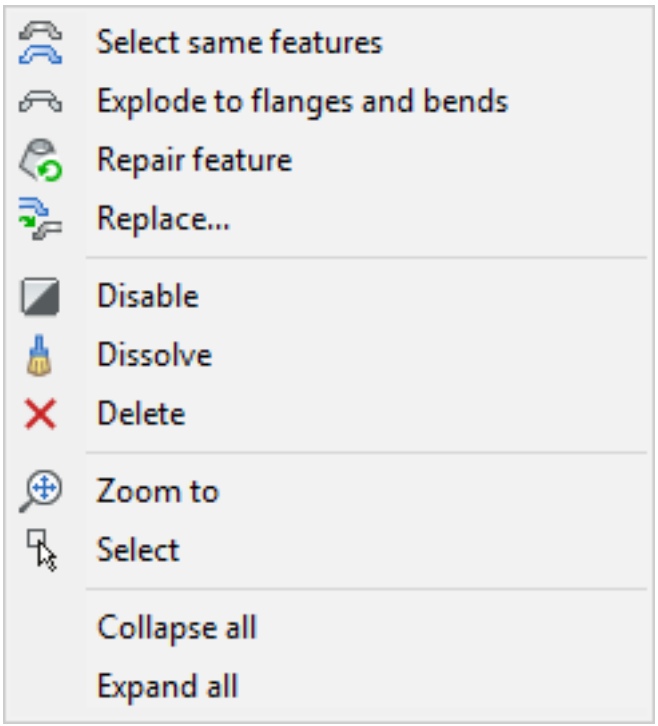


Figure 3.312

Flange properties	
Name	Flange_1
Thickness	2 mm

Figure 3.313

Bend properties	
Name	Bend_1
Angle	90°
Radius	1 mm
Type	Use global value

Figure 3.314

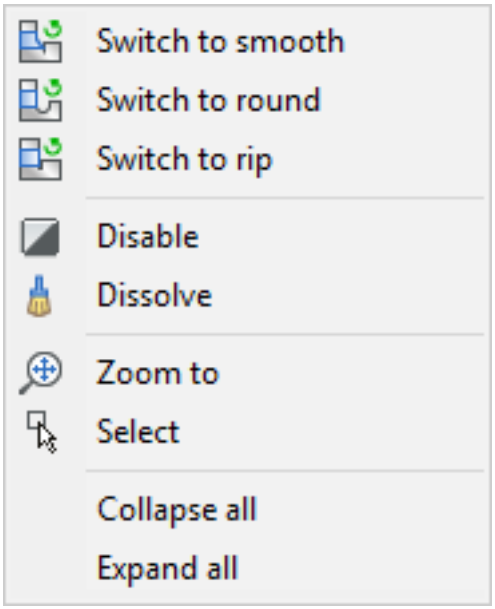
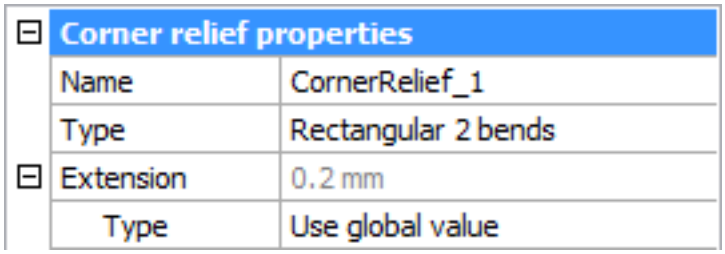
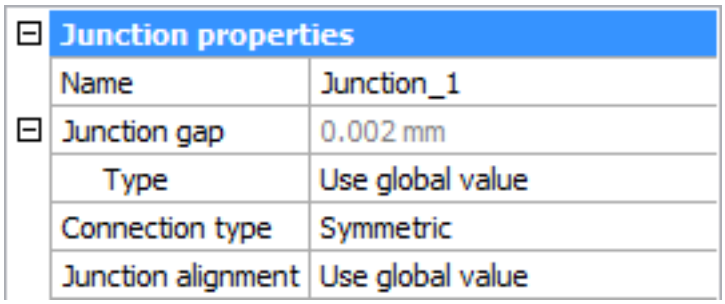

Switch to smooth
Switch to round
Switch to rip
Disable
Dissolve
Zoom to
Select
Collapse all
Expand all

Figure 3.315



Corner relief properties	
Name	CornerRelief_1
Type	Rectangular 2 bends
Extension	0.2 mm
Type	Use global value

Figure 3.316



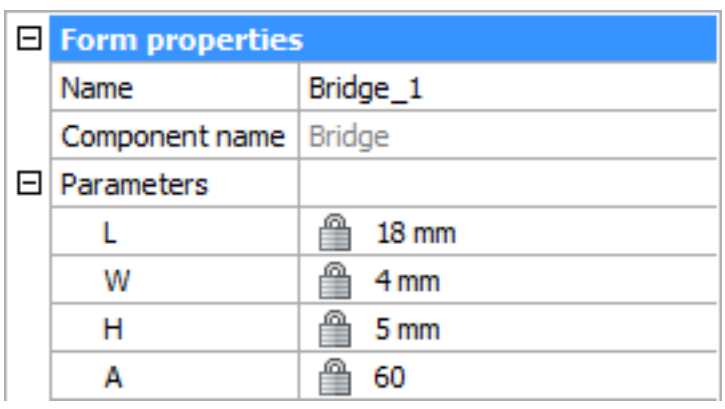
Junction properties	
Name	Junction_1
Junction gap	0.002 mm
Type	Use global value
Connection type	Symmetric
Junction alignment	Use global value

Figure 3.317



Miter properties	
Name	Miter_1
Miter gap	0.2 mm
Type	Use global value

Figure 3.318



Form properties	
Name	Bridge_1
Component name	Bridge
Parameters	
L	18 mm
W	4 mm
H	5 mm
A	60

Figure 3.319

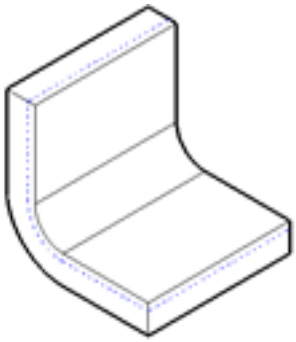
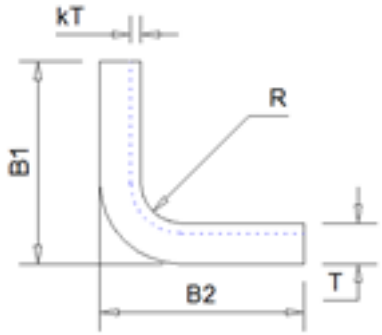


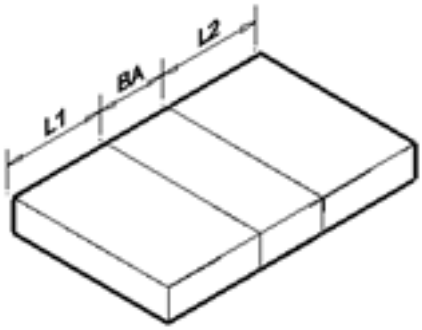
Figure 3.320



3.8.7 K-Factor

When a metal sheet is bent with a press break, its material is deformed, which is the reason of the difference in overall length of a sheet in folded and unfolded states. BricsCAD automatically takes this deformation into account when you unfold a sheet metal part. Two approaches are used to compute it: a simplified one (K-Factor) and a more advanced one (Bend Table (3.8.8)). The material deformation properties for bending is based on the assumption that there is internal surface for the sheet metal part which is not deformed when the sheet is bent. From the variety of internal surfaces the one is selected, which is equidistant from one side of the bend. The surface is local for each bend, and for simple cases it can be propagated on entire part, like the one on the picture. We assume that this surface is not stretched stretch during bending. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/s
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/
T: thickness of the sheet metal part R: internal radius of the bend The K-factor (k) is the ratio of the location of the neutral surface to the material thickness. As a result the neutral surface lies at a distance kT from the internal surface of the bend. The bend radius of the neutral surface equals R + kT. The K-Factor is a simple geometric calculation of the location of the neutral surface. Forming stresses and other unknown (error) factors are not taken into account. The K-Factor depends on many factors, including the kind of material, bending type, tools, etc ... The K-Factor typically lies between 0.3 and 0.5. The default K-Factor for a bend radius equal to the material thickness (T) is 0.27324. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03Drawi
In the unfolded state of a sheet metal part, the flanges (L1 and L2) are not stretched. The bend part is indicated as BA: bend allowance. The BA equals the length of the unfolded neutral surface: BA = BendAngle * (R + kT) In real world practice it is hard to measure the K-Factor or the bend allowance. The following formula allows to calculate the tangential Bend Deduction (BD): BD = B1 + B2 - Unfolded Length = B1 + B2 - (L1 + L2 + BA) For simple cases the K-Factor value can be overridden in the browser, for maximal precision a bend table containing tangential bend deductions has to be provided. For each sheet metal part, you can either specify a K-factor or use the default one.

Figure 3.321



Changing the K-Factor for a Sheet Metal Part To change the K-Factor for a sheet metal part, select the root node in the Mechanical Browser (3.2.2) and type a value in the **K-Factor** field. The value must lie in the range [0, 1], since the neutral surface is located inside the sheet metal part. In BricsCAD the K-Factor is defined for bend radius equal to the material thickness (R/T = 1); to compute the value of K-Factor for an arbitrary bend radius, BricsCAD uses a special interpolation technique which is proved to be valid for industrial sheet metal applications. The image below shows the interpolation strategy: for R/T < 1 the linear interpolation is done between 0 and 0.27324, for 1 < R/T < 4 the K-Factor is linearly interpolated between 0.27324 and 0.5 and if R/T is greater than 4, the K-Factor is constant and equals 0.5. Physically it means that if the bend radius is much larger than the thickness (at least 4 times), the material stretch is can be neglected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/SMKFactorGraph1
The only K-Factor parameter one can control in the browser is for R/T. I.e. if we set the value to 0.6, the interpolation law leads to the following result: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Dra
Pic: the varying K-Factor in the browser leads to the following effective curve. The distinguishing feature of the curve is that varying R in range of (0, T], L1 + L2 + BA will give the same value. This is explained by the fact that for radii less than the thickness they relate to are for design convinience rather than for some physical process modeling, since during bending tool radius is usually greater or equal to thickness. Lets review the example with varying K-Factor by control curve setting (the bend radius is kept constant). T = 2, R = 2, B1 = 100, B2 = 100 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/SMKFactorImage1
Setting K=0 gives L1=L2=96, Unfolded Length = 96, thus BA is 3.14. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel
This value fits the formula exactly: a straight angle in radians roughly equals 1.57 (PI/2). The default K-Factor 0.27324 gives an Unfolded Length = 196. Or: L1=L2=96 (flanges are not deformed), the BA = 4 as we can compute from the dimensions (196 - 2*96) or from the Bend Allowance formula: BA = BendAngle * (R + kT) = 1.57 * (2 + 2*0.27324) = 4 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03D
Finally, maximizing K-Factor results in an Unfolded Length = 198.28. This is the maximal value one can achieve in this example since the neutral surface is taken from the external side of the bend. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/SMKFactorImage4
The Bend Deduction (BD) for k = 0.27324: BD = B1 + B2 - Unfolded Length = 100 + 100 - 196 = 4

Figure 3.322

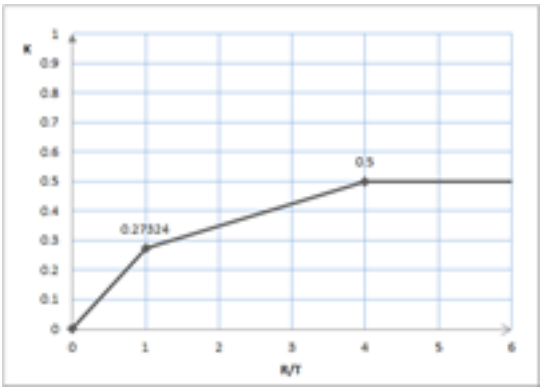


Figure 3.323

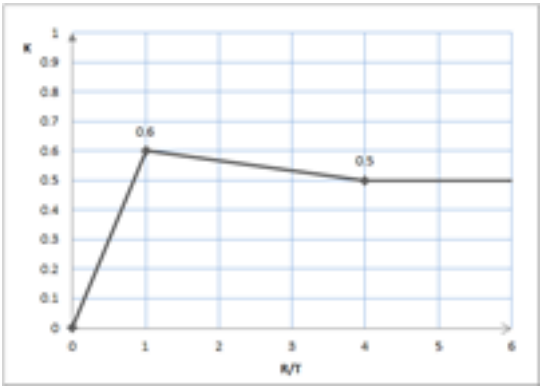


Figure 3.324

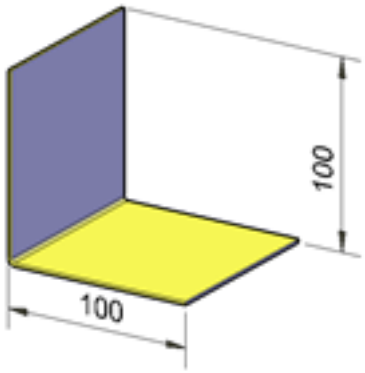


Figure 3.325

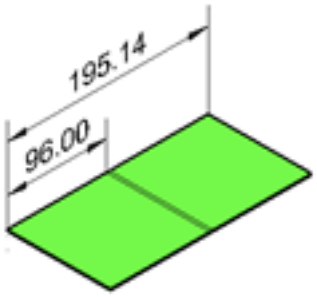


Figure 3.326

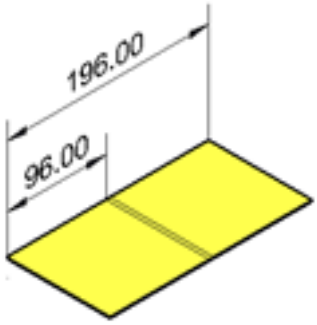


Figure 3.327

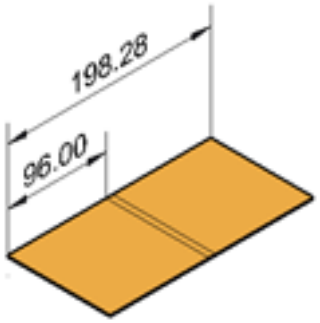


Figure 3.328

	A	B	C	D	E	F	G
1	BricsCAD						
2	Version	1					
3	AngleType	Internal					
4	LengthType	BendDeductionTangent					
5							
6	Thickness	1					
7	AngleValues		45	60	90	120	135
8	Radius	1	1	1.333	2	0.976	0.657
9	Radius	2	0.466	1.081	2.31	1.004	0.641
10	Radius	3	-0.069	0.828	2.621	1.033	0.624
11							
12	Thickness	1					
13	DieWidth	10					
14	AngleValues		45	60	90	120	135
15	Radius	1	1.225	1.533	2.15	1.076	0.732
16	Radius	2	0.616	1.214	2.41	1.071	0.691
17	Radius	3	0.006	0.895	2.671	1.066	0.649

3.8.8 Bend Table

Bend tables are a more reliable way to express material deformation properties than K-Factor (3.8.7). When you physically bend a flat sheet of a particular material, you can measure its length before and after this process. You can repeat this procedure for different bend angles, bend radius, and sheet thickness and save the measurements in a bend table. Then these measurements can be taken into account to compute the correct unfolded length for all parts made of the same material. In a bend table, you can add the measurements for as many different bend angles as you want. However, in most cases it suffices to measure the length for a 90 degrees angle. BricsCAD will automatically compute the length for other bend angles using a reliable interpolation technique. Bend tables are stored in a *.csv (comma separated values) file format. You can create such a table in an ASCII text editor, such as Notepad, but a more reliable and easier way is to create a table in a spreadsheet, such as Microsoft Excel or LibreOffice and then export it to .csv file format. Bend tables in BricsCAD have the following structure: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/S
The following rules apply:

- The content of the first cell reads ”BricsCAD”.
- **Version**: currently supported version is 1.
- **AngleType**: Currently only internal bend angles are supported: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
- **LengthType:** the semantics of the value in the cells of the bend table corresponding to a particular bend angle and bend radius. Now only bend deduction measured from the tangent point is supported (”BendDeductionTangent”). Bend deduction (BD) is the difference between the sum of the lengths of two flanges measured to the tangent point on a 3D model and the length of the same fragment in the unfolded state: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/BendTableInternalAngle2.pn
In depth: methodics to compute tangential bend deduction for acute and obtuse angles. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/BendTableInternalAngle3.png})
- Within the defined header, several tables can exist. For each table, the first key is the of the sheet metal part, the second key, **DieWidth**, is optional. The **Thickness / DieWidth** pairs must be unique. If the **DieWidth** is not defined, only one unique **Thickness** value key is allowed, but multiple thickness values can be added. When a bend table is used to control the unfolding process, the **Thickness** and **DieWidth** , defined in the sheet metal part, must exist in the bend table. If the table is not found the default K-Factor policy is used.
- Once the bend table is found in the .csv, it will be used to unfold the sheet metal part. Using the sample bend table above, let’s have a look ata few cases: T = Thickness, R = Internal Bend Radius, A = Bend Angle
  - T = 1, R = 2, A = 45. Then after unfolding it will obtain the BD = 0.466, because both R and A are found in the bend table for the given thickness.

Figure 3.329

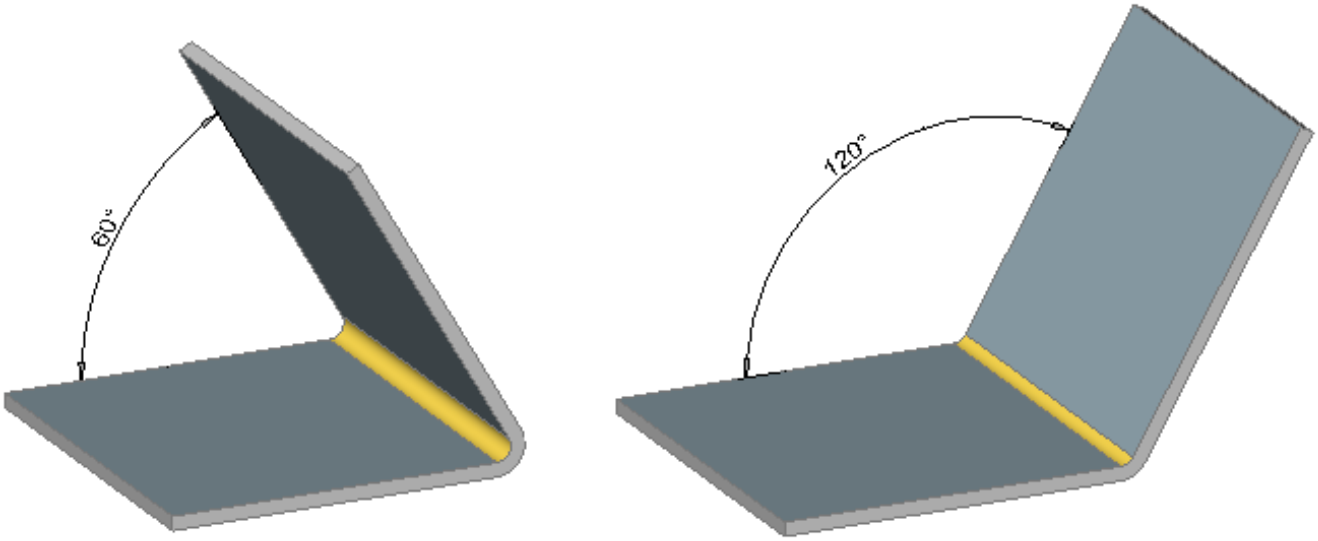


Figure 3.330

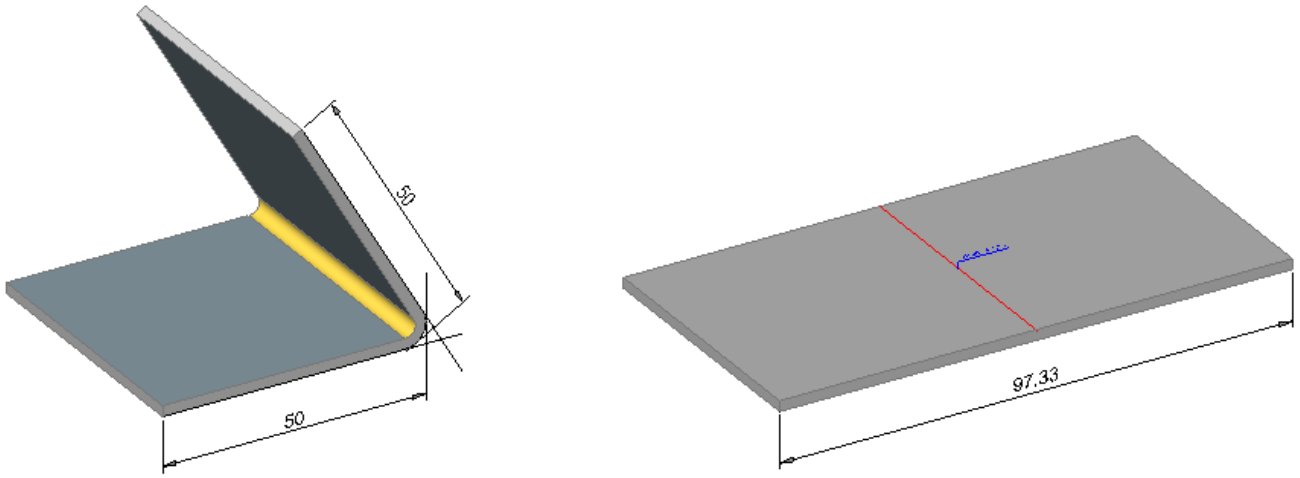
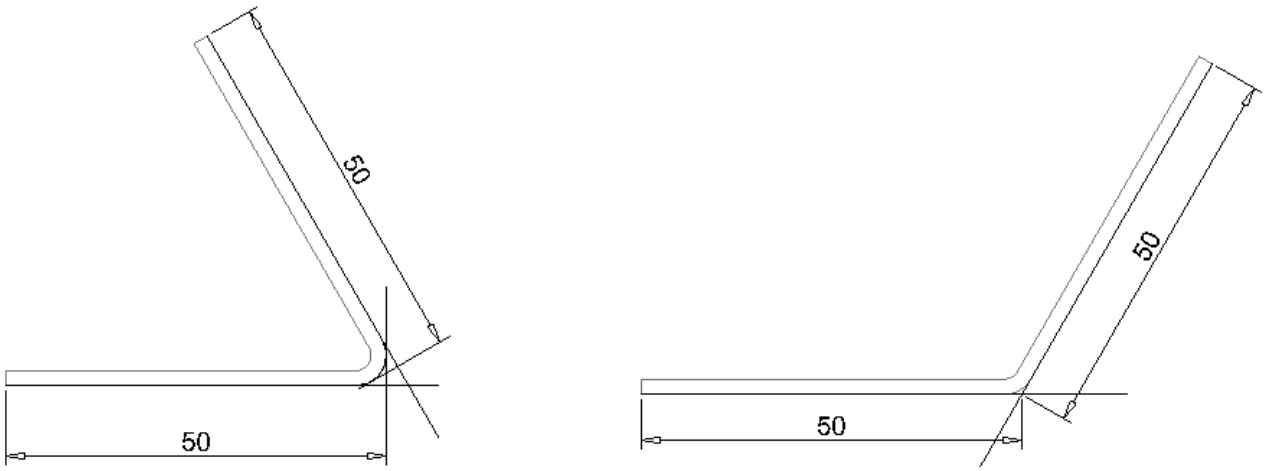


Figure 3.331



  - T = 1, R = 2.5, A = 45. A = 45 is found, but R = 2.5 is missing: the interpolation of BD between R=2 and R=3 is used.
  - T = 1, R = 2.5, A = 75. Neither R nor A is found in the table: the following cells are used to calculate the value of BD: (R=2, A=60), (R=3, A=60), (R=2, A=90) and (R=3, A=90).

  The top-of-the-edge algorithm is used to calculate the interpolation. For example, linear interpolation on adjacent BD values render unnatural results, which will not match the real bending results. Also keep in mind that for a given T, R and A not all BD values make sense: a random one might place the neutral surface outside the sheet metal part, or expressed in K-Factor (which can be deduced from BD), the K-Factor can be out of range [0, 1]. In this case, the BD for the default K-Factor will be used.

- If you produce your sheet metal parts with different tools, you can describe them in one bend table using **DieWidth** parameter. A sample bend table is provided for your convenience in the **[BricsCAD Installation folder]\Samples\Mechanical\bend_tables**. Please notice that writing rights to this folder might be limited. In such case you need to copy the bend tables to a different folder in which you have full access rights.

To set the bend table for your sheet metal part:

1. Select the root node in the Mechanical Browser (3.2.2).
2. Select the **Bend table** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Dr
3. Click the Browse button ( ).
4. Select the *.csv file then double click or press the **Open** button.

Figure 3.332



To update the bend table set previously:

1. Select the root node in the Mechanical Browser (3.2.2).
2. Select the **Bend table** field.
3. Press the **Update** button ().

If the bend table is not assigned to the document or has no **DieWidth**secondary key, the **DieWidth** parameter can not to assigned to single bend or all the bends. In order to make the assignment, do the following steps: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/S

1. Select .csv file **DieWidth** secondary keys and thickness matching the current part. On given example we assume that part has thickness of 1 mm and bend radius also equals to 1. We will vary bend deduction to different tools: the first subtable is when the tool is not specified, second subtable is for the case when tool **DieWidth** is equal to 20, and the last one is for a tool with **DieWidth** equal to 30.
2. Assign the bend table to the sheet metal part.
3. Check that in the Mechanical Browser (3.2.2) new fields appeared: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscon In this global setting we can select any default **DieWidth** value suitable for the part thickness. The value will be assigned by default to the bends in the part, which can be checked by the property of the bend (click on the bend in the Mechanical Browser (3.2.2)): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui
4. However, we can select a specific tool which will be used for the given bend by expanding the drop-down list: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingS
5. With the given selection, anunfolding of Bend_1 will use a third subtable from our bend table for bend deduction and uses the value 1.8. For the rest of the bends, unless **DieWidth** is set for them, the default no-die-width value will be used, which will trigger first the bend table and uses value 2.1 for 90-degrees with a bend radius equal to 1.

NOTE	After assigning the **DieWidth** to the bends and saving the document, the value is saved to the bends. However, next time the drawing is opened, in order to assign the **DieWidth** value to another bend, you have to reload the bend table first (press the green arrow button), see **To update the bend table set previously**. After the reload the values from the bend table will be available.

Figure 3.333

	A	B	C	D
1	BricsCAD			
2	Version	1		
3	AngleType	Internal		
4	LengthType	BendDeductionTangent		
5				
6	Thickness	1		
7	AngleValues		90	
8	Radius	1	2.1	
9				
10	Thickness	1		
11	DieWidth	20		
12	AngleValues		90	
13	Radius	1	1.7	
14				
15	Thickness	1		
16	DieWidth	30		
17	AngleValues		90	
18	Radius	1	1.8	

Figure 3.334

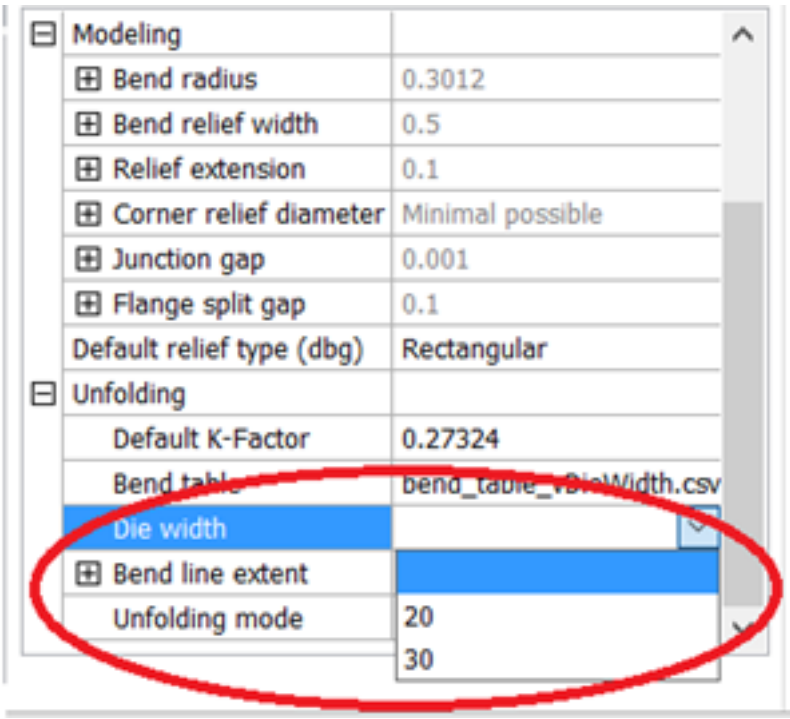


Figure 3.335

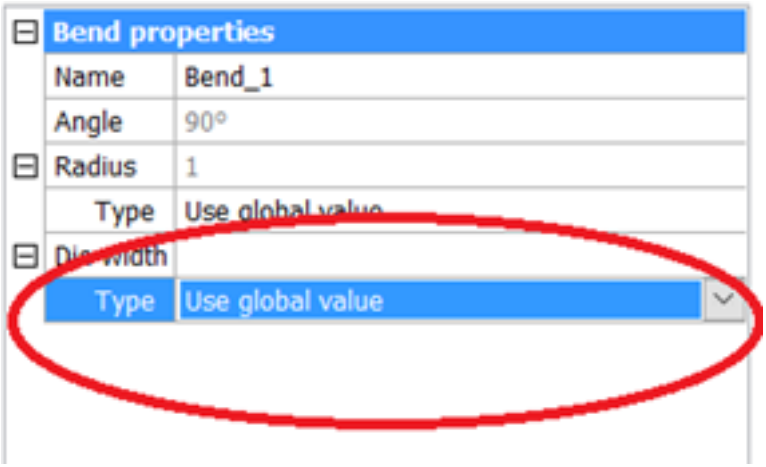
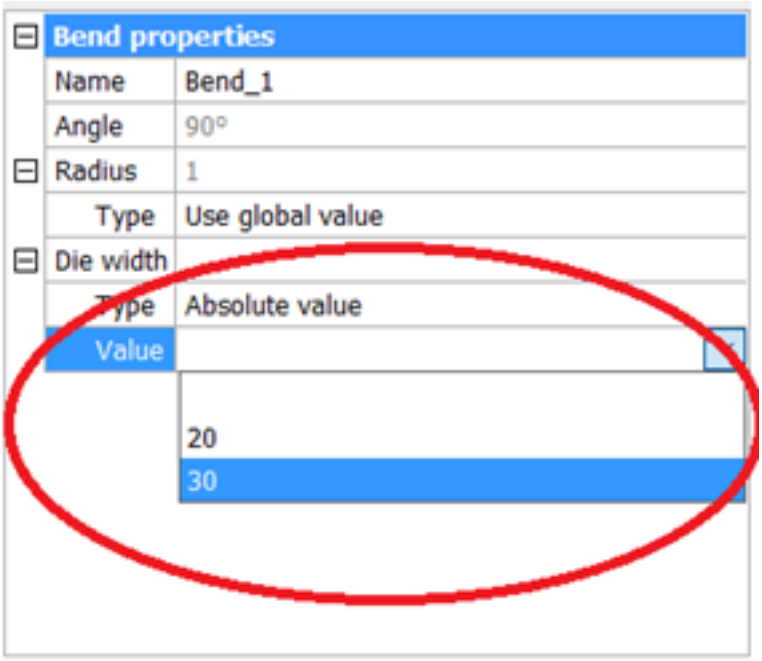


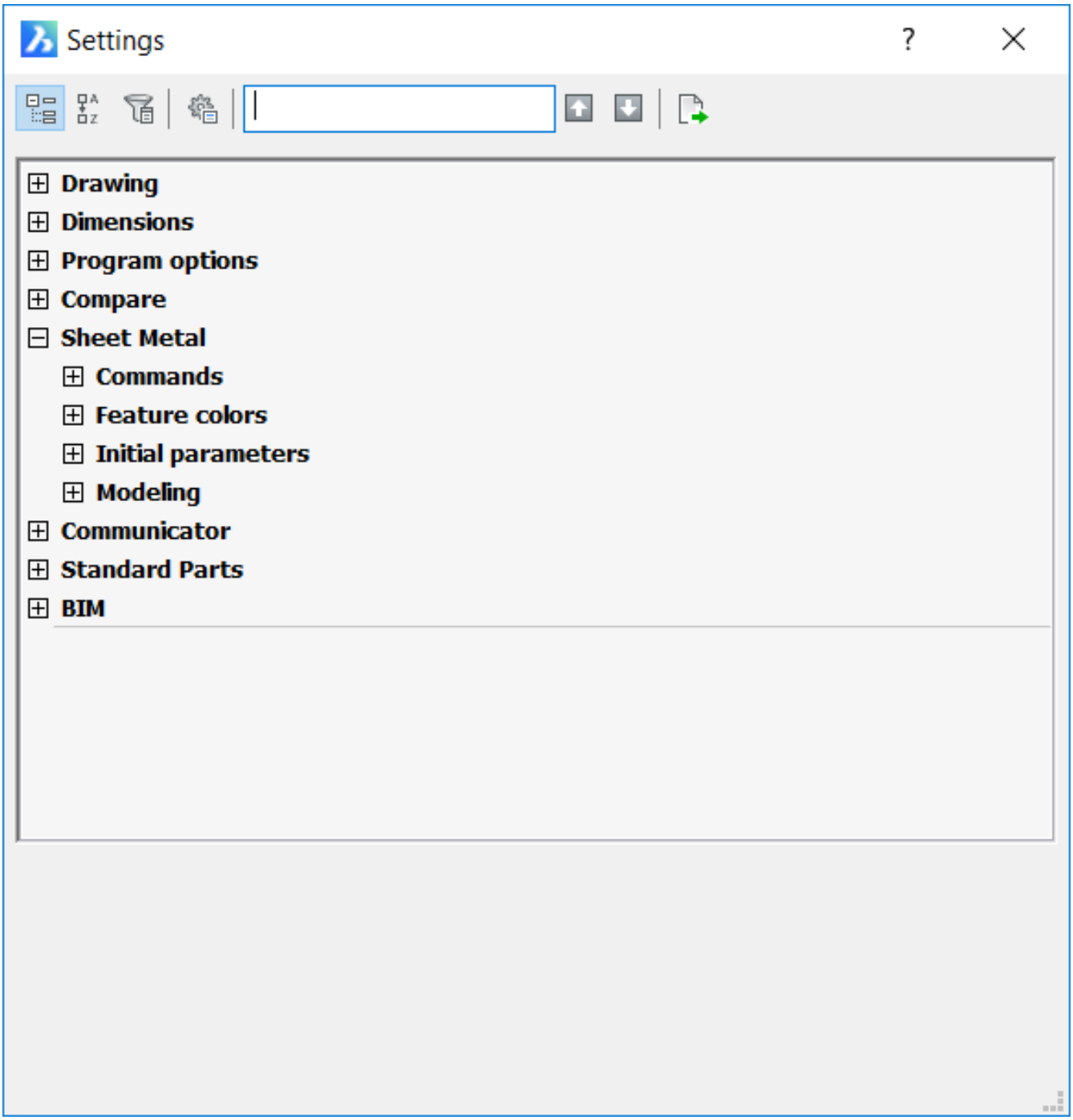
Figure 3.336



3.8.9 Sheet Metal Settings

Sheet Metal section inSettings dialog (1.13.11) controls different properties of Sheet Metal application. Properties are split into four groups: **Commands**, **Feature colors**, **Initial parameters** and **Modeling**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmSettingsSettings.png}) **Note:** Most of the sheet metal settings are stored in the drawing and it is recommended create or to start with one of the sheet metal template files (1.14.1) DWT.

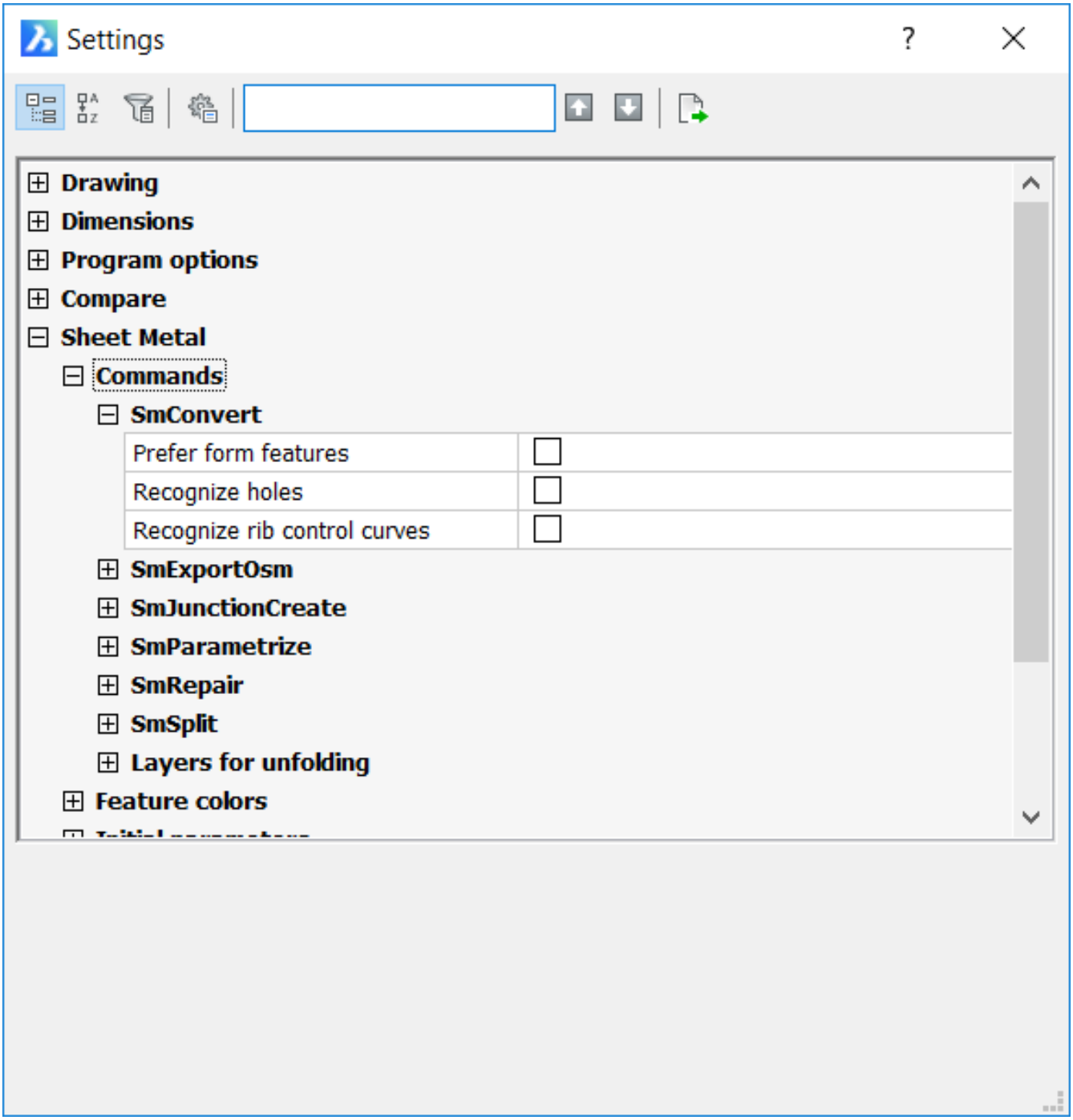
Figure 3.337


Settings
?
Drawing
Dimensions
Program options
Compare
Sheet Metal
Commands
Feature colors
Initial parameters
Modeling
Communicator
Standard Parts
BIM

Commands

The **Commands** section contains settings which control the behavior of Sheet Metal commands. **Note:** Before V19 these controls were available as options in the commands and were moved to simplify command interface. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmSettingsCommand.png}) The Layers for Unfolding setting determines the appearance of unfolded parts, see Unfolding Sheet Metal Parts and Exporting to CAM (3.8.15)for more details.

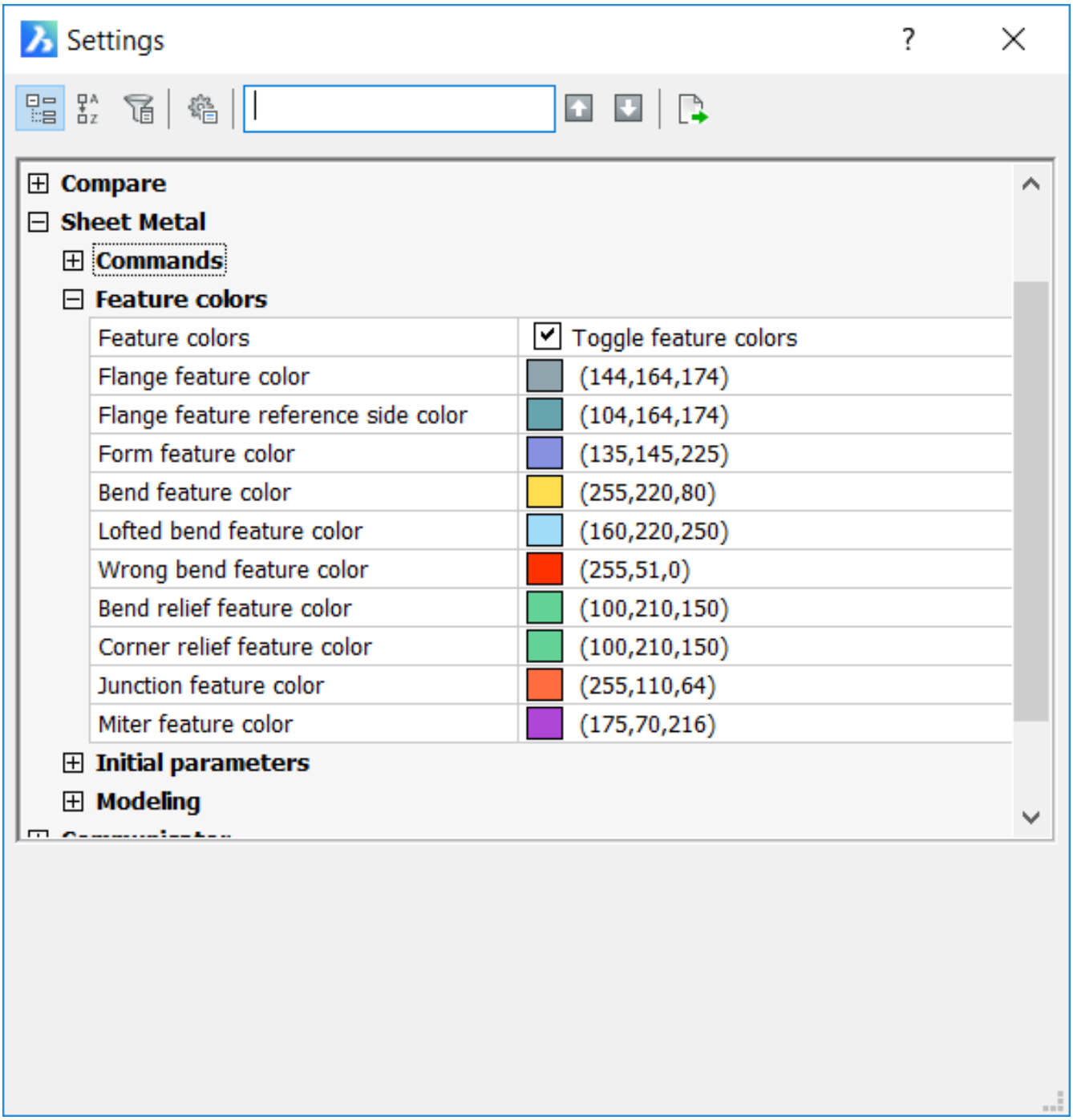
Figure 3.338


Settings
Drawing
Dimensions
Program options
Compare
Sheet Metal
Commands
SmConvert
Prefer form features
Recognize holes
Recognize rib control curves
SmExportOsm
SmJunctionCreate
SmParametrize
SmRepair
SmSplit
Layers for unfolding
Feature colors

Feature Colors

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmSettingsFeatureColors.p
Feature colors settings are used to visual distinguishing different types of sheet metal features.You can adjust colors per-feature, seeFeature Coloring (3.8.4) for details.

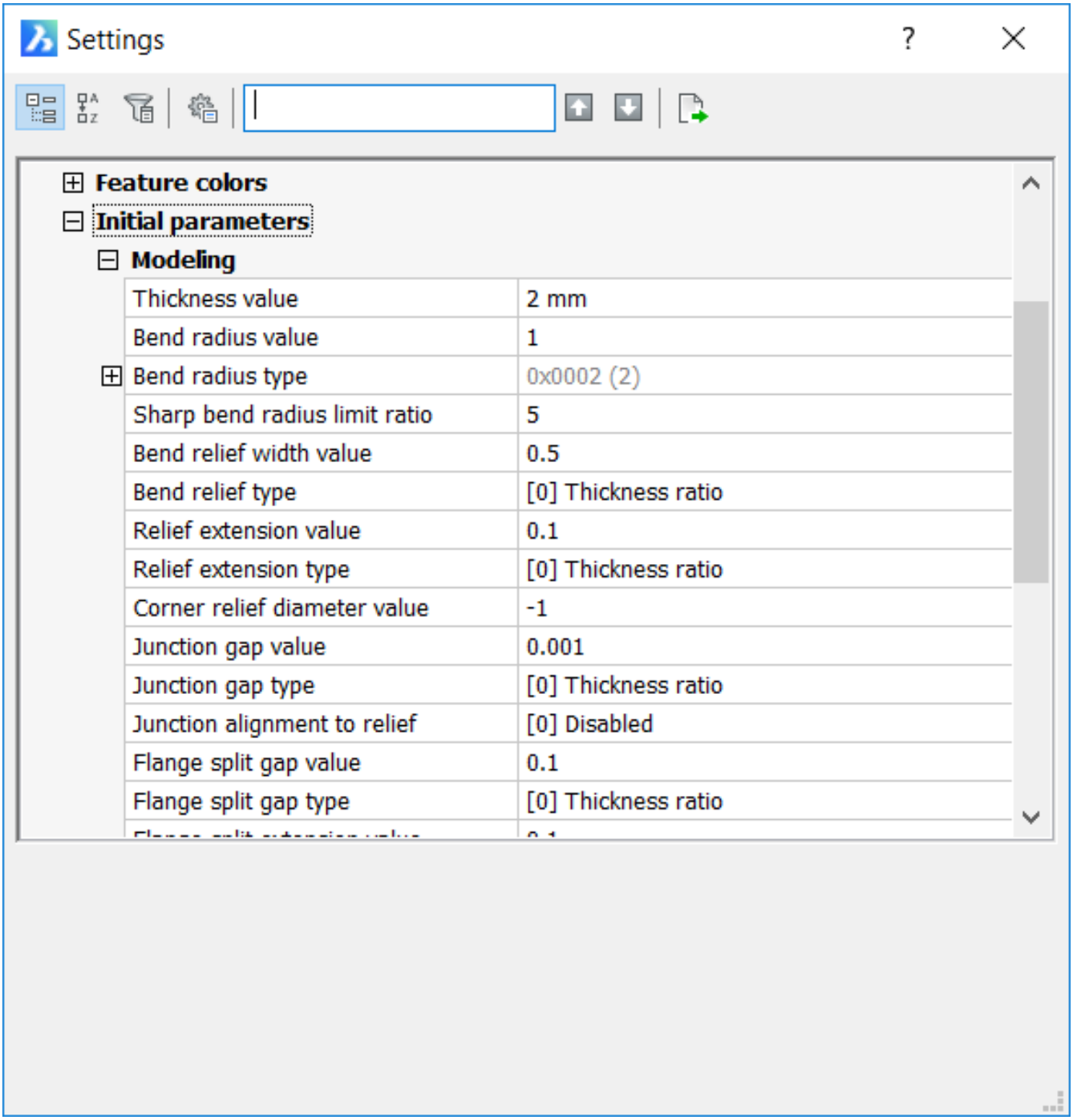
Figure 3.339


Settings
Compare
Sheet Metal
Commands
Feature colors
Feature colors
Toggle feature colors
Flange feature color
(144,164,174)
Flange feature reference side color
(104,164,174)
Form feature color
(135,145,225)
Bend feature color
(255,220,80)
Lofted bend feature color
(160,220,250)
Wrong bend feature color
(255,51,0)
Bend relief feature color
(100,210,150)
Corner relief feature color
(100,210,150)
Junction feature color
(255,110,64)
Miter feature color
(175,70,216)
Initial parameters
Modeling

Initial Parameters

These parameters are responsible for initialization of **Sheet Metal Context** associated with the drawing. SeeThe Mechanical Browser for Sheet Metal (3.8.5)for more details. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS

Figure 3.340


Settings
?
Feature colors
Initial parameters
Modeling
Thickness value
2 mm
Bend radius value
1
Bend radius type
0x0002 (2)
Sharp bend radius limit ratio
5
Bend relief width value
0.5
Bend relief type
[0] Thickness ratio
Relief extension value
0.1
Relief extension type
[0] Thickness ratio
Corner relief diameter value
-1
Junction gap value
0.001
Junction gap type
[0] Thickness ratio
Junction alignment to relief
[0] Disabled
Flange split gap value
0.1
Flange split gap type
[0] Thickness ratio

Modeling

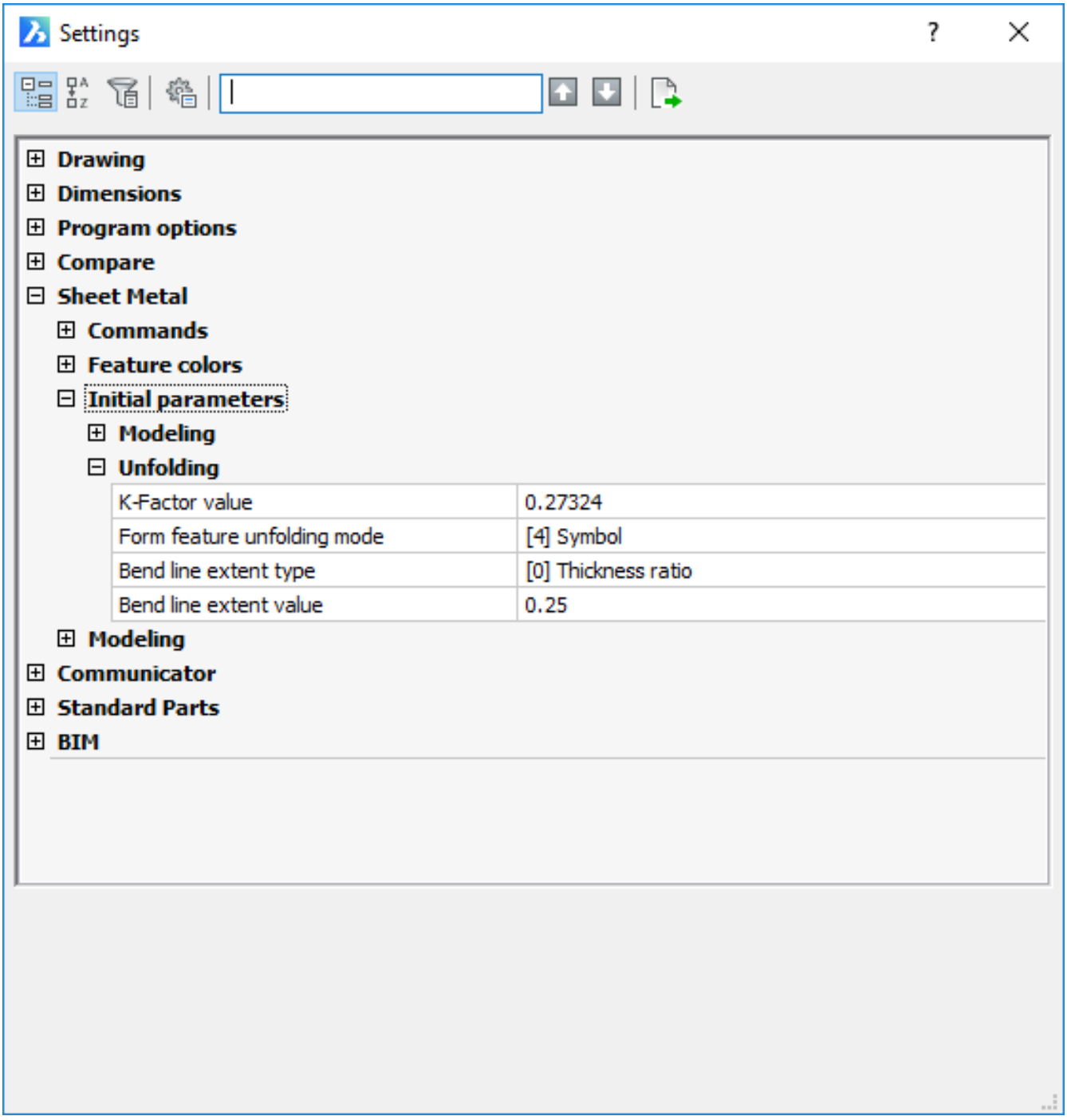
Modeling settings controls the behavior of Bends, Junction Gap, Flange, Rib and Corner relief.

Unfolding

Unfolding settings controls the behavior of K-Factor, Form feature mode and Bend extent type and value. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmSettingsUnfolding.png})

Tutorial: Parametrizing sheet metal parts Click here (3.7.22) to watch

Figure 3.341


Settings
Drawing
Dimensions
Program options
Compare
Sheet Metal
Commands
Feature colors
Initial parameters
Modeling
Unfolding
K-Factor value
0.27324
Form feature unfolding mode
[4] Symbol
Bend line extent type
[0] Thickness ratio
Bend line extent value
0.25
Modeling
Communicator
Standard Parts
BIM

3.8.10 Creating Flanges

A flange is the main feature of most sheet metal parts. In BricsCAD you can create flanges using many different techniques:

- Create a base flange from a closed 2D profile, which defines its boundary.
- Create one or several edge flanges by pulling edges of existing flanges.
- Create several flanges at once by extruding a 2D polyline (every linear edge of the polyline becomes a flange).

These techniques are described in the article below, but there are also other techniques of creating flanges described in other articles:

- Flanges can be created as elements oflofted sheet metal parts (3.8.12).
- Flanges can be recognized (3.8.14) in existing geometry.

In contrast to history-based MCAD, in BricsCAD applies the same set of tools to modify (3.8.20) any flange - independently on the particular way of its creation.

Creating a Base Flange

In most cases you start to design your sheet metal part with creating its base flange. To create a base flange, select a closed planar profile and call the SmFlangeBase (6.28.10) command. Do one of the following:

- Click the **Create Base Flange** tool button () on the **Sheet Metal** toolbar.
- Choose **Create Base Flange** in the **Sheet Metal** menu.
- Choose **Create Base Flange** in the **Sheet Metal** group of the Quad (2.2.3) cursor menu:

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03Drawir
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03Drawir
A base flange is a body created by extruding the selected profile to a height equal to the default value of the **Thickness** property of the sheet metal part. To change the thickness of your sheet metal part, type the appropriate value in the **Thickness** field in the Mechanical Browser (3.2.2).

Figure 3.342

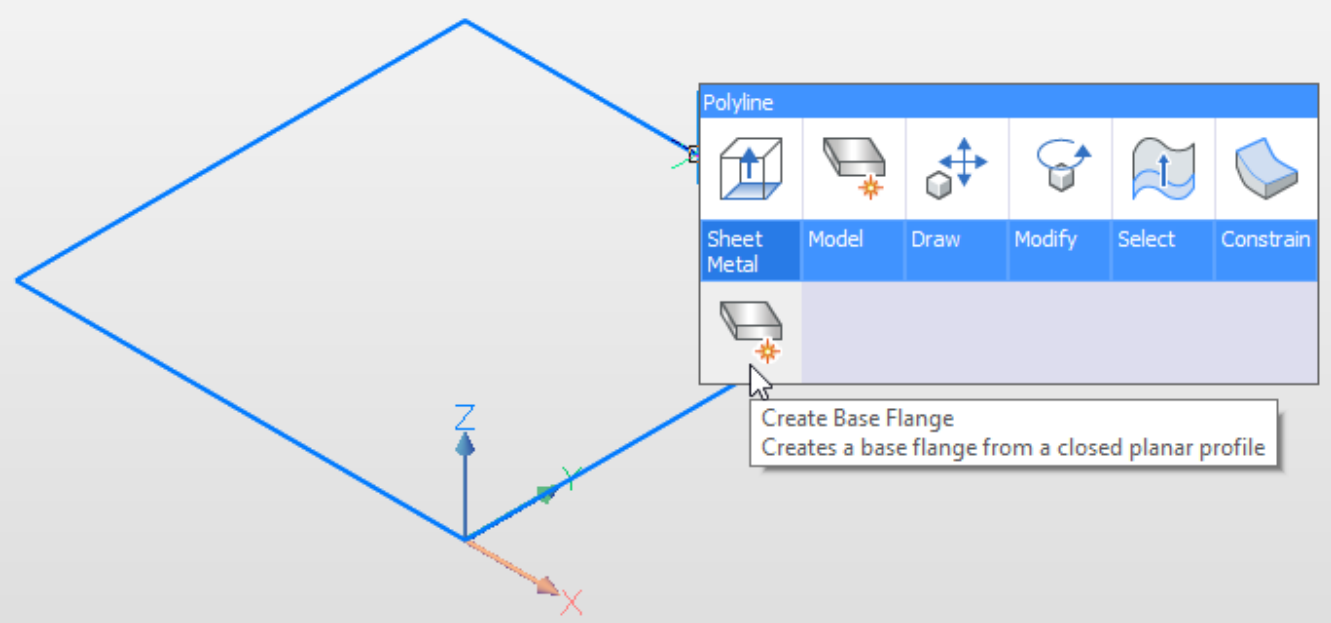
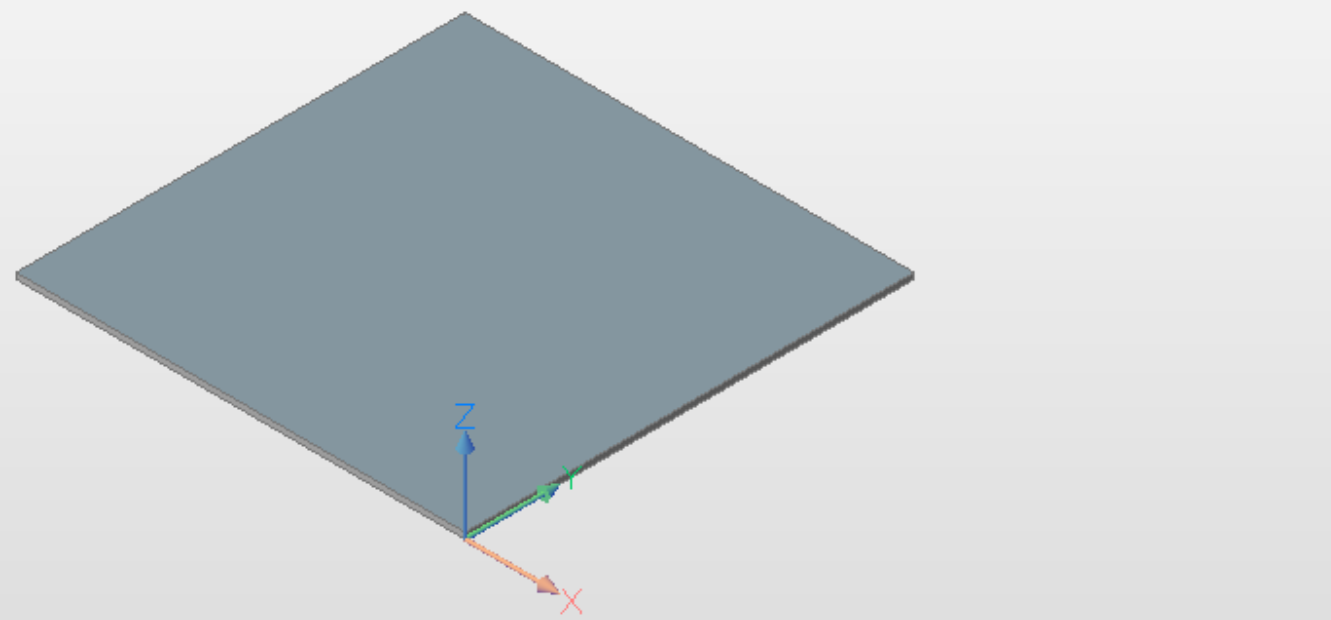


Figure 3.343



Creating Edge Flanges

You can add an additional flange to your sheet metal part by pulling a linear edge of an existing flange. Select an edge and select the SmFlangeEdge (6.28.13) command in the **Sheet Metal** section of the Quad (2.2.3) cursor menu. Move your mouse pointer to define the desired length of the edge flange and the angle between two flanges. You can also use the corresponding dynamic dimensions to type the desired values. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsysc
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03Drawir
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03Drawir
To create an edge flange of which the width is different from the width of the edge, select the **Width** option of SmFlangeEdge (6.28.13) command and define offsets from both sides of the edge. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/SmStep6
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03Drawir
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03Drawir
Note that when you create an edge flange of different width, the corresponding bend reliefs are created automatically.

Figure 3.344

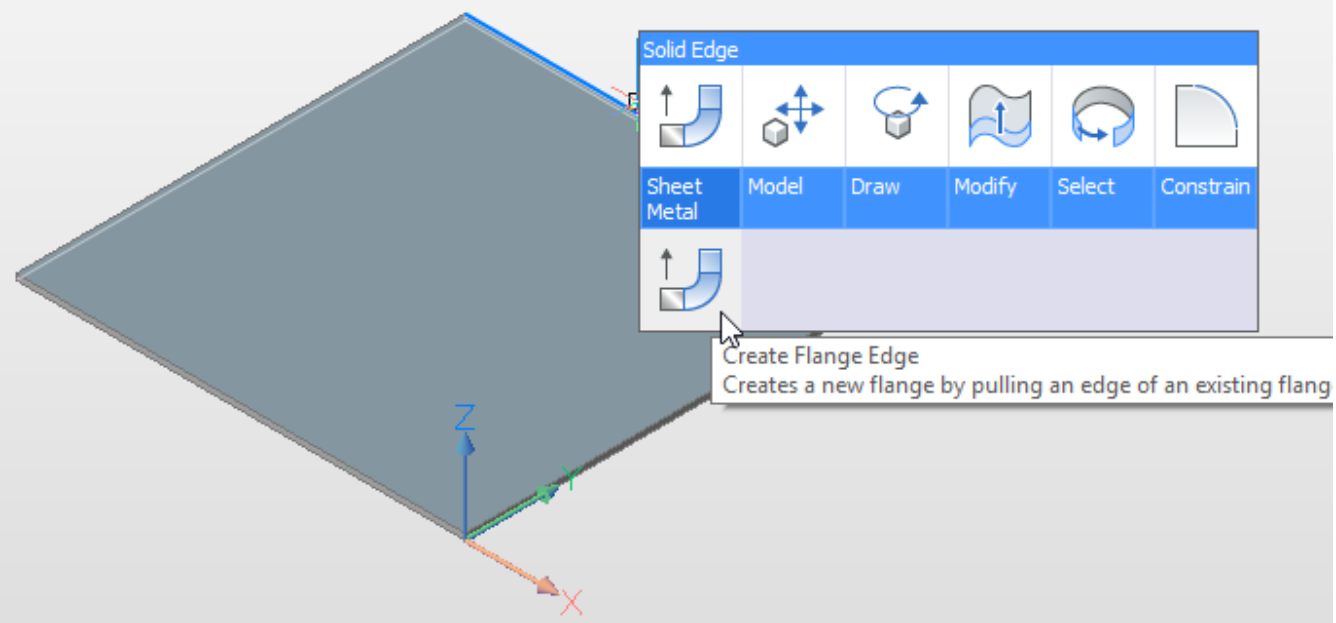


Figure 3.345

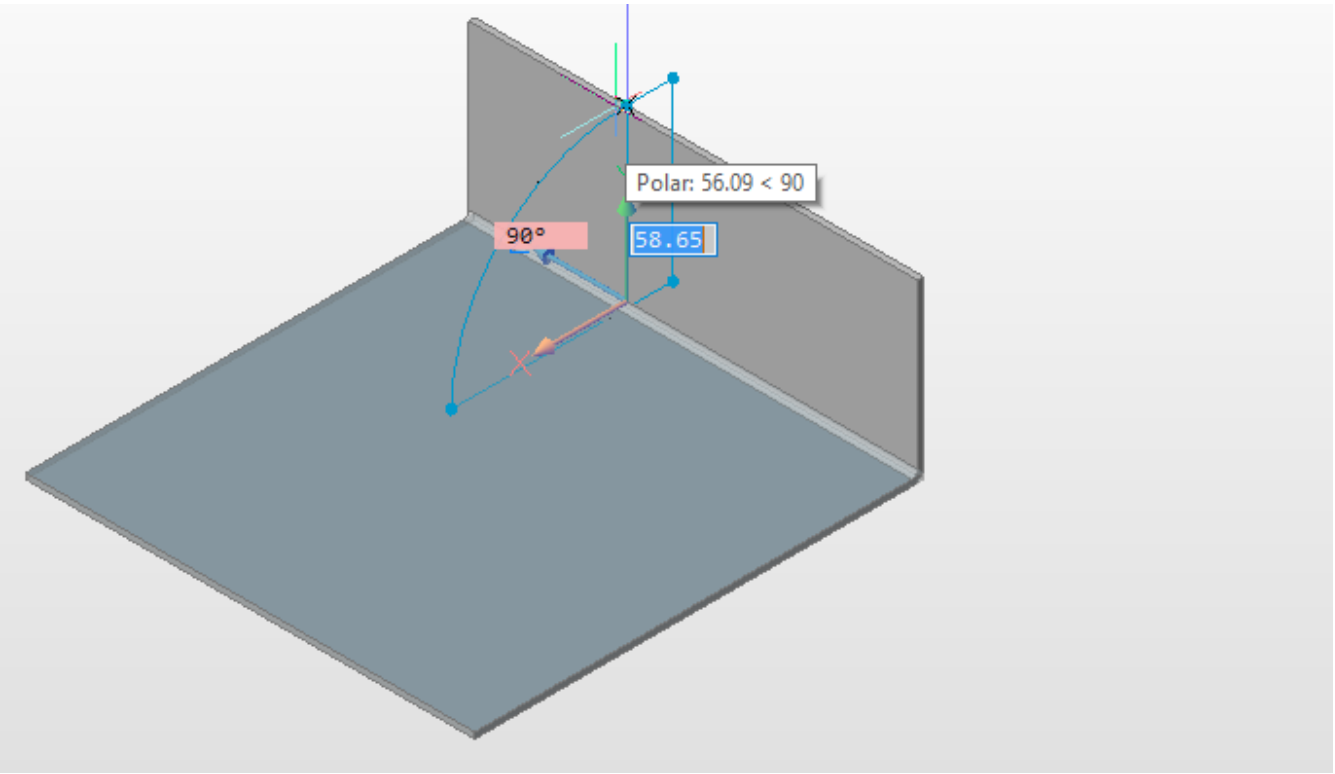


Figure 3.346

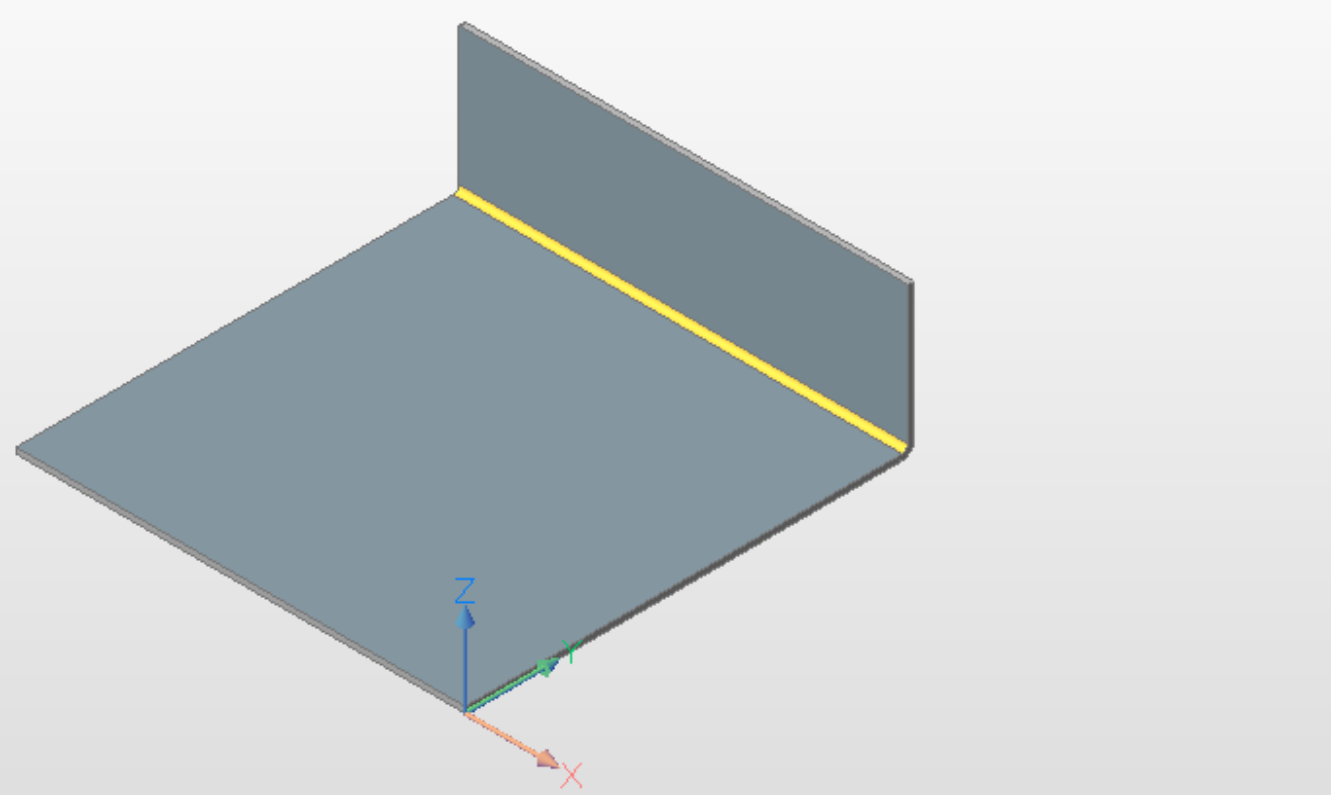


Figure 3.347

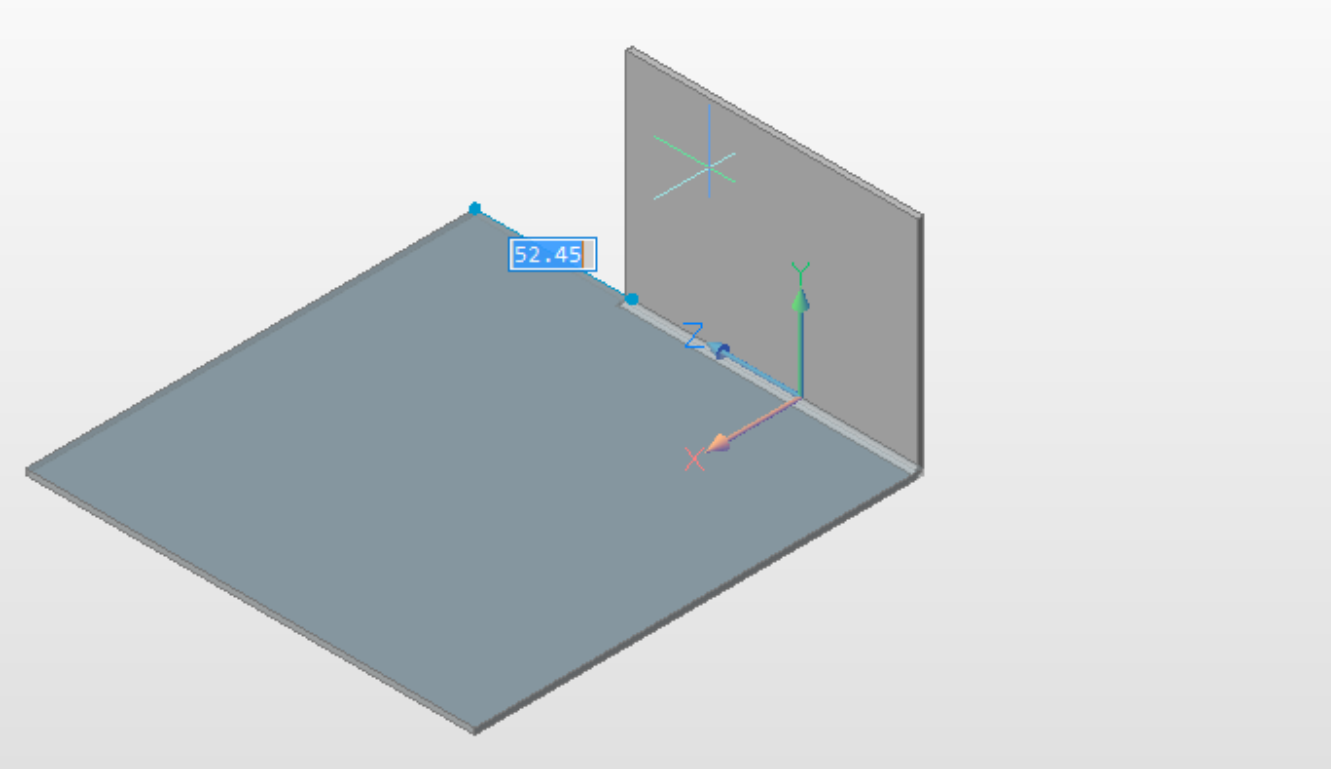


Figure 3.348

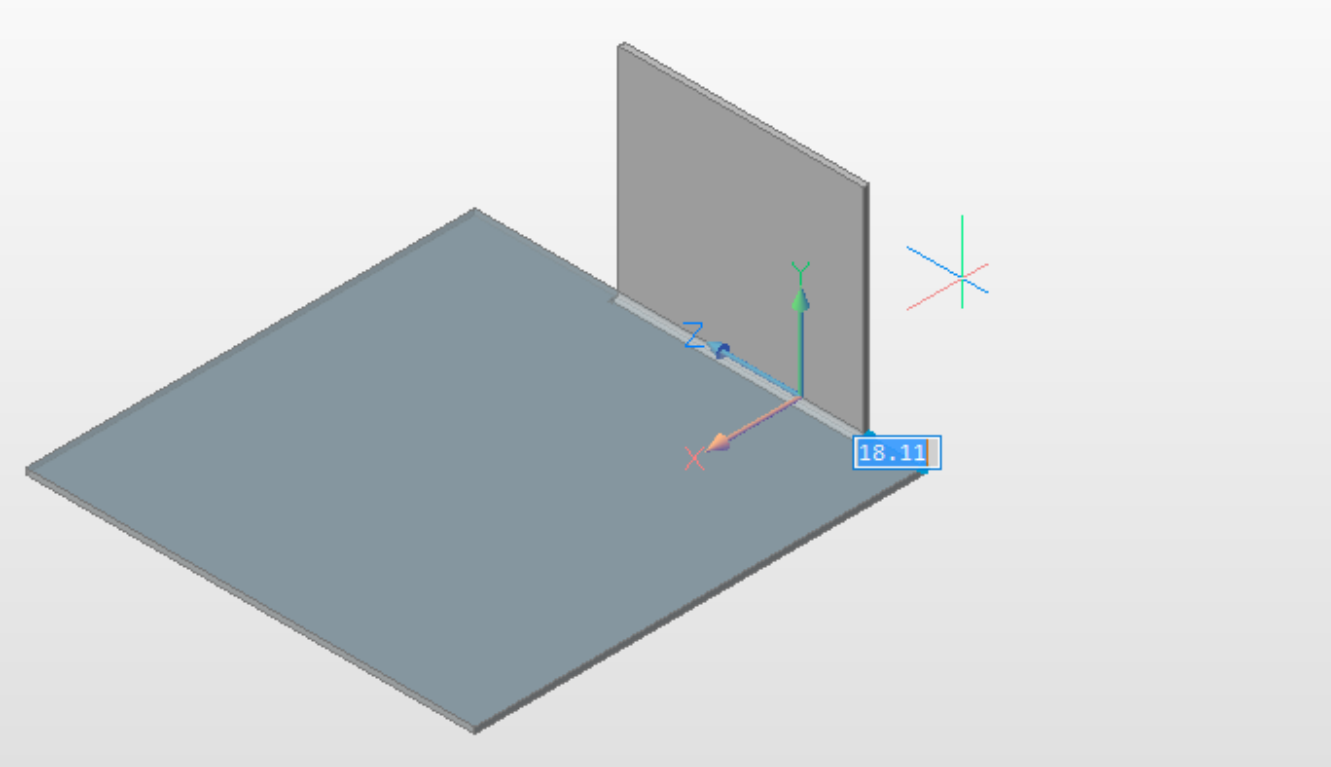
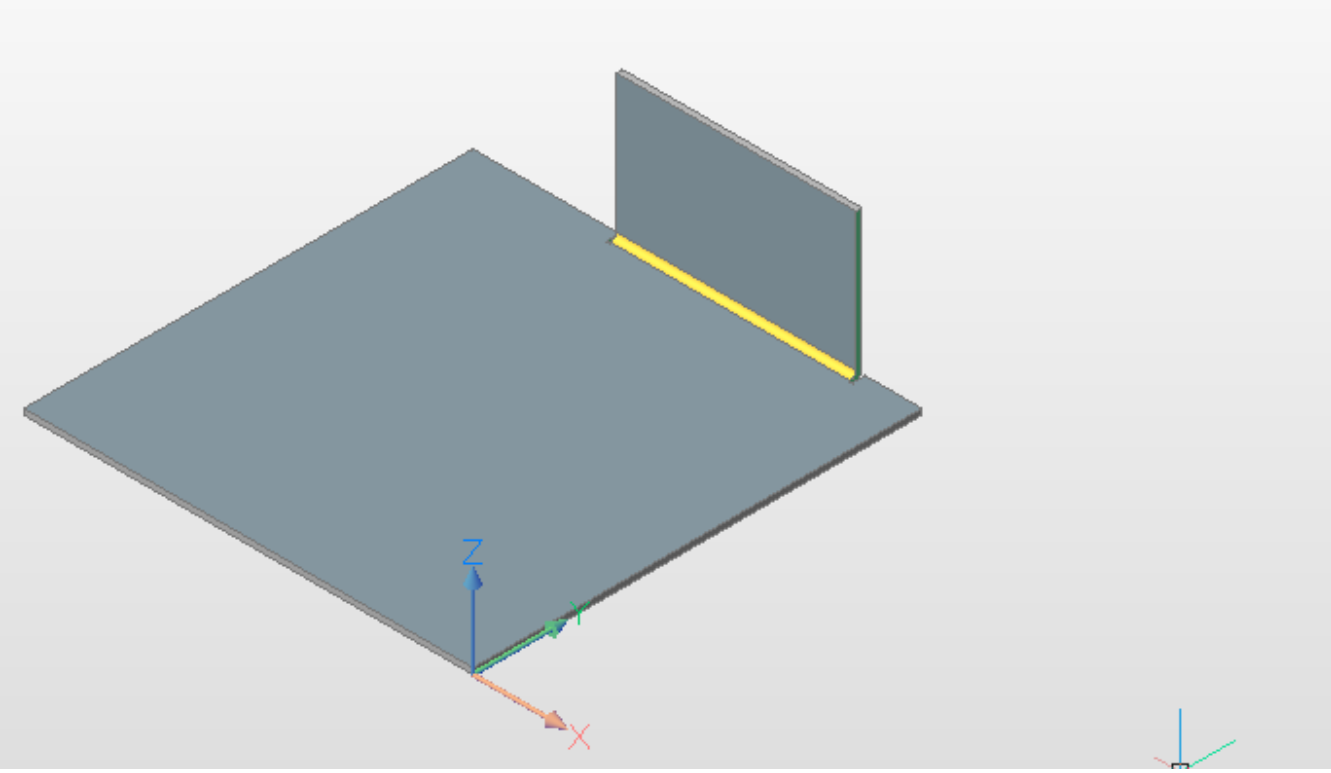


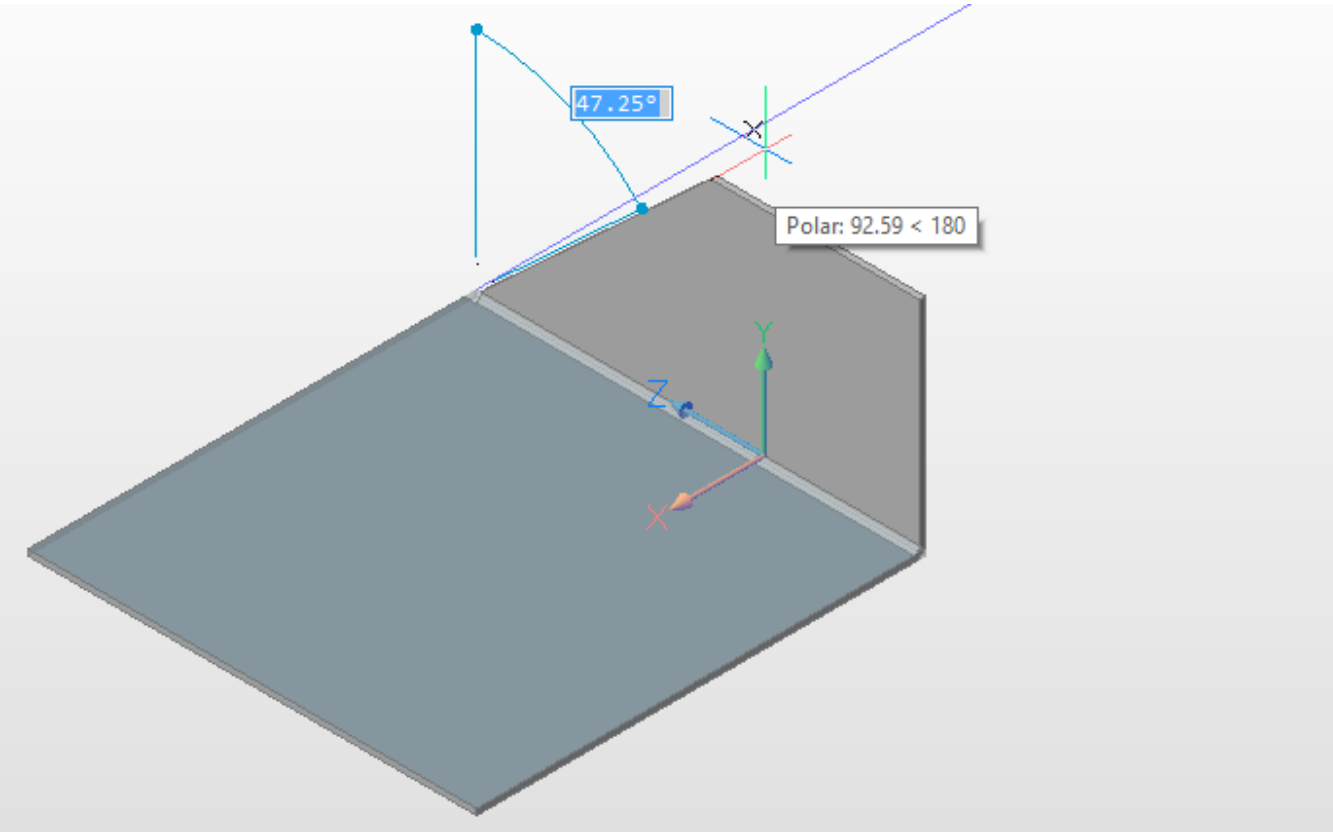
Figure 3.349



Create Edge Flange With Taper Angle(s)

You can create trapezoidal edge flanges using **Taper angle** option of SmFlangeEdge (6.28.13) command: (\vref
{fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolid
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03Drawin

Figure 3.350



(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03Dra

Figure 3.351

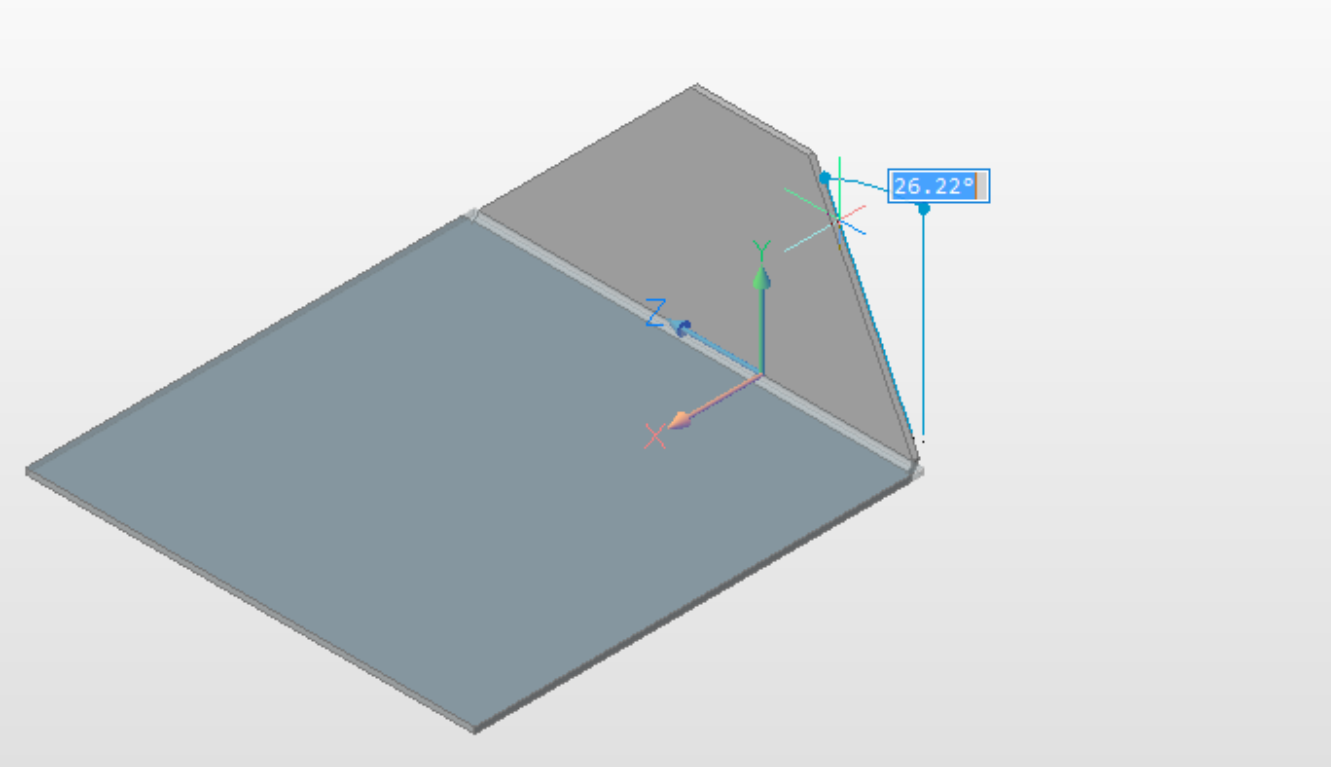
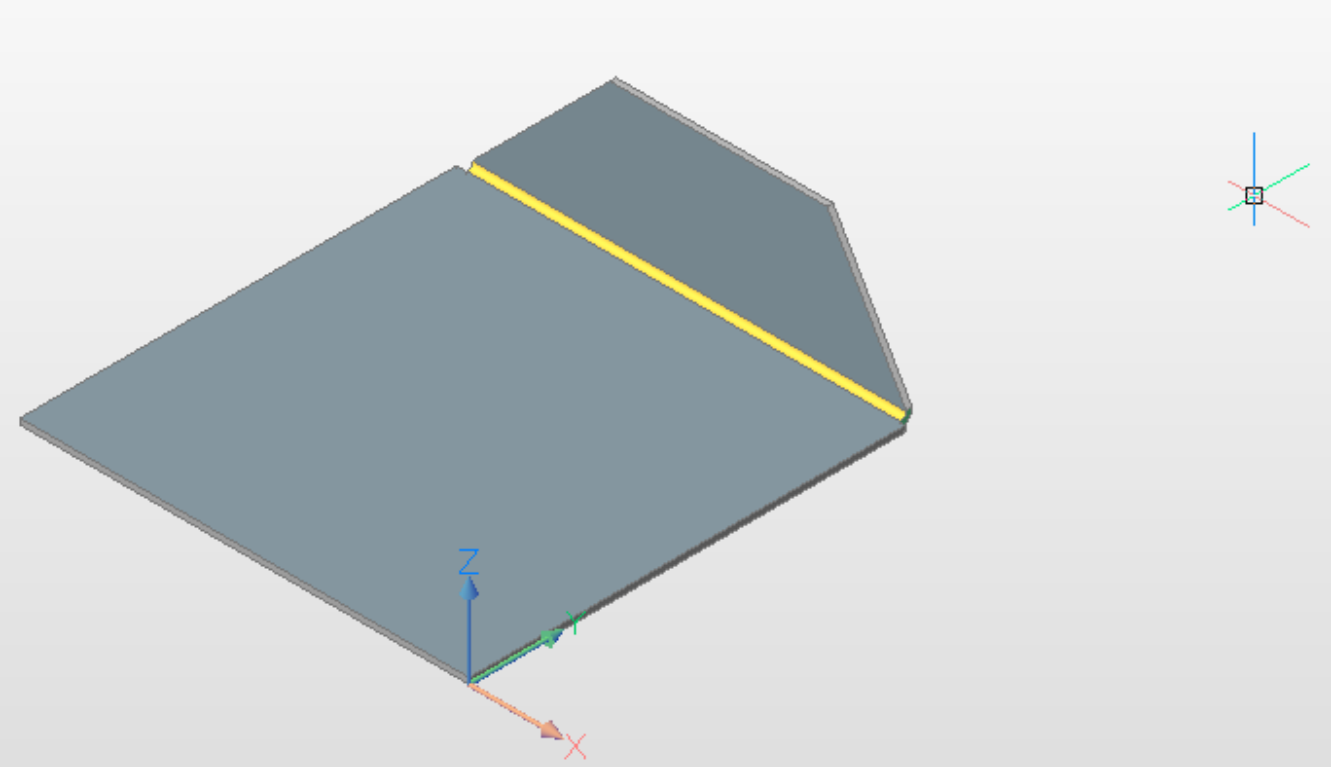
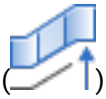


Figure 3.352



Creating Multiple Flanges by Extruding a Polyline

There is a simple but important class of sheet metal parts which are seen as thickened polyline if we view on the part from its ”front” side: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/Creating
Such part can be built with sequential calls ofSmFlangeEdge (6.28.13) command, or user can draw polyine,DmExtrude (6.12.13) it by required height, thenDmThicken (6.12.31),SmConvert (6.28.4)andSmBendCreate (6.28.2)on it. And yet the result may be far from desired. What if you have to draw your part in complex of assembly with other parts, so they shouldn’t interfere? (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/Cre
The problem can be solved if you can control thickening direction of every line segment individually: one should understand if segment is ”external” for the part implied by polyline and inflate it into the interior. That is exactly whatSmExtrude (6.28.9) does. Draw (6.24.27) the polyline which represent a sketch of future sheet metal part. One given example polyline is drawn along the edges of existing parts: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/
Do one of the following:

- Click the **Extrude Polyline** tool button () on the **Sheet Metal** toolbar.
- Choose **Extrude Polyline**in the **Sheet Metal** menu.
- Choose **Extrude Polyline**in the **Sheet Metal** group of the Quad (2.2.3) cursor menu:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/CreatingFlangesCreatingMu
Hover mouse to select extrusion height or enter the value and press Enter to accept: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/CreatingFlangesCreatingMultipleFlangesbyExtrudi
The body is created. The extensions are respected: build body doesn’t exceed the dimensions of original polyline, same it behaves on thickness change:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	(\vref {fig:C:/Dev/Task-	(\vref {fig:C:/Dev/Task-
Thickness 1 mm	Thickness 2 mm	Thickness 4 mm

Note that policy of preserving external dimensions also works for sheet metal parts created by other workflows: via sequntial flange building bySmFlangeEdge (6.28.13), part bending by SmFlangeBend (6.28.11) and sheet metal design recognition bySmConvert (6.28.4). If you want to change extrusion direction for a given flange, you should applySmFlip (6.28.15)command to it.

Figure 3.353

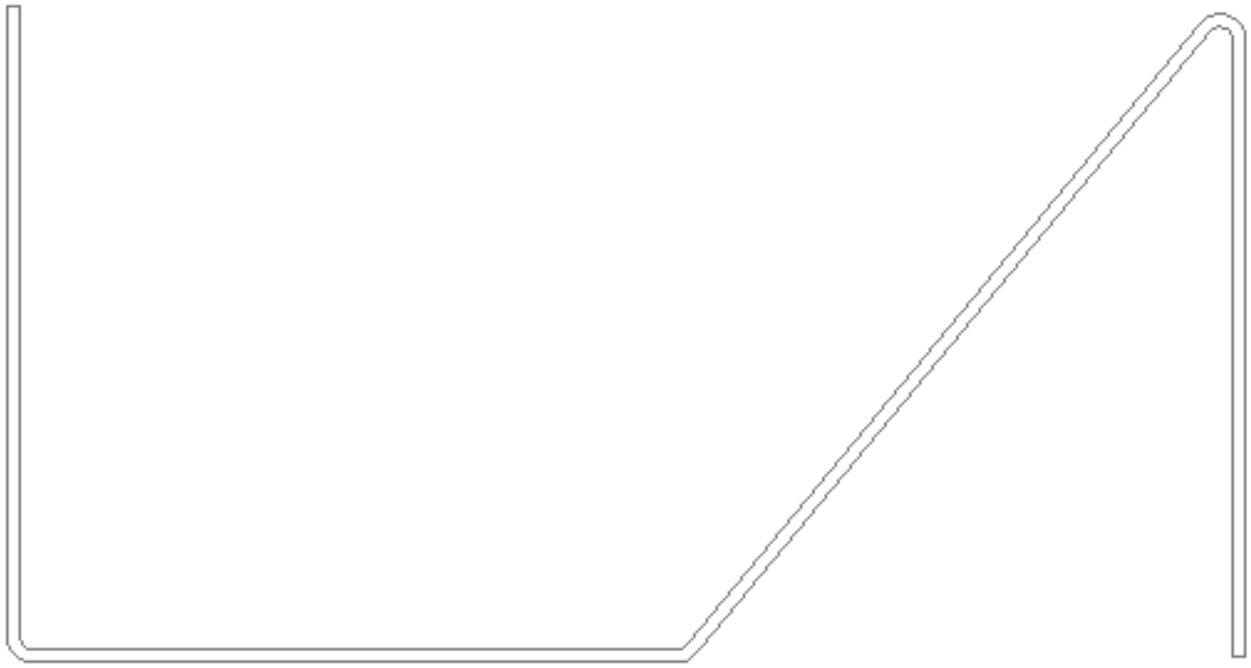


Figure 3.354

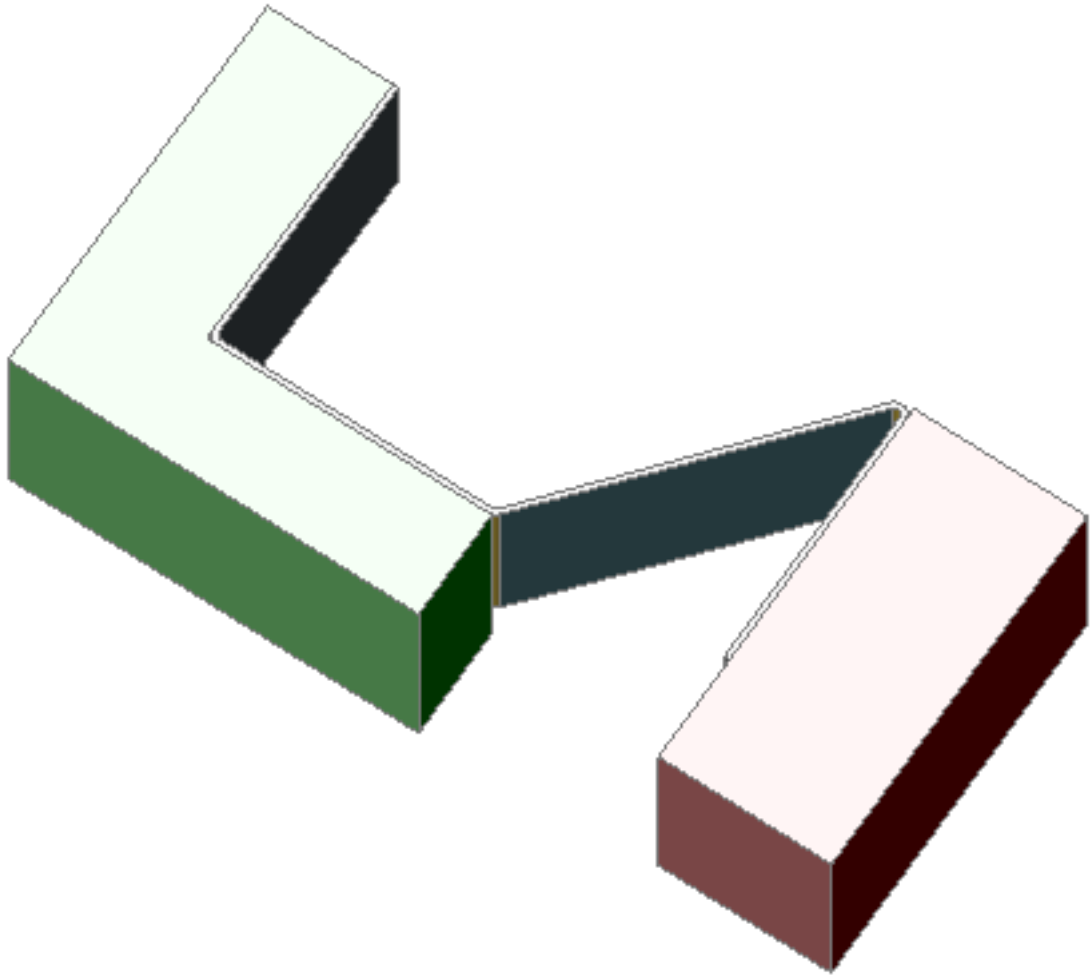


Figure 3.355

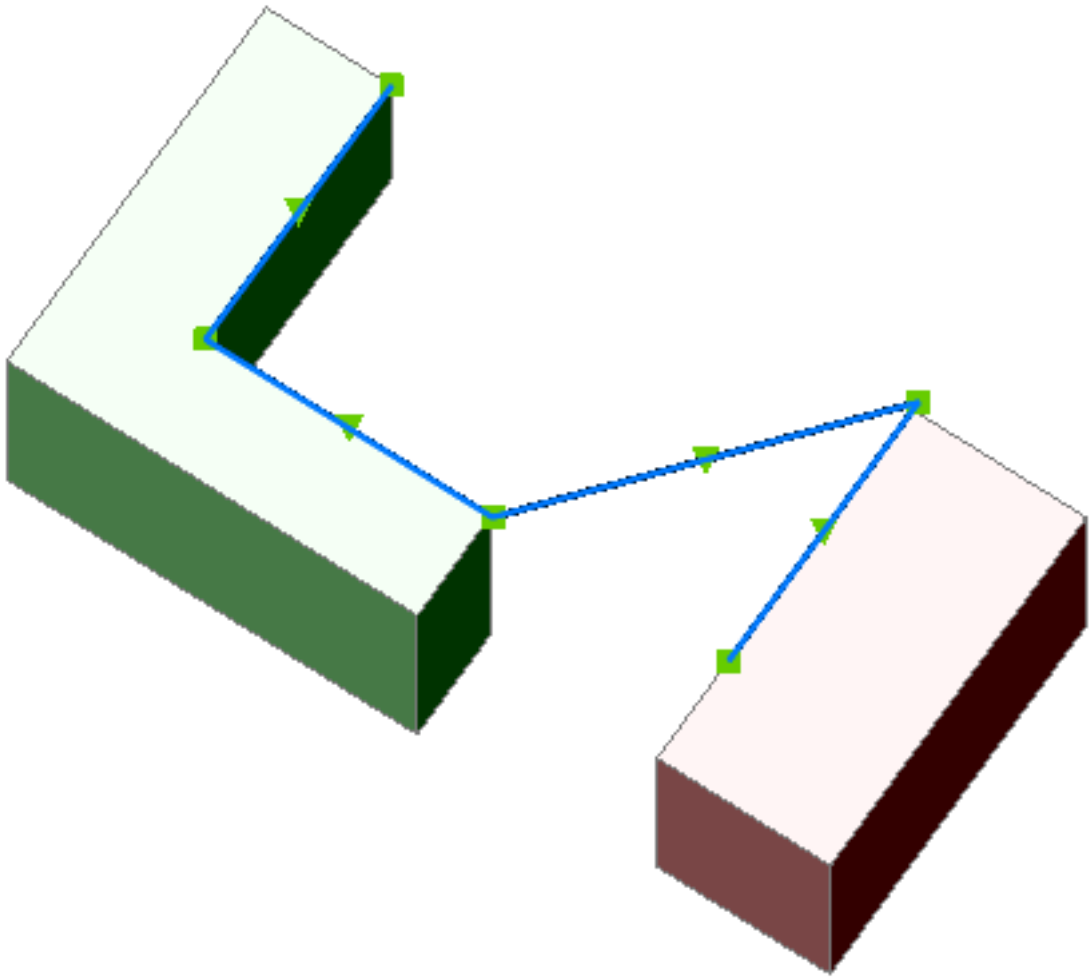


Figure 3.356

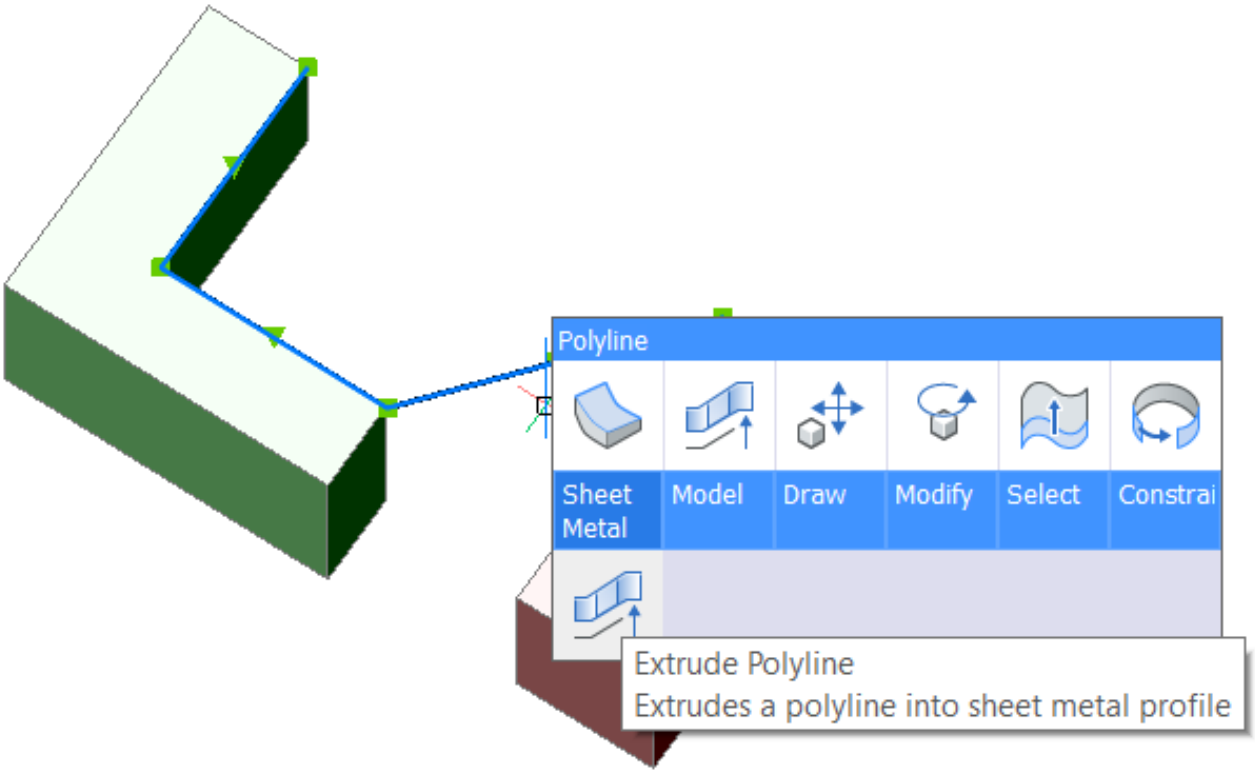
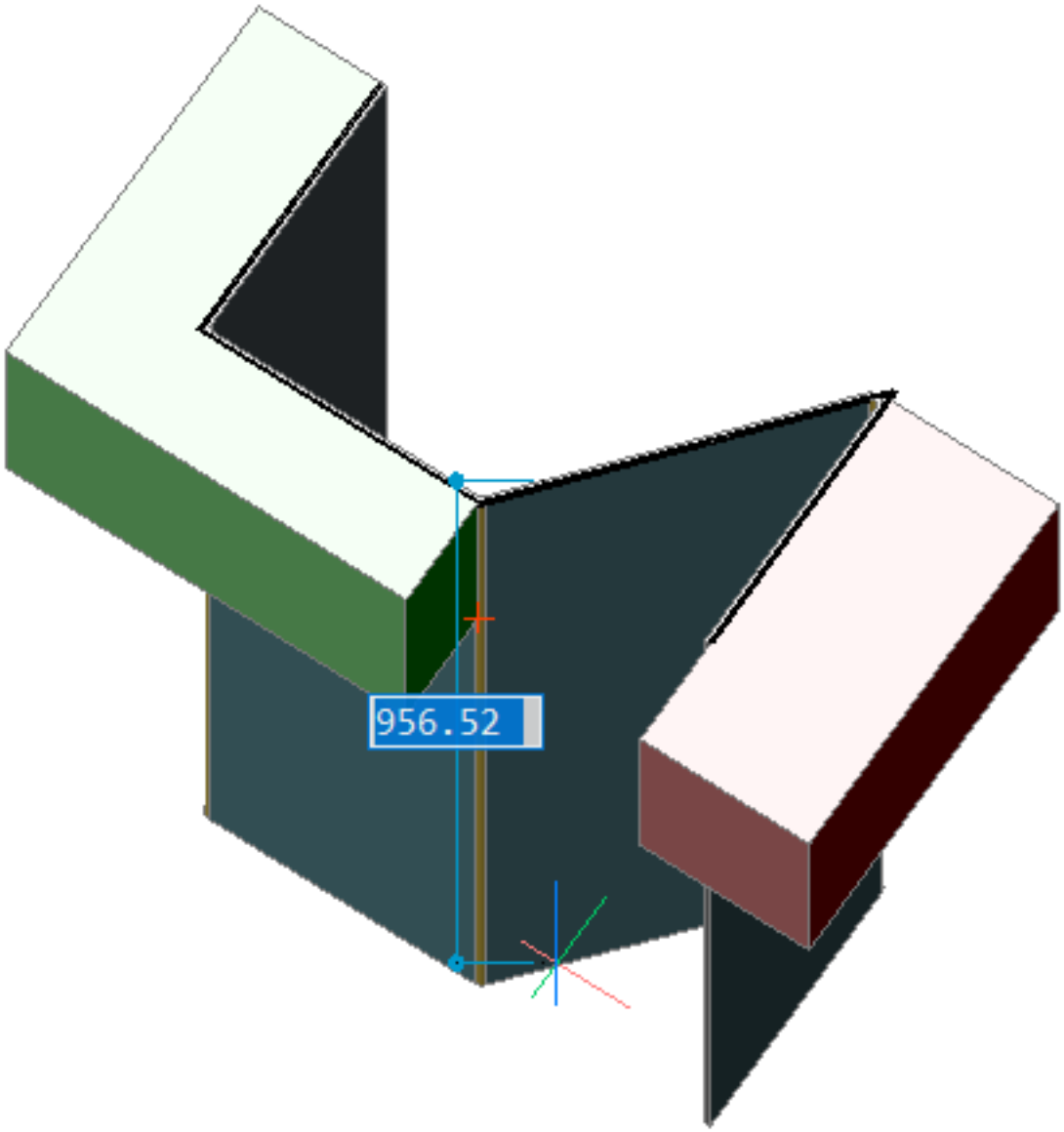

Polyline
Sheet Metal
Model
Draw
Modify
Select
Constrai
Extrude Polyline
Extrudes a polyline into sheet metal profile

Figure 3.357

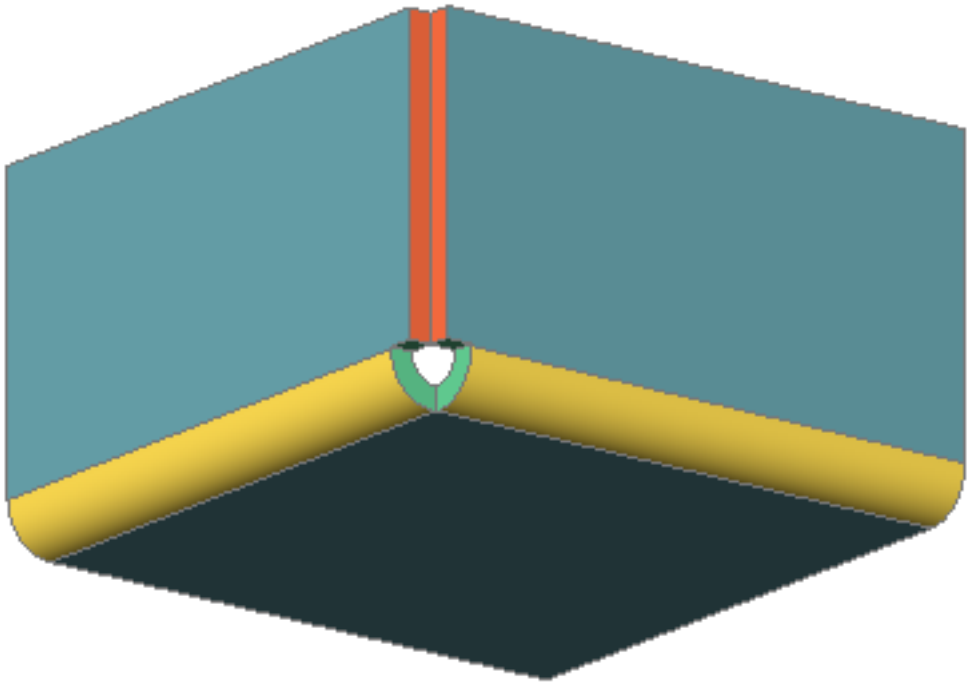


Bends, Corner Reliefs and Junctions

When you pull an edge flange with SmFlangeEdge (6.28.13) command, a bend feature between the edge flange and the base flange is created automatically. If the pulled edge isadjacent to a bend edge, a corner relief feature is created automatically in the corner where three flanges meet together.This operation also creates a junction between two flanges, which are not connected with a bend. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Me
Please read 'Working with Bends and Junctions (3.8.11)' to learn more about these features.

Tutorial: Designing from scratch in BricsCAD Sheet Metal Click here (3.7.8) to watch

Figure 3.358



Creating Multiple Flanges

You can create several flanges simultaneously. Do one of the following:

- Select multiple edges, then choose **Create Flange Edge** ( ) in the **Sheet Metal**command group in the Quad (2.2.3).
- Launch the SmFlangeEdge (6.28.13) command and select multiple edges.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/UsrGui/source/10DrawingEntities/03Drawir
1.png})

- Create the edges dynamically. Define the angle and the flange height using the dynamic entry fields. Hit the TAB key to switch between the **Angle** and the **Distance** field. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/brics
1.png})
- Select the inside edges of the newly created flanges: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/he
2.png})

Figure 3.359

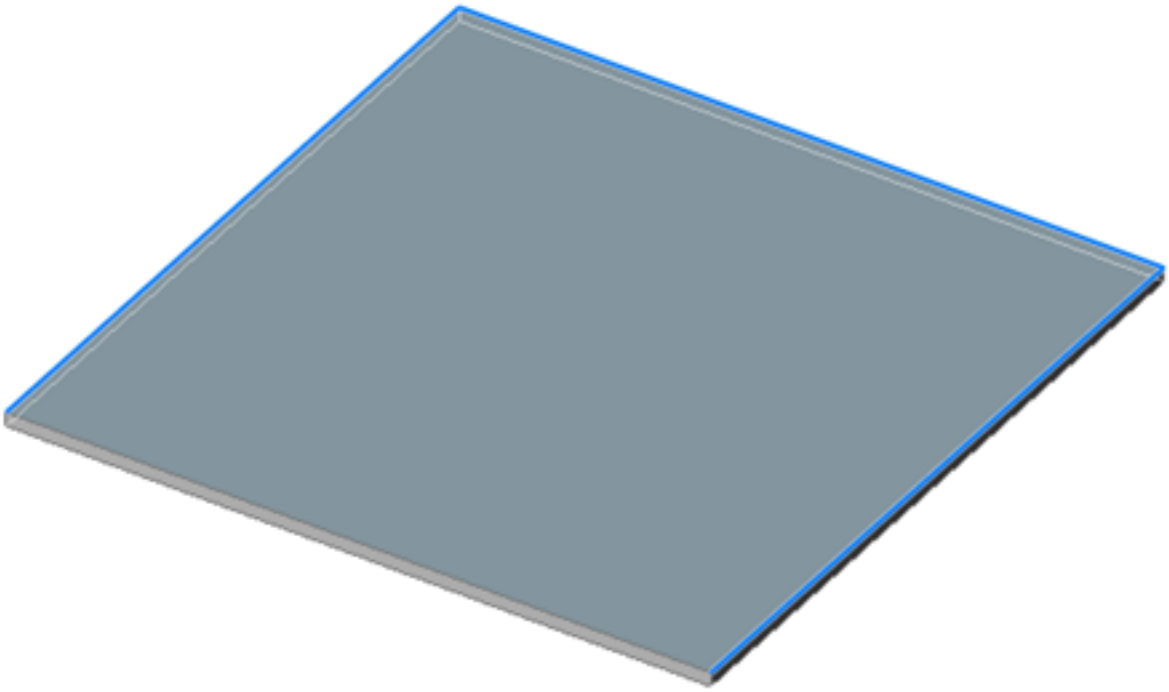
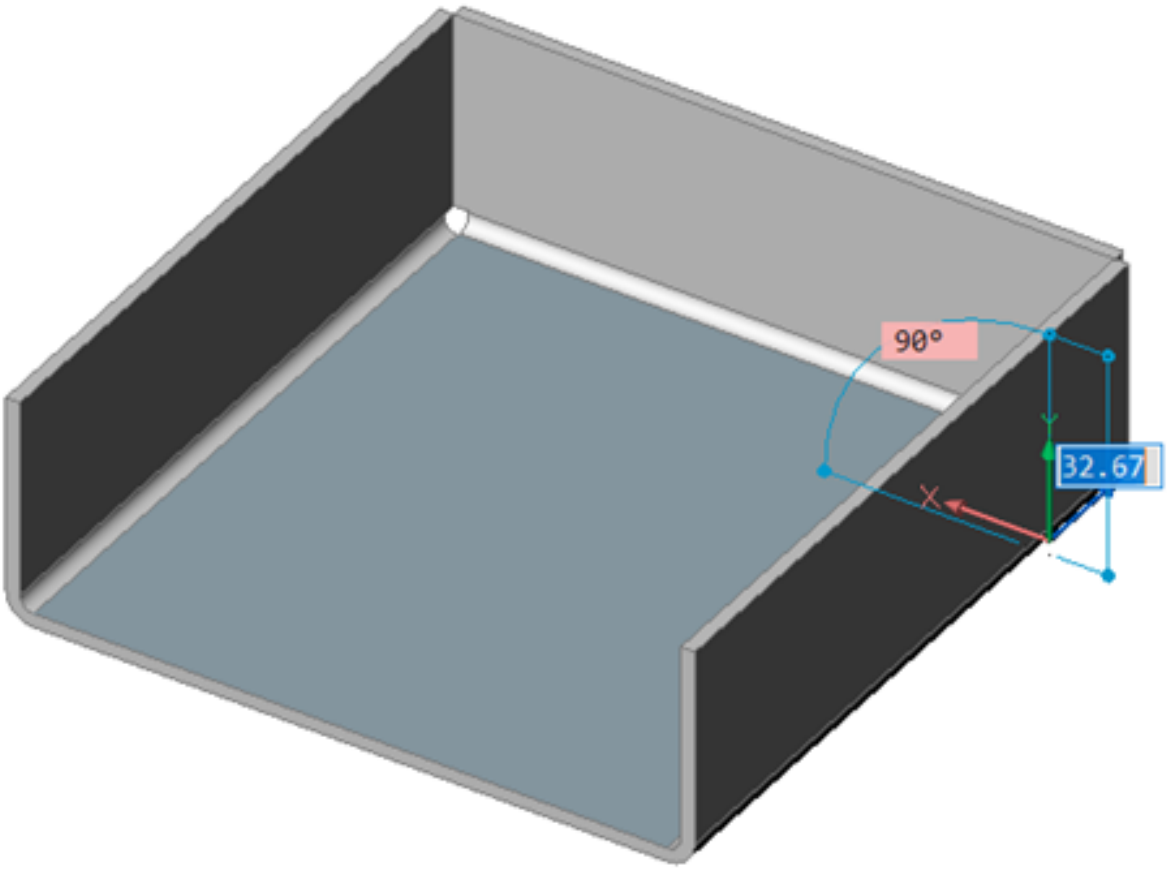


Figure 3.360



- Create the flanges. The command creates self-intersecting of the flanges and depending on the bend angle creates a junction or miter features. At 90 degrees miters are created: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsysc 2.png}) For other angles junctions are created: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enU 3.png}) In all cases material conflicts are avoided: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/ 4.png})

If the flanges are created externally, **Toggle** option (as well as widget control) helps to control connection type for miters: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/CreatingFlangesCrea (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/CreatingFlangesCreatingMu and junctions: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/CreatingFlang (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/CreatingFlangesCreatingMu

Figure 3.361

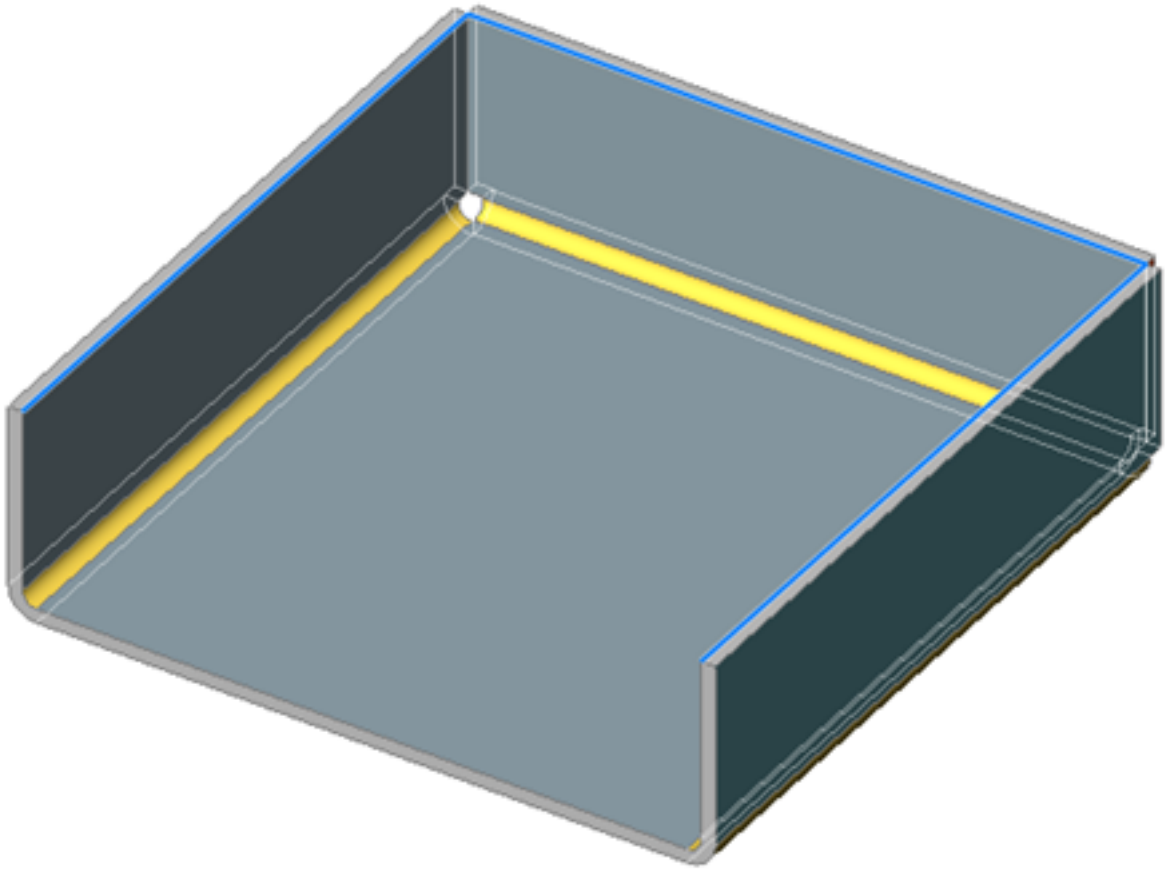


Figure 3.362

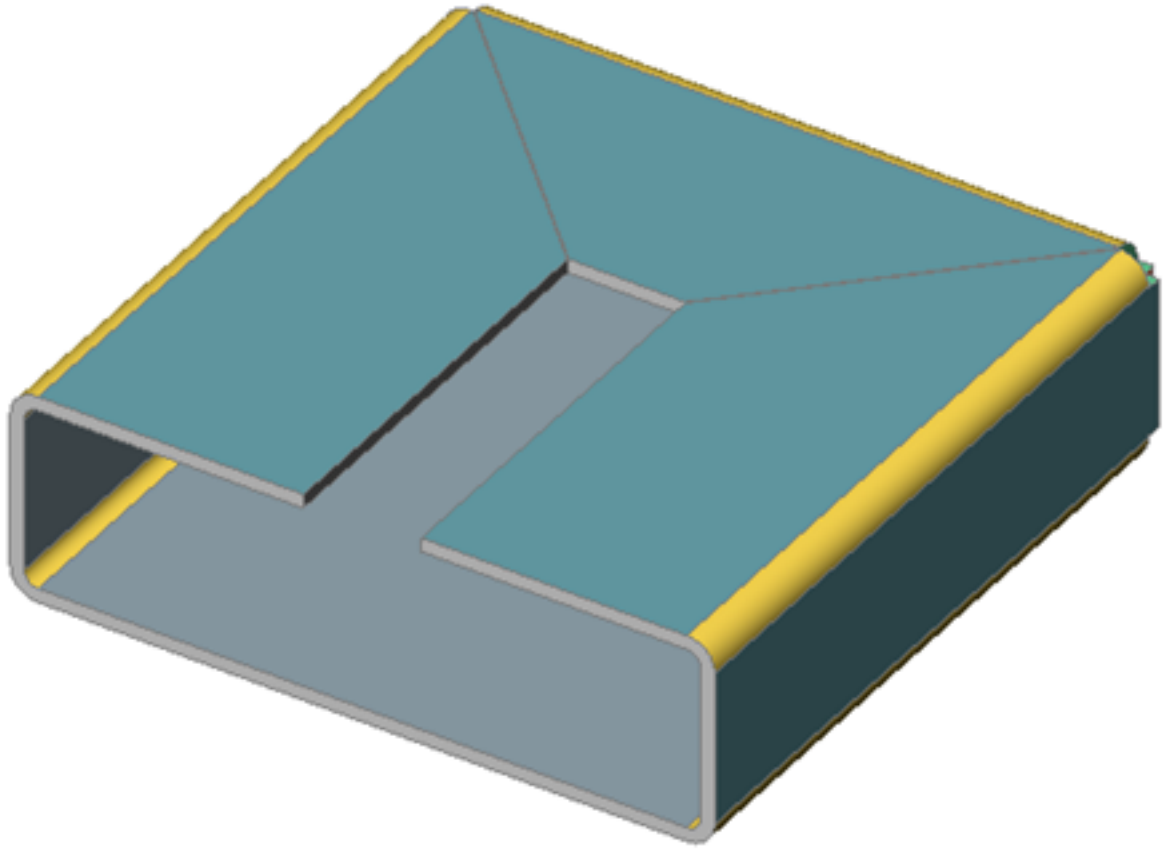


Figure 3.363

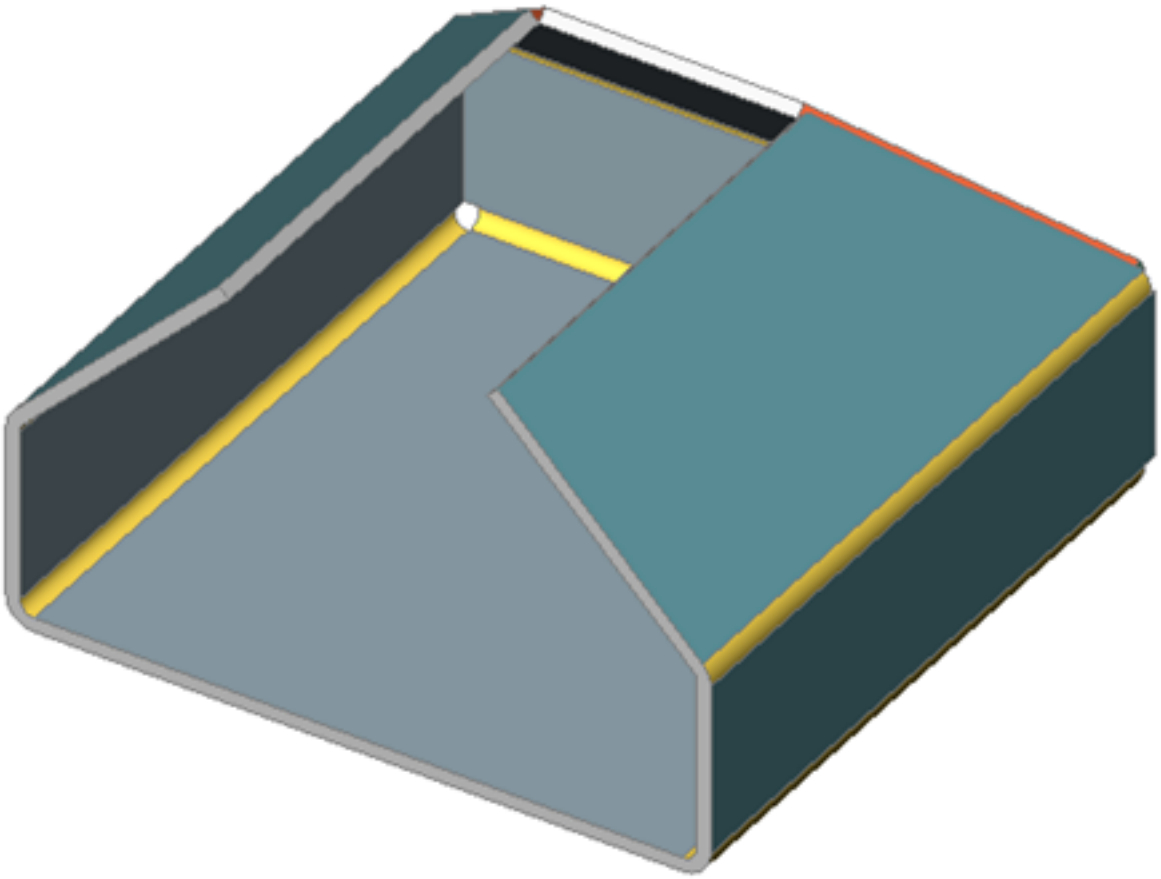


Figure 3.364

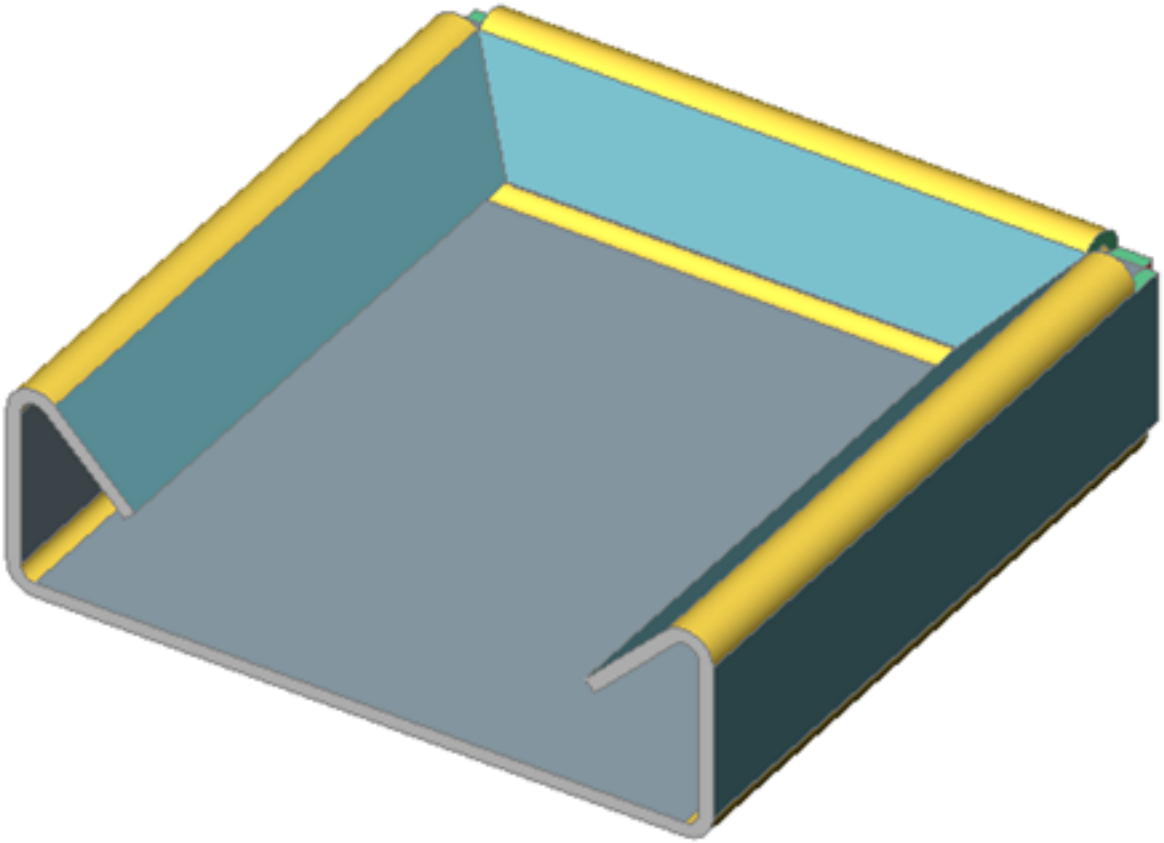


Figure 3.365

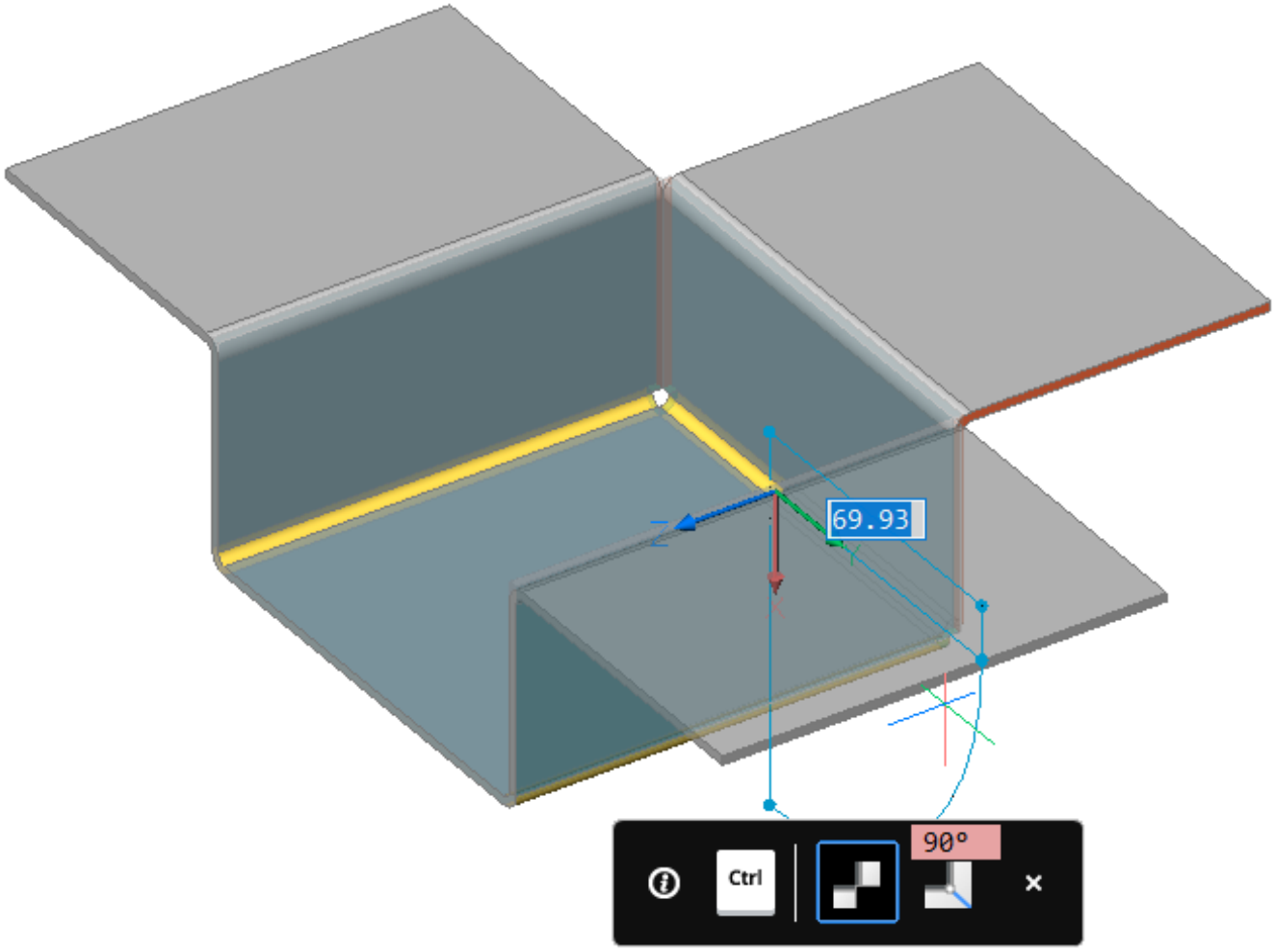


Figure 3.366

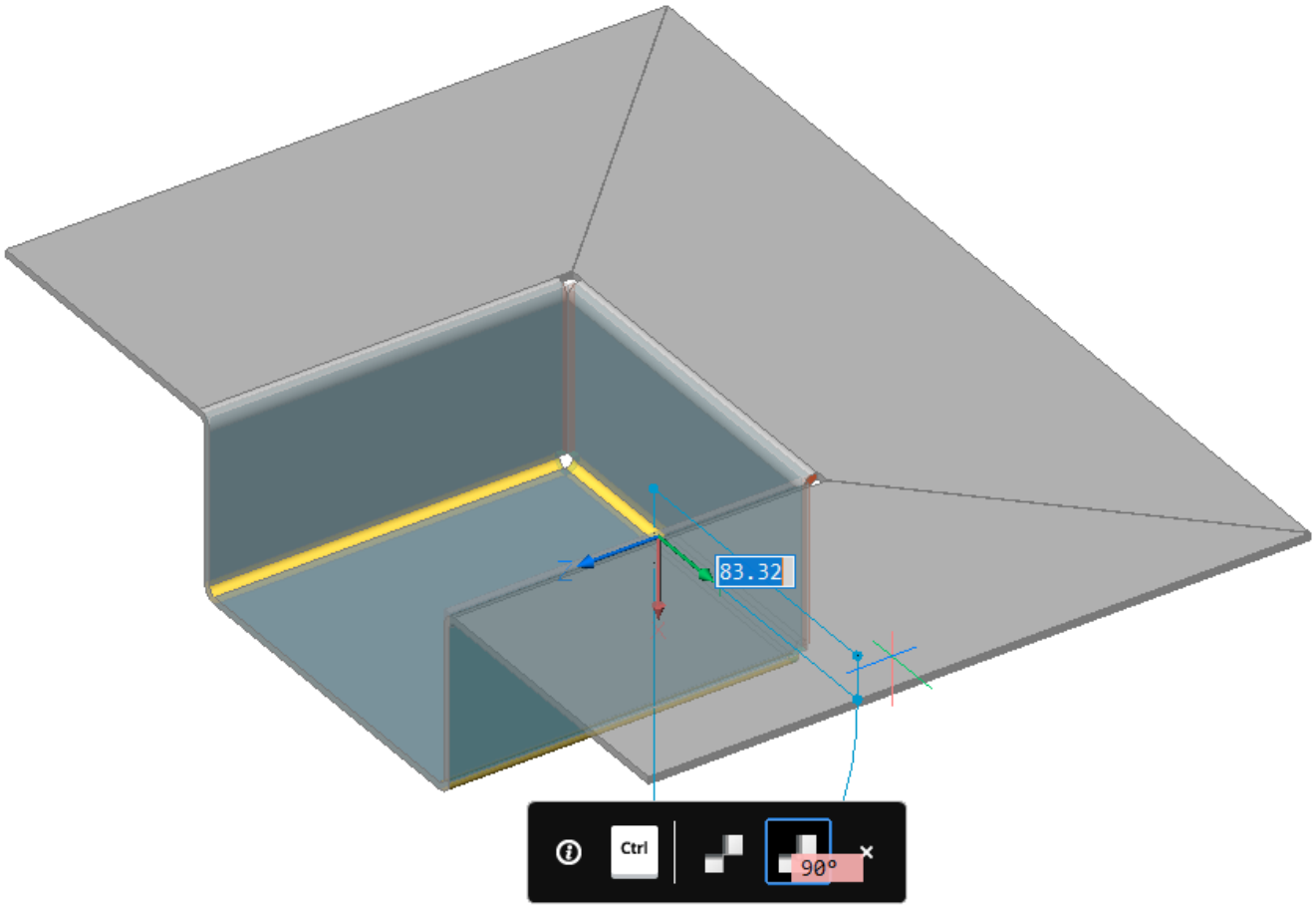


Figure 3.367

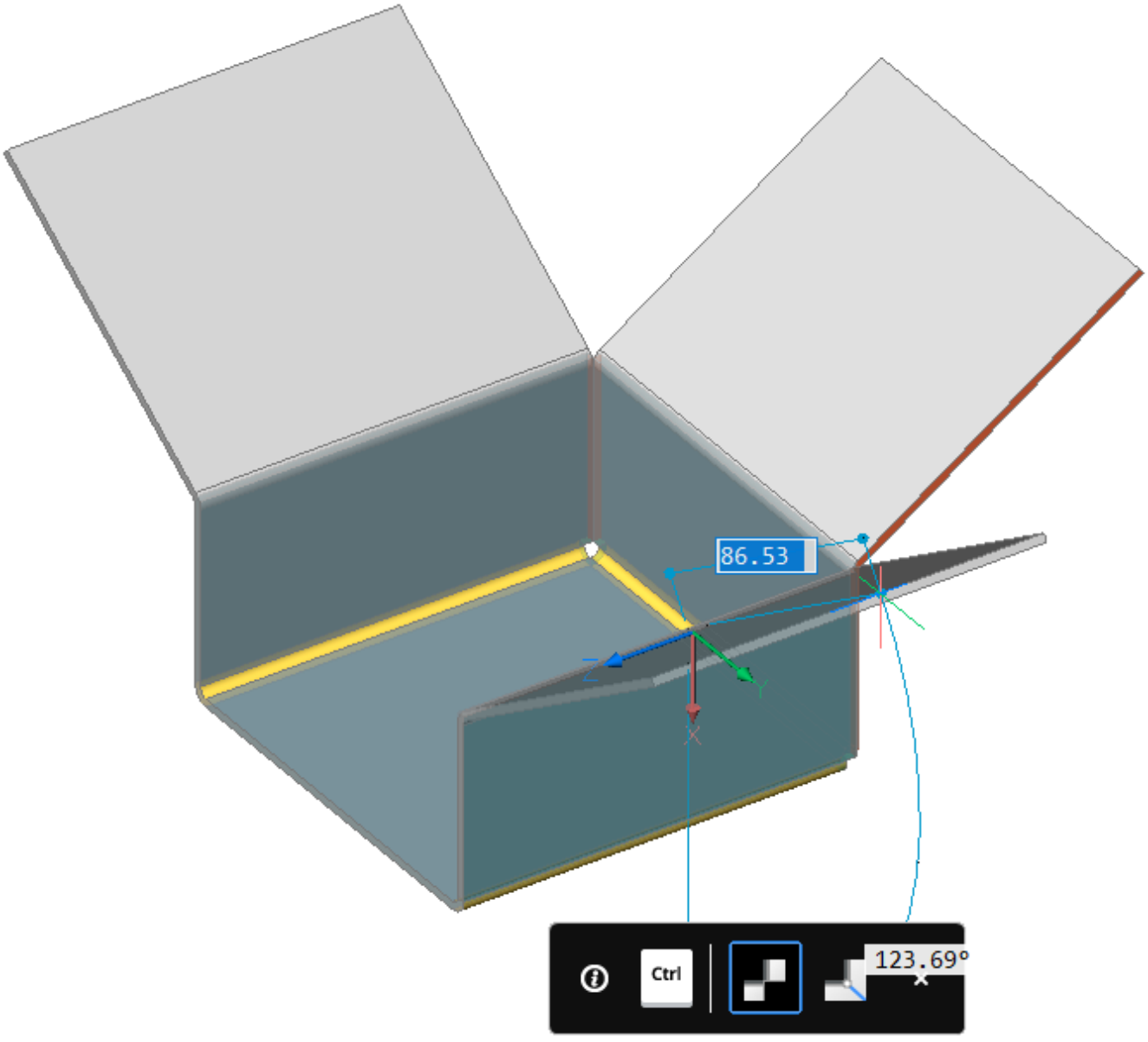


Figure 3.368

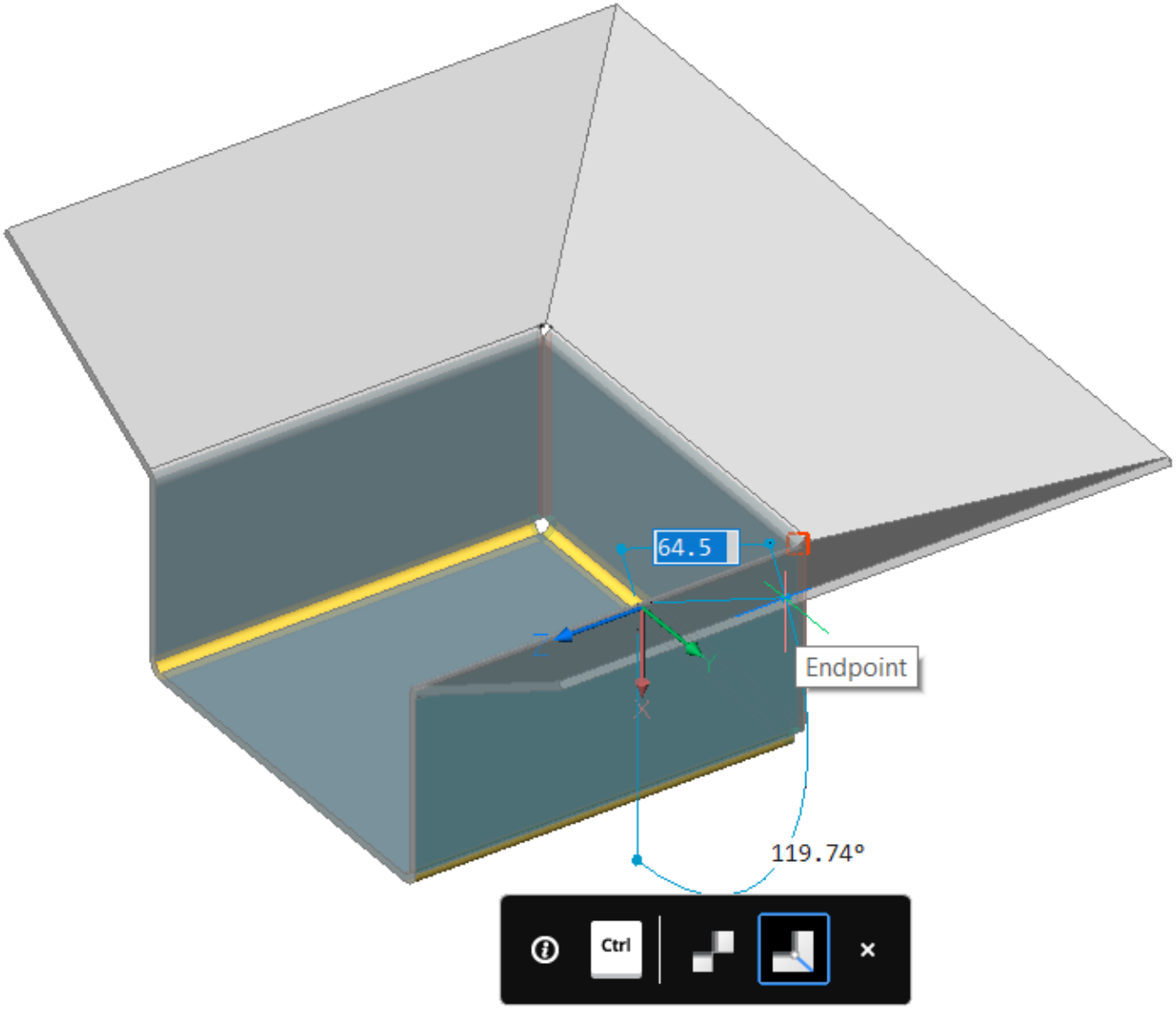


Figure 3.369

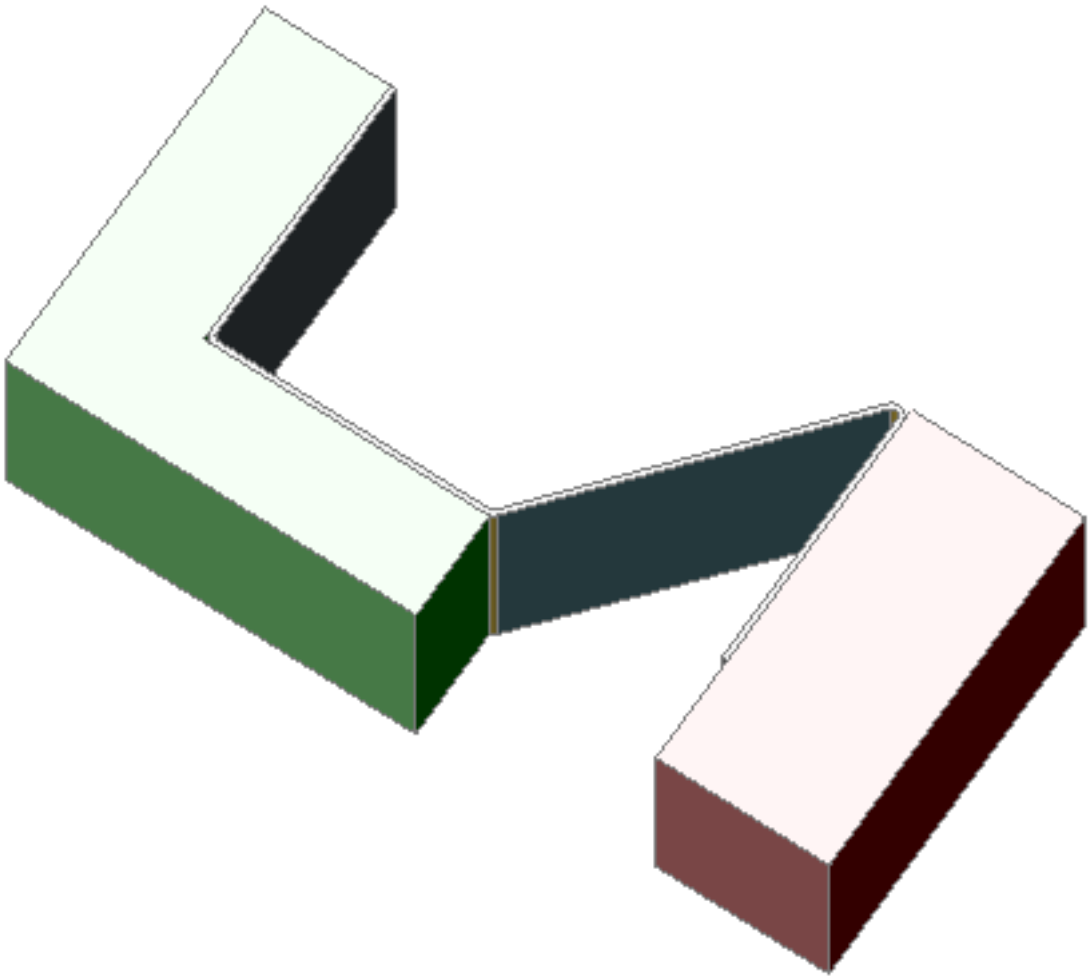


Figure 3.370

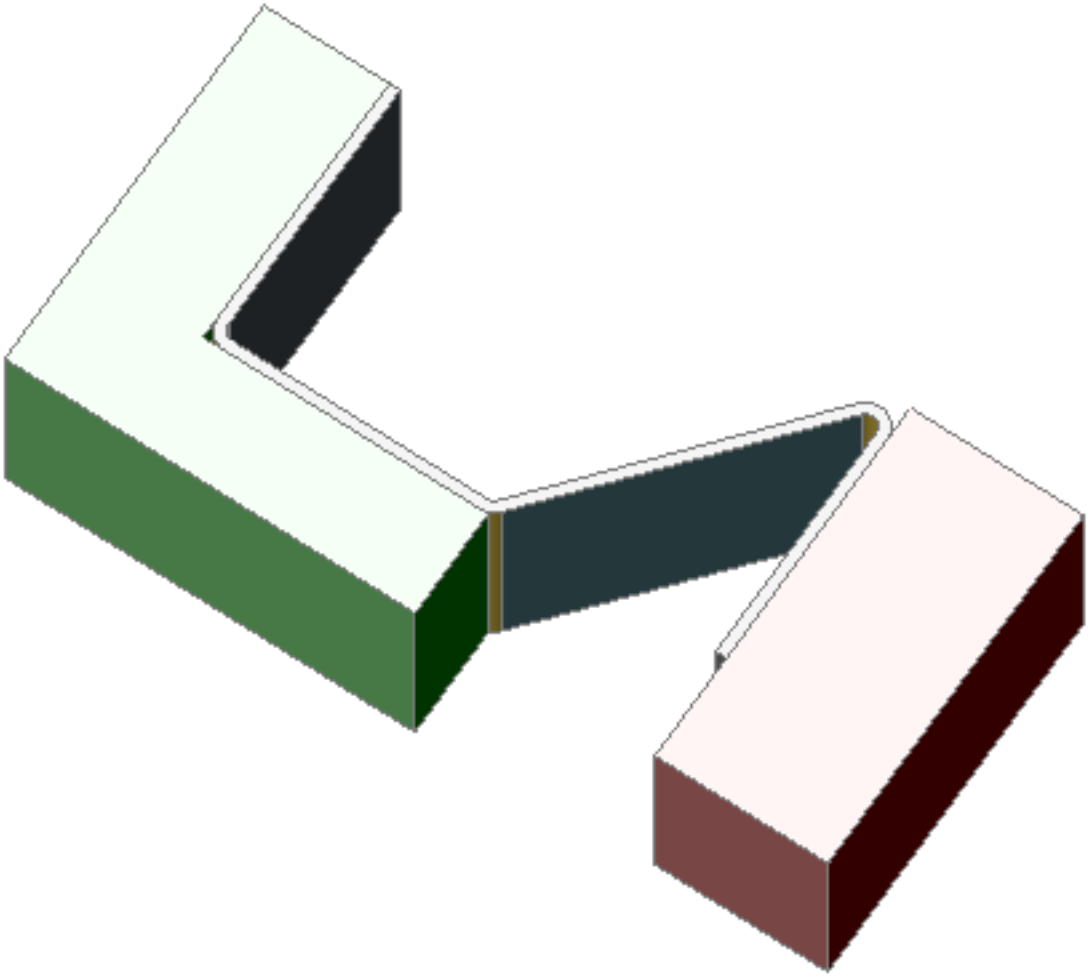
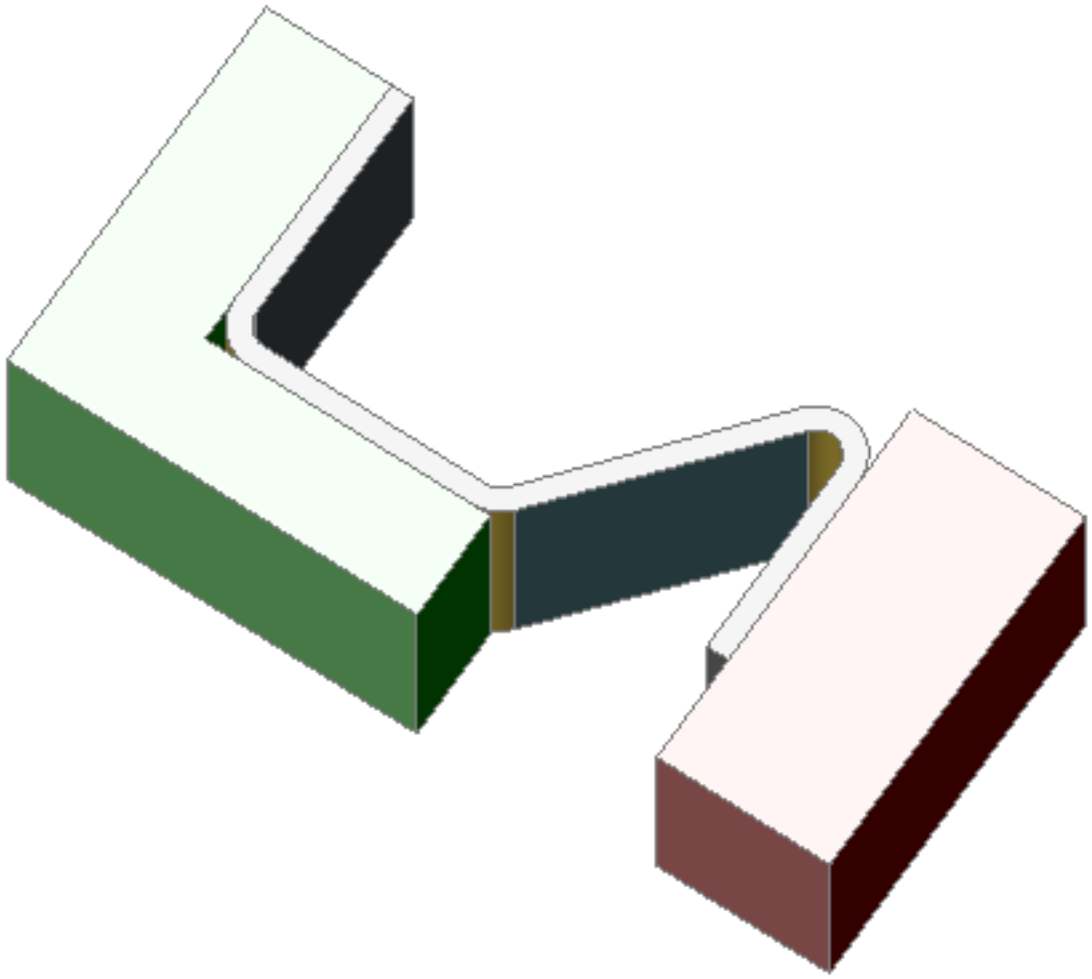


Figure 3.371



3.8.11 Working with Bends and Junctions

Two adjacent flanges of sheet metal parts are connected by a bend or a junction.As different from many other mechanical CAD systems, in BricsCAD bends and junctions are separate features: a bend can be swtiched to a junction and vice versa, which allows you to fix or optimize the unfolded representation of a sheet metal part easily.

Converting Hard Edges to Bends or Junctions

If adjacent flanges of a sheet metal part are not connected by a bend or junction, the part cannot be unfolded or exported to a CAM system. Therefore such hard edges must be converted to bends or junctions.

Converting hard edges to bends

1. Do one of the following:
   - Hover over a hard edge and click the **Create Bend** ( ) tool in the Quad (2.2.3) menu
   - Click the **Create Bend** ( ) tool button in the **Sheet Metal** ribbon tab
   - Choose **Create Bend** in the **Sheet Metal** menu
   - Type **smbendcreate** or the command bar.

   You are prompted: Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>:
2. Select a flange face or a hard edge. **You are prompted:** Entities in set: 1 **Select flange faces or hard edges:**
3. Do one of the following:
   - Select more faces or hard edges.
   - Right click or press Enter to create the bends or junctions on the selected items.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Sheetmetal/bendbr4.png}) Selected a hard edge	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee A bend is created

Converting Hard Edges to Junctions

1. Do one of the following:
   - Hover over a hard edge and click the **Create Junction** ( ) tool in the Quad (2.2.3) menu;
   - Click the **Create Junction** ( ) tool buttons in the **Sheet Metal** ribbon tab;
   - Choose **Create Junction** in the **Sheet Metal** menu;
   - Type **smjunctioncreate** in the command bar.

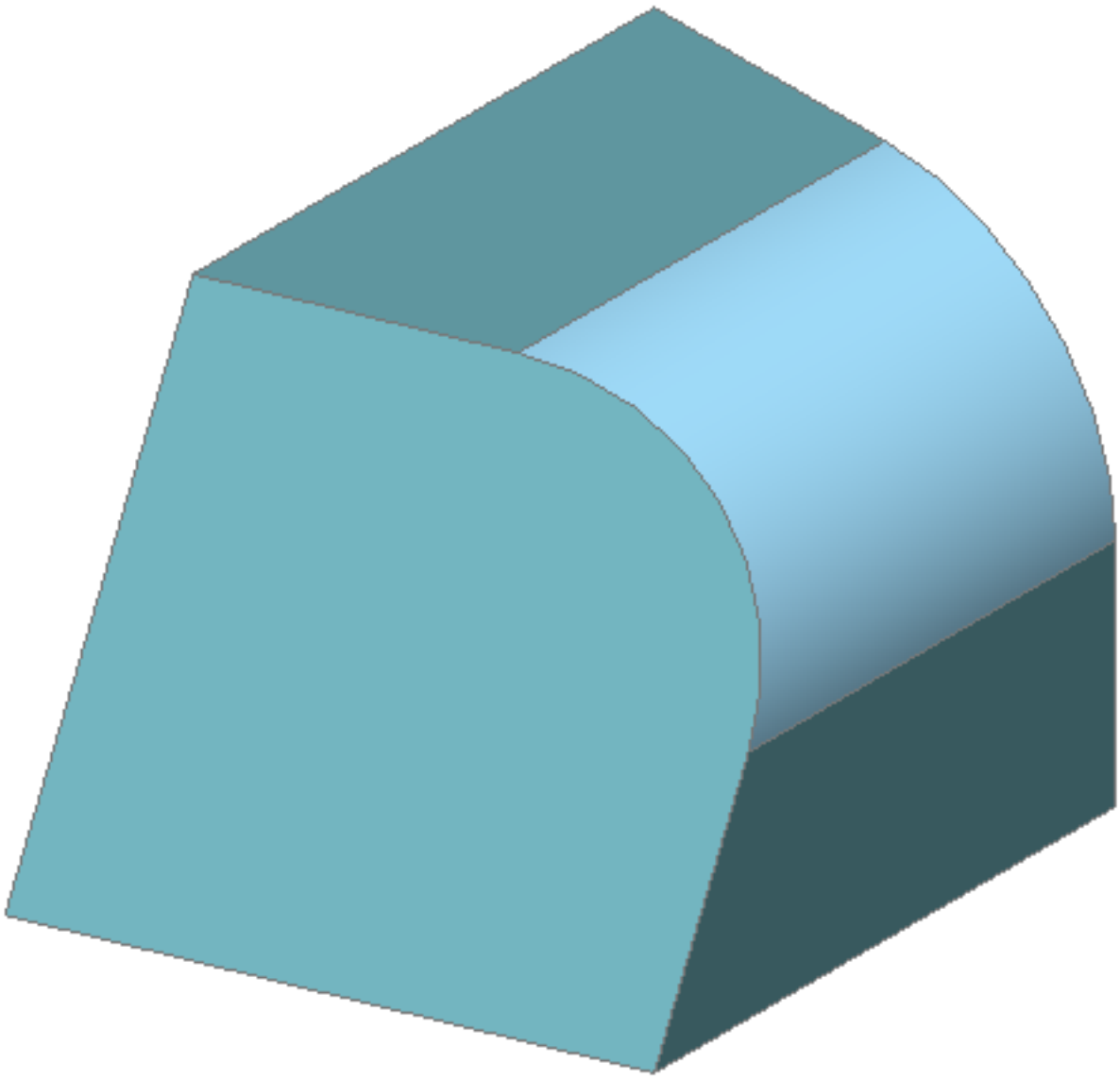
   You are prompted: Select hard edges or bends or flanges or 3D solids [Entire model] <Entire model>:
2. Select a flange face or a hard edge. **You are prompted:** Entities in set: 1 Select flange faces or hard edges:
3. Do one of the following:
   - Select more faces or hard edges.
   - Right click or press Enter to create the bends or junctions on the selected items.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Sheetmetal/bendbr5.png}) Selected hard edges. Notice that a corner relief has to be created first	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Shee Junctions created. Notice that unnecessary material cuts have been removed due to Smart Features (3.8.16)

Creating Curved Junctions

Lets have a look at a more complex scenario where a junction has to be cut over a non-linear edge. To make the part unfoldable, we need to:

Figure 3.372



- Cut the bend relief for one of the linear hard edges.
- Cut the junctions for the remaining linear and curved hard edges.
- Create a bend on the remaining hard edge. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mech

1. Create bend reliefs. Do one of the following
   - Click the**Create Relief**tool button ( ) on the**Sheet Metal**ribbon tab.
   - hoose**Create Relief**in the**Sheet Metal**menu.
   - Type**smreliefcreate**in the command bar.
2. **You are prompted:** Select a hard edge, bend face, flange face or 3D solid [Entire model]: Select a hard edge. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/bends9.png}) **You are prompted:** Select another hard edge or bend face for the corner relief:
3. Press Enter or right click. **You are prompted:** Enter relief size by bend radius ratio or [force Bend reliefs/Auto] <Auto>:
4. Press Enter or right click.Two bend reliefs are created for the edge: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom We created two bend reliefs because: A hard edge side with a ”free” vertex needs a relief in order to make orthogonal thickness face We created a bend relief on the side adjacent to the lofted bend because this feature is different from a regular bend: creating a corner relief there would be too big.

Figure 3.373

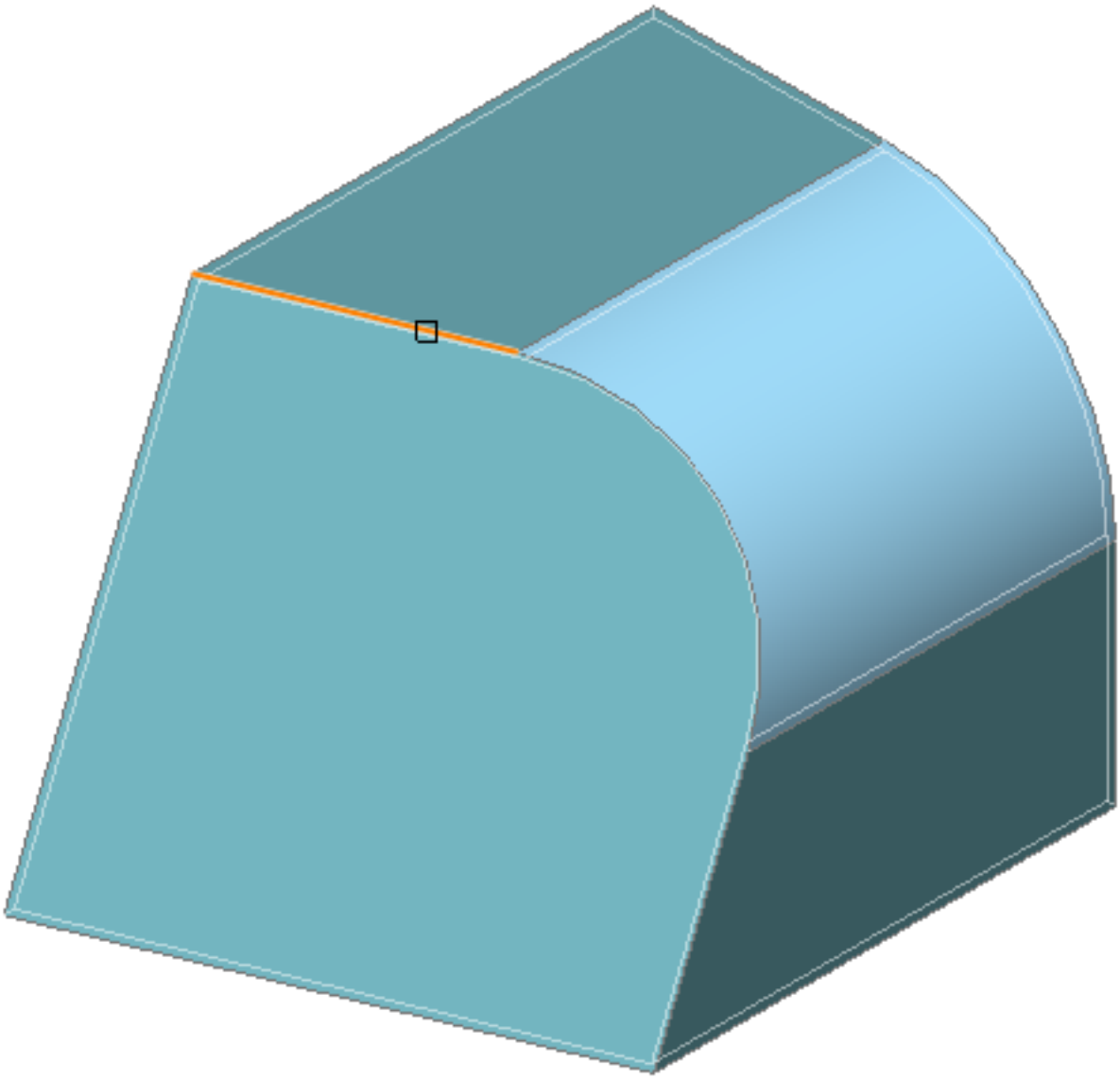
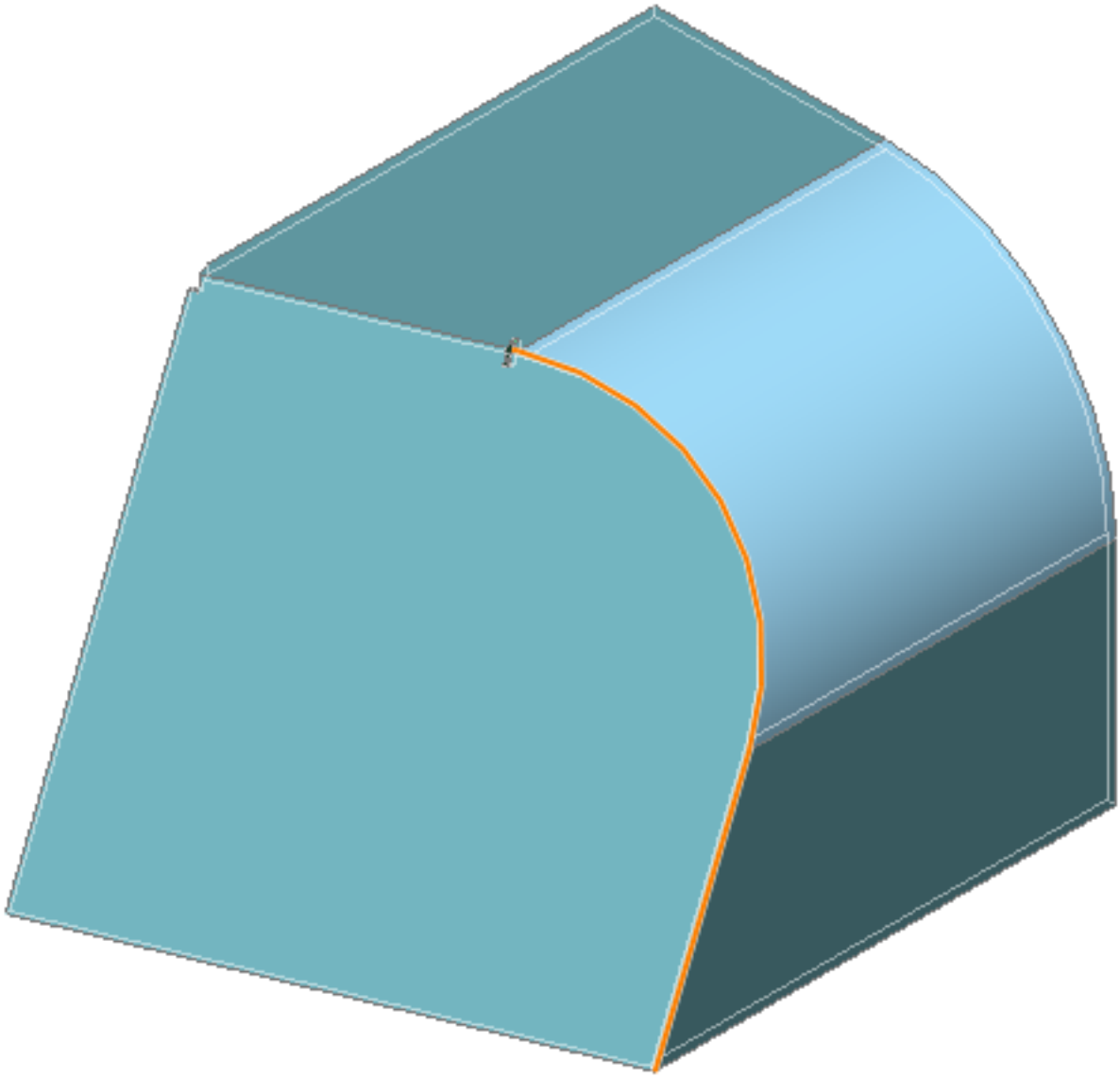


Figure 3.374



Figure 3.375



5. Create two junctions. Do one of the following:

   - Click the **Create Junction** ( ) tool buttons in the **Sheet Metal** ribbon tab;
   - Choose **Create Junction** in the **Sheet Metal** menu;
   - Type **smjunctioncreate** in the command bar.

6. **You are prompted:** Select hard edges or bends or flanges or 3D solids [Entire model] <Entire model>: Select following three hard edges: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetM
   Don't forget about small hard edge: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/
   The cut is created. Parts which corresponded to linear edge get regular junction features.

7. Create bend on the hard edge. Since it is the only hard edge left on the model, we can apply the command to entire model.

   - Click the **Create Bend** ( ) tool buttons in the **Sheet Metal** ribbon tab.
   - Choose **Create Bend** in the **Sheet Metal** menu.
   - Type **smbendcreate** or the command bar.

8. **You are prompted:** Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>: Press Enter. Bend is created. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Sheet

9. Unfold the part.

Figure 3.376

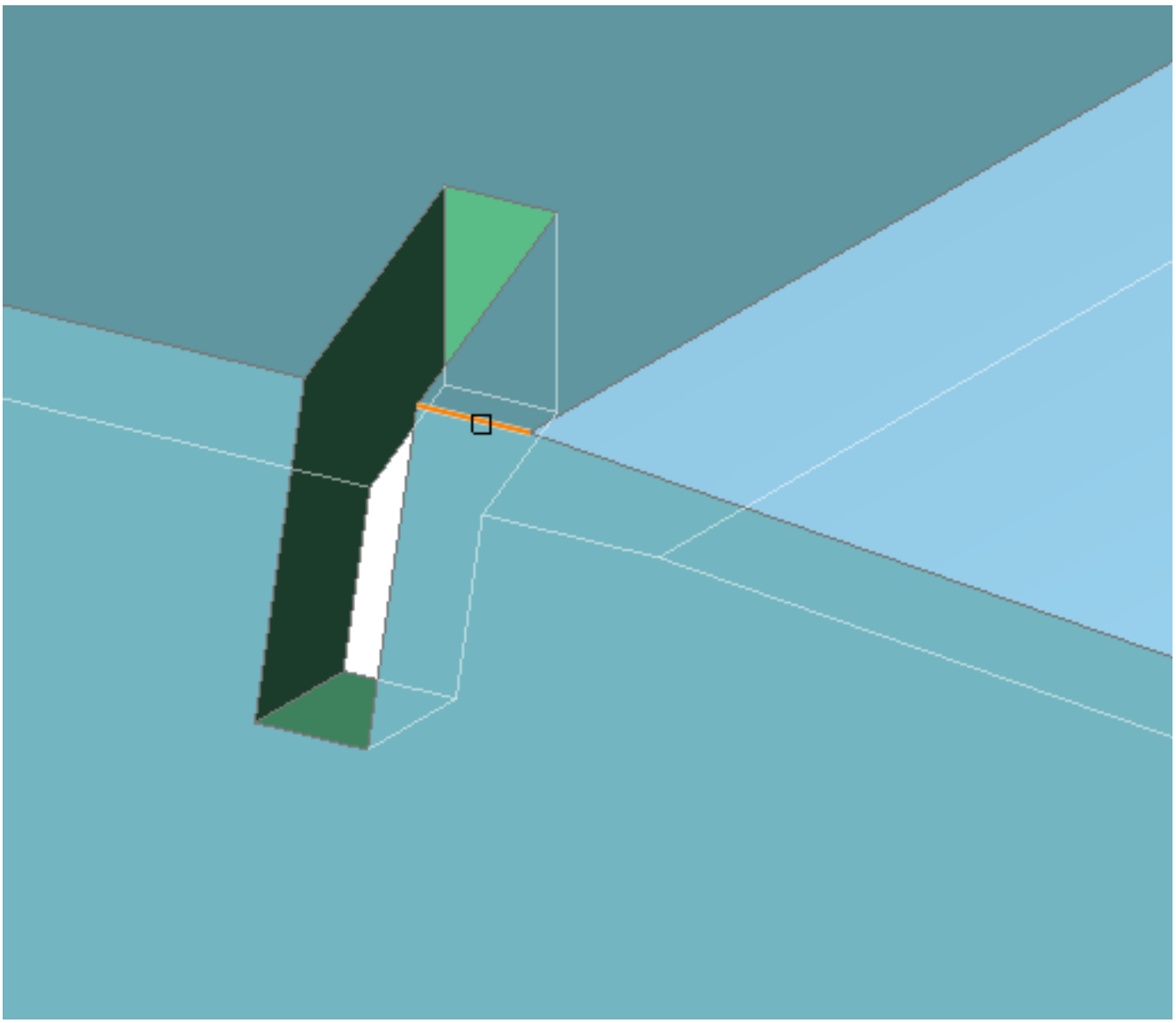


Figure 3.377

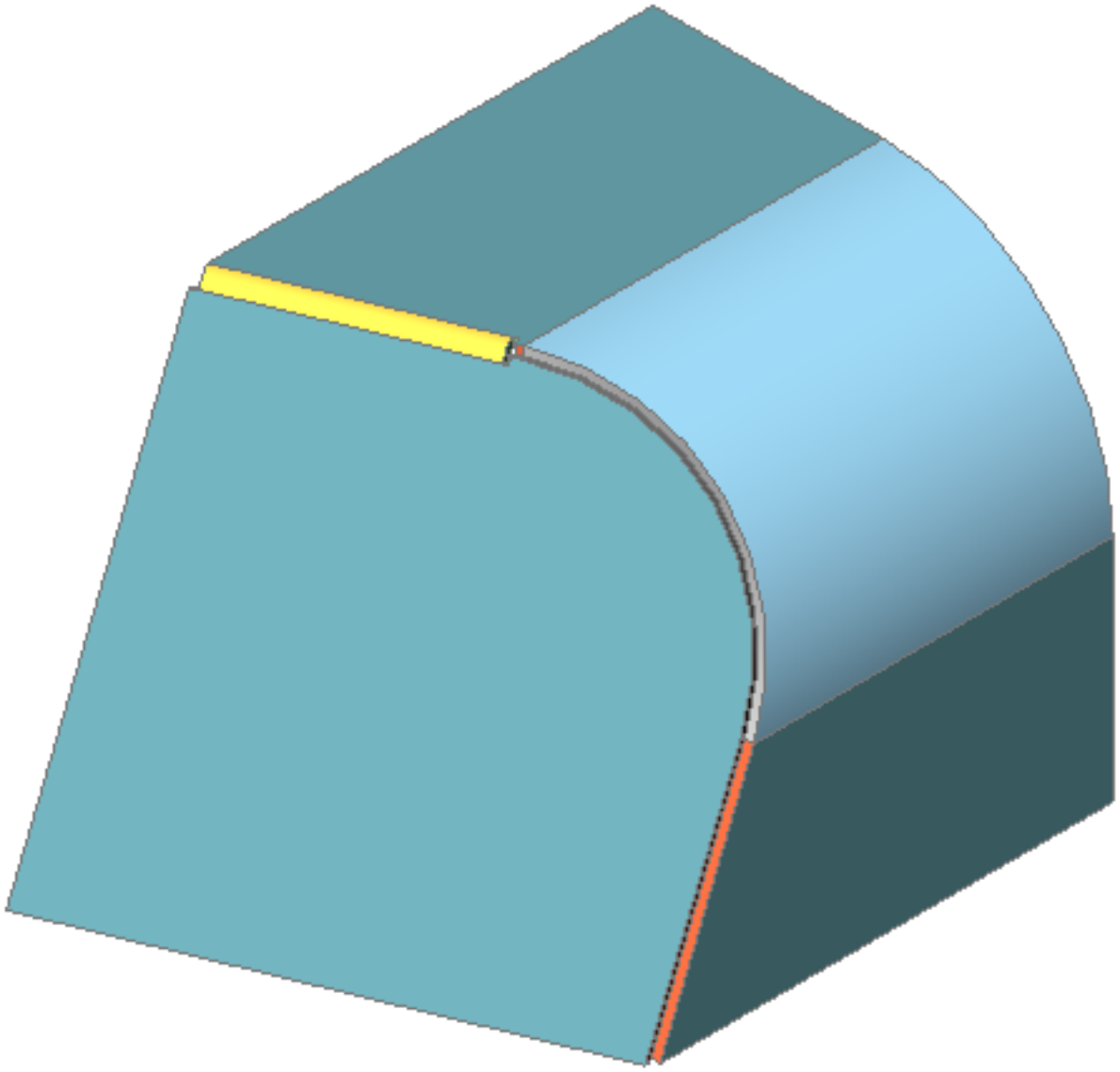
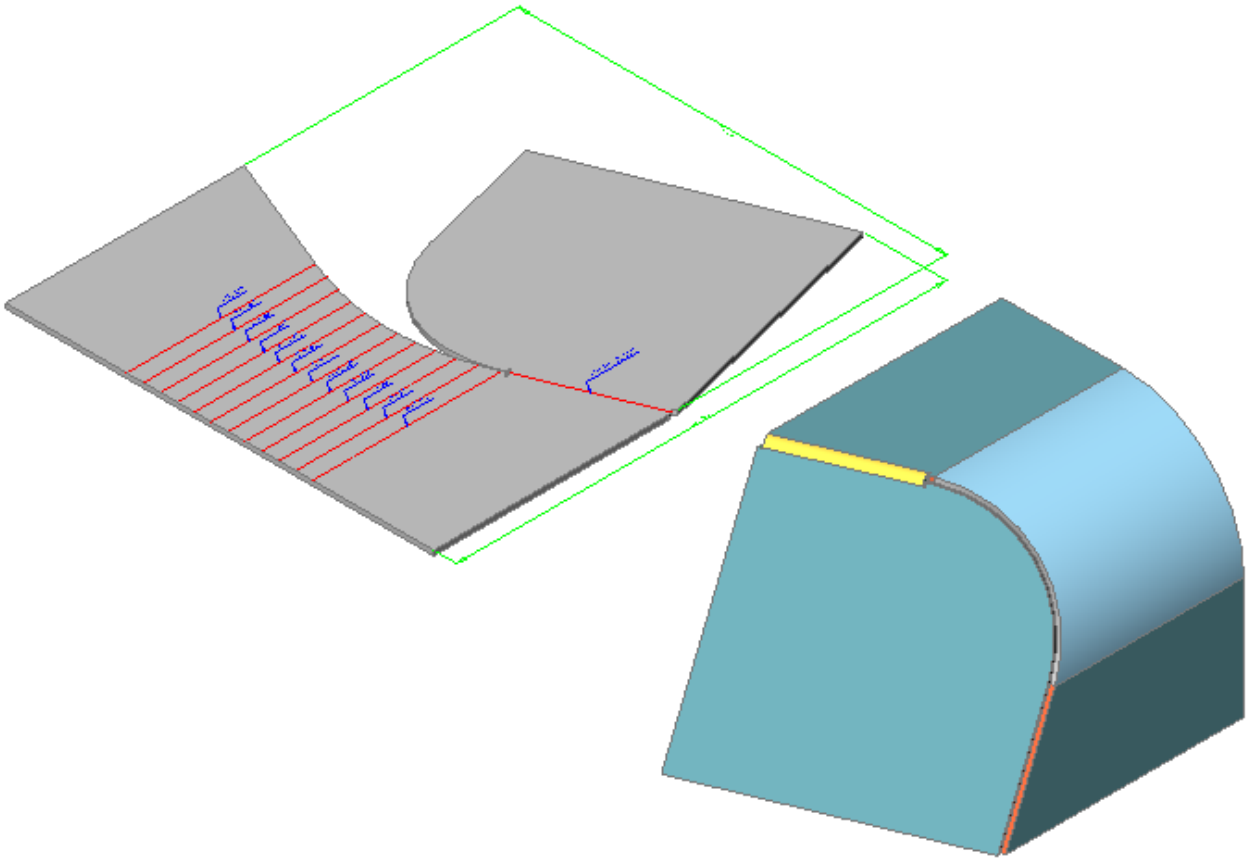


Figure 3.378



- Click the **Unfold Body** ( ) tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Unfold Body** in the **Sheet Metal** menu.
- Type **smunfold** or the command bar.

10. **You are prompted:** Select a flange or lofted bend face to start unfolding [Associative]: Select some flange face.
11. **You are prompted:** Select position of the unfolded body: Place the unfolded part.
12. **You are prompted:** Validate the unfolded body and select an option [save 2D geometry/save 3D geometry/Optimize bend annotations/Keep] <Keep>: Press Enter to keep the result. (\vref {fig:C:/Dev/Task-2448/images/staticl

To Split a Thickness Face

To extend a flange which shares the same thickness face with other flanges, the thickness face needs to be split.

1. Launch the SmImprint (6.28.17) command ( ). Prompts you: **Select a thickness face:**
2. Select the shared thickness face. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source
3. Right click or press Enter to split the face. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGu
4. The flanges can now be extended separately: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Us

Figure 3.379

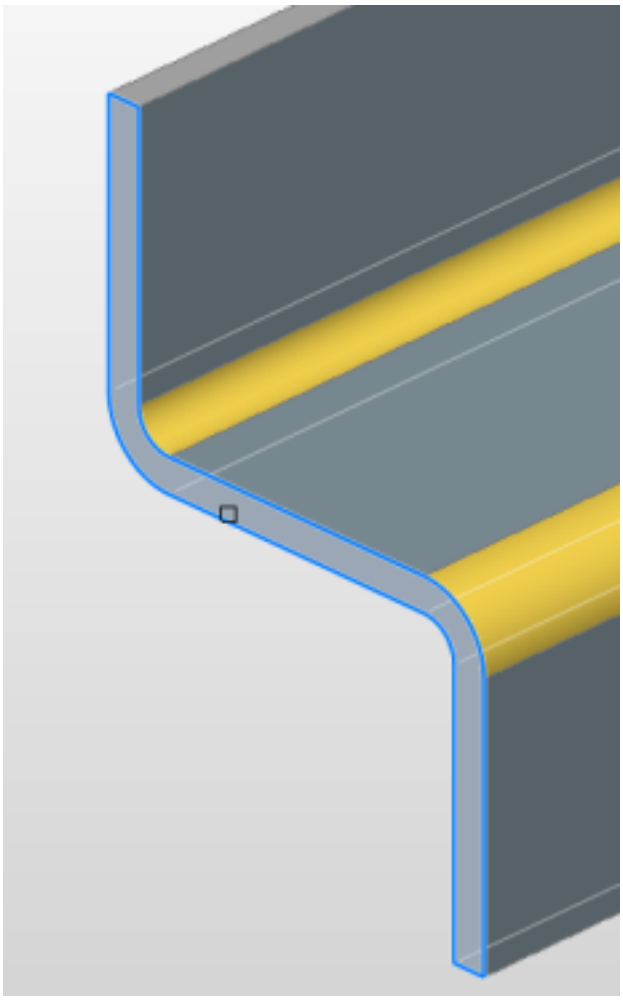
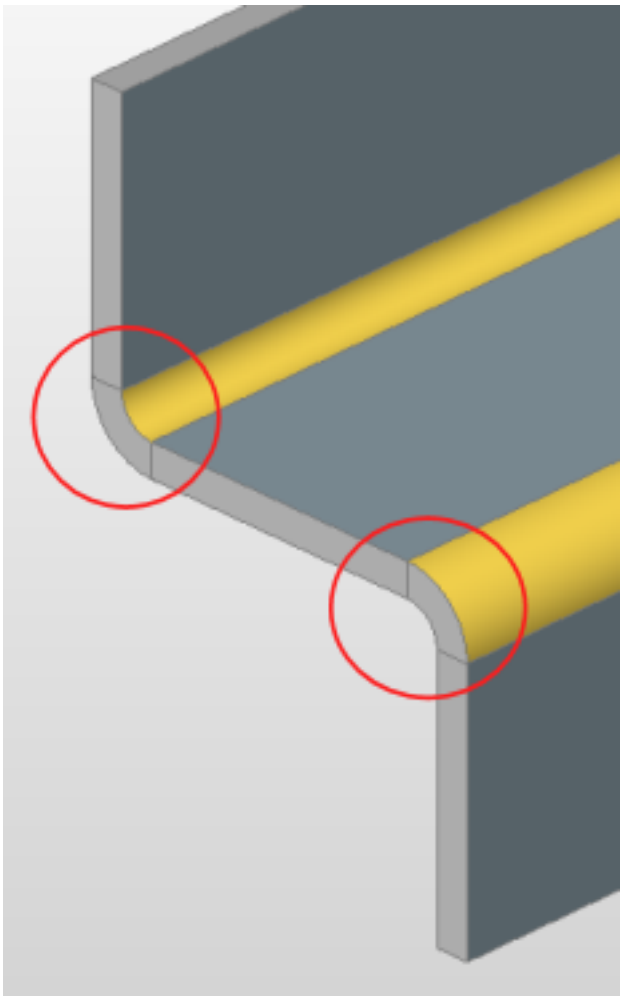


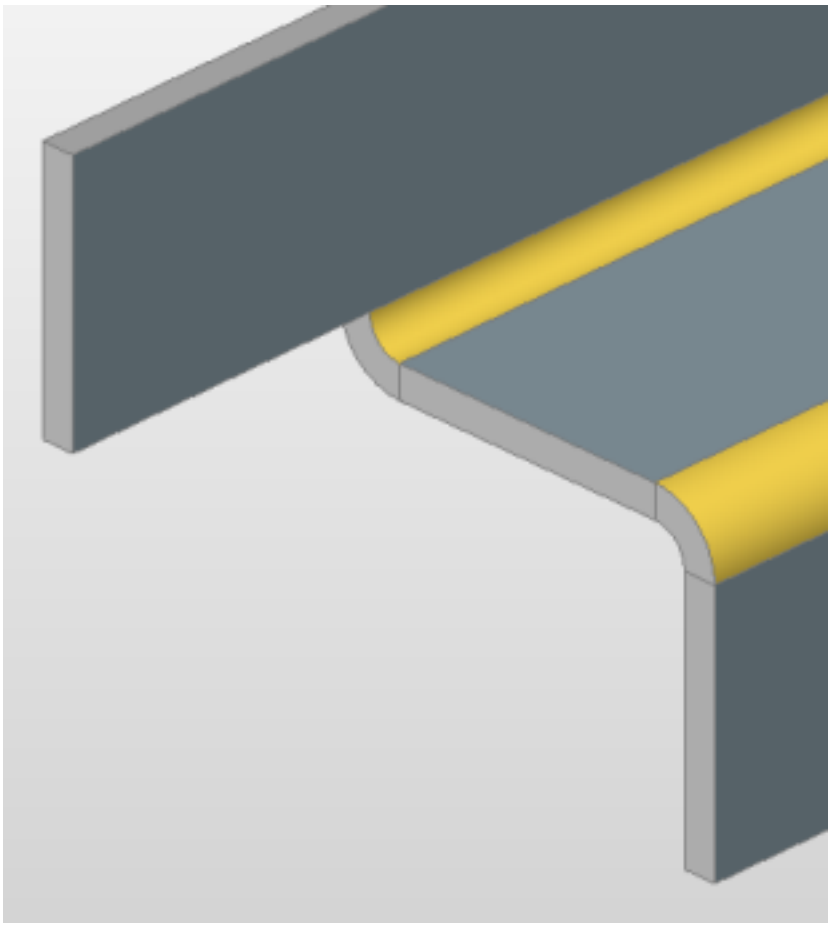
Figure 3.380



5. Execute the DmSimplify (6.12.27) command to remove the unneeded splits, thus unifying the coplanar thickness faces.

Tutorial: Split thickness faces into regions Click here (3.7.3) to watch

Figure 3.381



Changing the Bend Radius

To change the overall bend radius:

1. Select the root node in the Mechanical Browser (3.2.2)
2. Find the **Bend radius** group in the bottom part of the Mechanical Browser (3.2.2).
3. Do one of the following:
   - Set **Type** to **Absolute Value**, then type the radius value in the **Value** field. The **Bend Radius** field changes accordingly.
   - Set **Type** to **Thickness ratio**, then type a value in the **Value** field. The **Bend Radius** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

To change the bend radius for a particular bend:

1. Select the bend node in the Mechanical Browser (3.2.2) The selected bend highlights in the model. The Bend Properties display in the bottom part of the Mechanical Browser (3.2.2).
2. Do one of the following:
   - Set the **Type** to **Global value.** The current global bend radius is applied to the selected bend.
   - Set the **Type** to **Absolute value,** then type the radius value in the **Value** field. The **Bend Radius** field changes accordingly.
   - Set the **Type** to **Thickness Ratio** and type a value in the value field. The **Bend Radius** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

Changing the Junction Gap

To change the overall junction gap:

1. Select the root node in the Mechanical Browser (3.2.2).

2. Find the **junction gap** group in the bottom part of the Mechanical Browser (3.2.2)
3. Do one of the following:
   - Set **Type** to **Absolute Value**, then type the gap value in the **Value** field. The **junction gap** field changes accordingly.
   - Set **Type** to **Thickness ratio**, then type a value in the **Value** field. The **junction gap** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

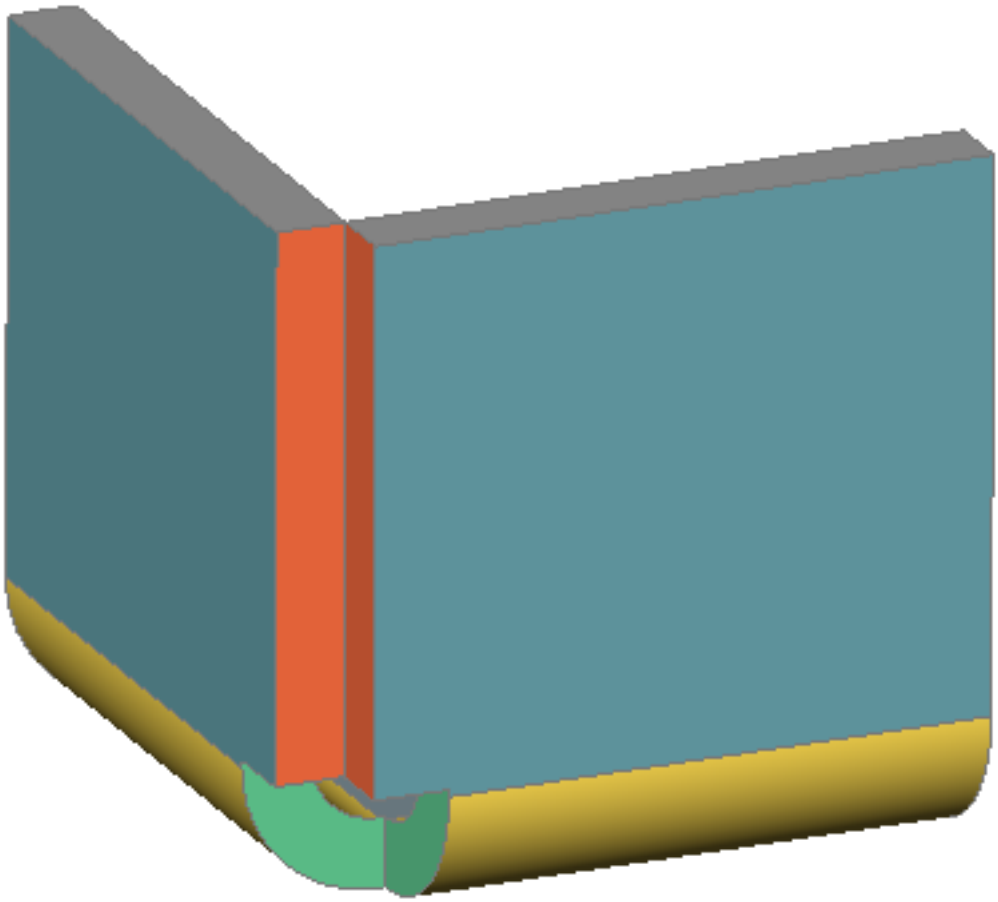
To change the junction gap for a particular bend:

1. Select the bend node in the Mechanical Browser (3.2.2) The selected bend highlights in the model. The Bend Properties display in the bottom part of the Mechanical Browser (3.2.2).
2. Do one of the following:
   - Set the **Type** to **Global value.** The current global junction gap is applied to the selected bend.
   - Set the **Type** to **Absolute value,** then type the gap value in the **Value** field. The **junction gap** field changes accordingly.
   - Set the **Type** to **Thickness Ratio** and type a value in the value field. The **junction gap** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

Changing the Junction Type

By default, BricsCAD creates symmetric junctions, as shown in the image below: (\vref {fig:C:/Dev/Task-2448/images/staticbri
The SmJunctionSwitch (6.28.19) command allows changing a symmetrical junction feature to one with overlapping faces.

Figure 3.382



To change the junction type using the Quad:

1. Hover the cursor over one the thickness faces of this junction.
2. Select **Switch Junction Type** in the Sheet Metal command group in the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/bends14.png}) The selected thickness face is aligned with the flange face. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mecha

Figure 3.383

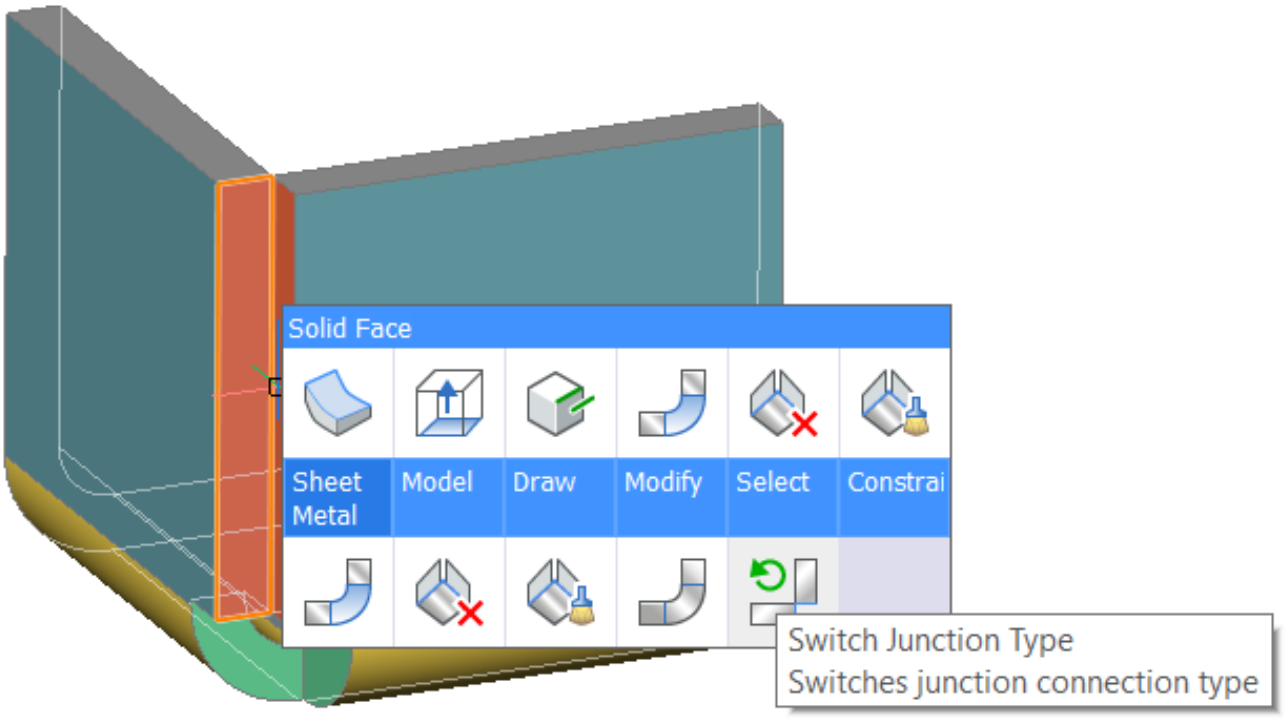
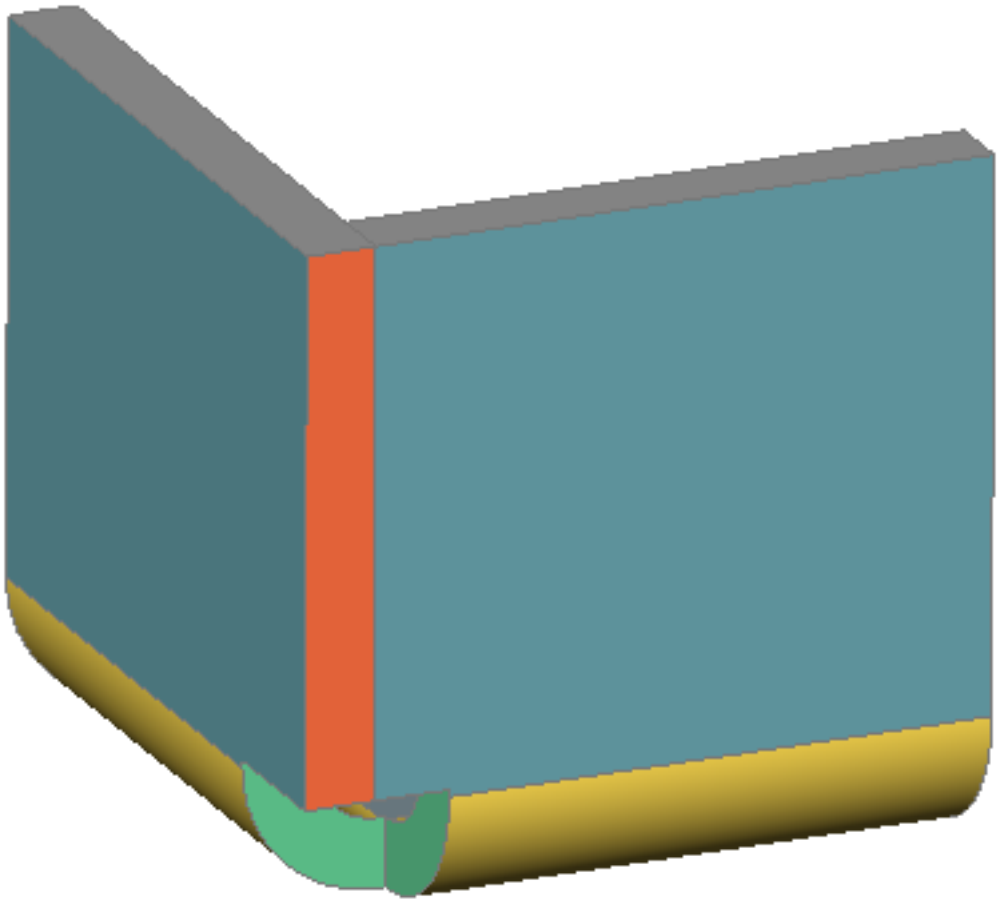


Figure 3.384



To change the junction type in the Mechanical Browser Method 1:

1. Select the corresponding junction in the tree. The junction properties display in the bottom part of the Mechanical Browser (3.2.2).
2. Select the **Connection type** property.
3. Choose the desired junction type. The options are: **Symmetric, Aligned to Flange_X** or **Aligned to Flange_ Y**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingS
The junction is updated accordingly.

Method 2:

1. Select the corresponding junction in the tree. A context menu displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
2. Right click and choose **Switch type of feature** in the context menu.

NOTE	Both symmetric and aligned junctions maintain the same gap value between connected flanges.

Figure 3.385

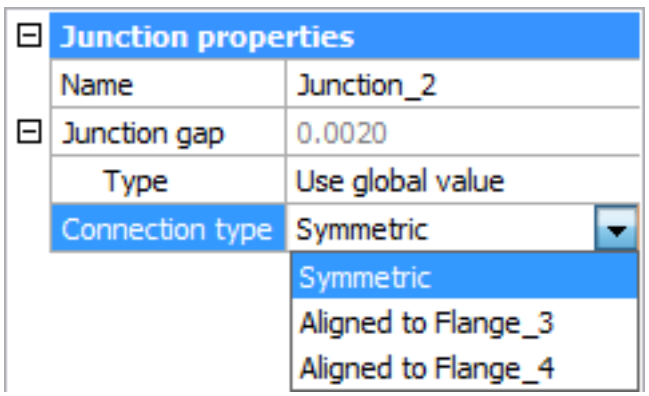


Figure 3.386

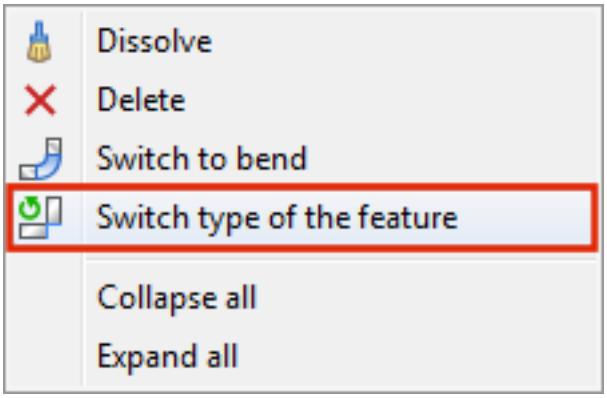


Figure 3.387

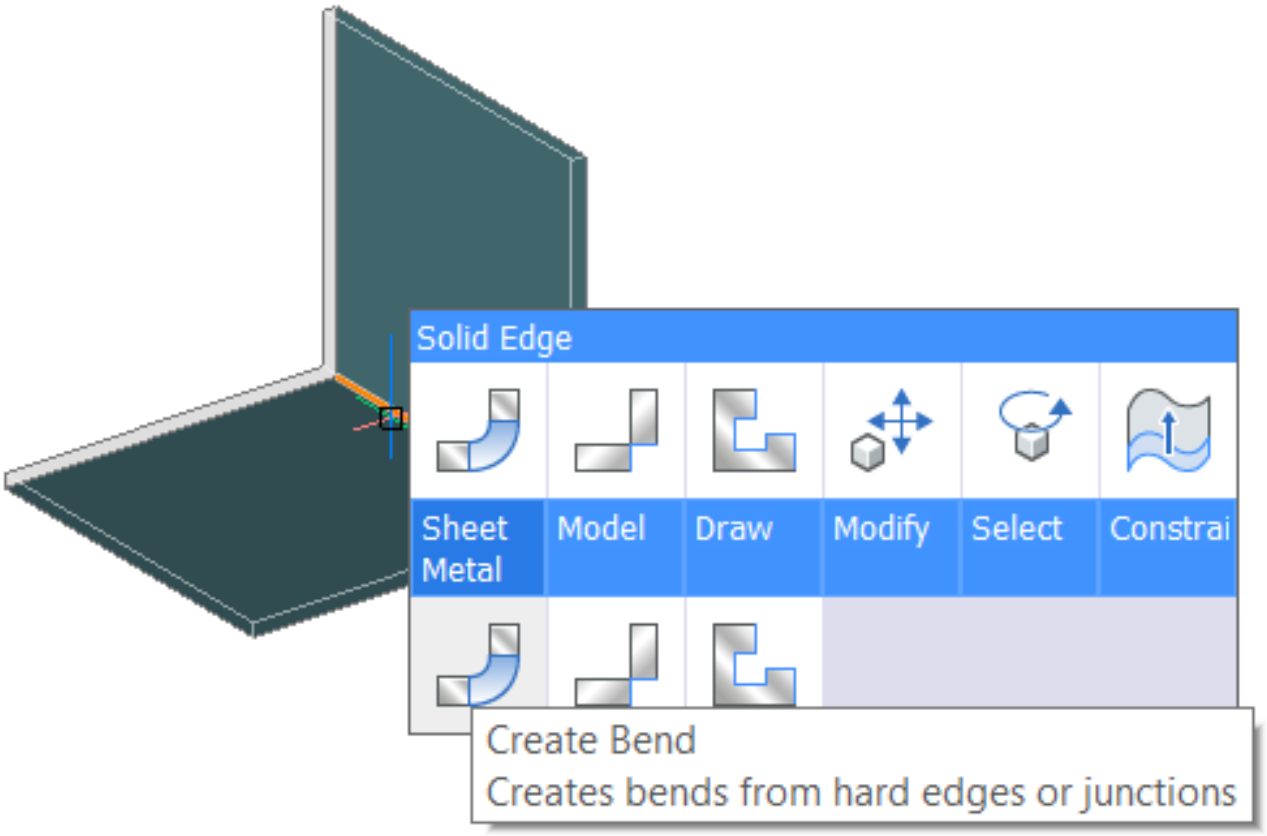


Figure 3.388

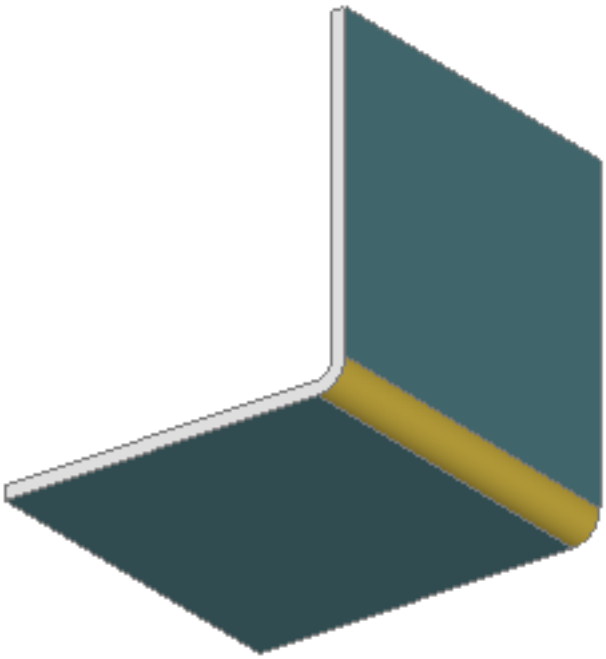


Figure 3.389

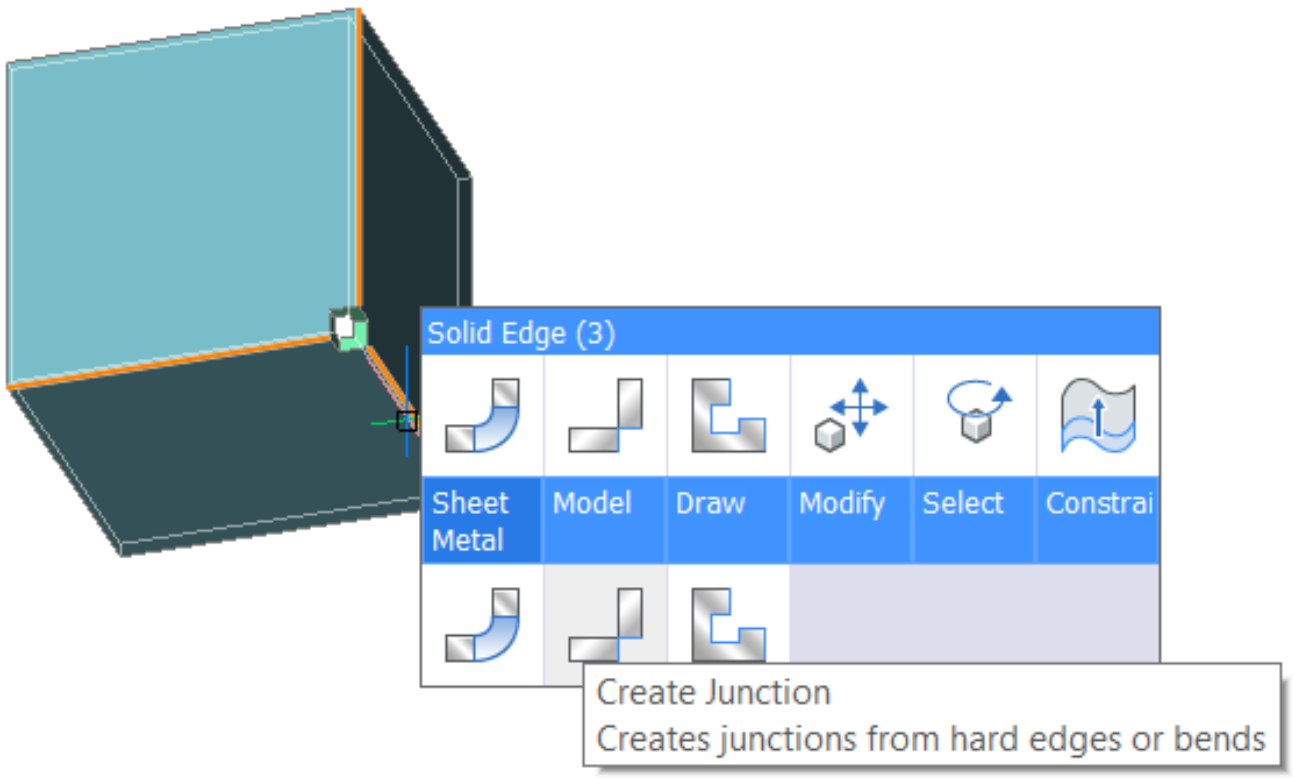
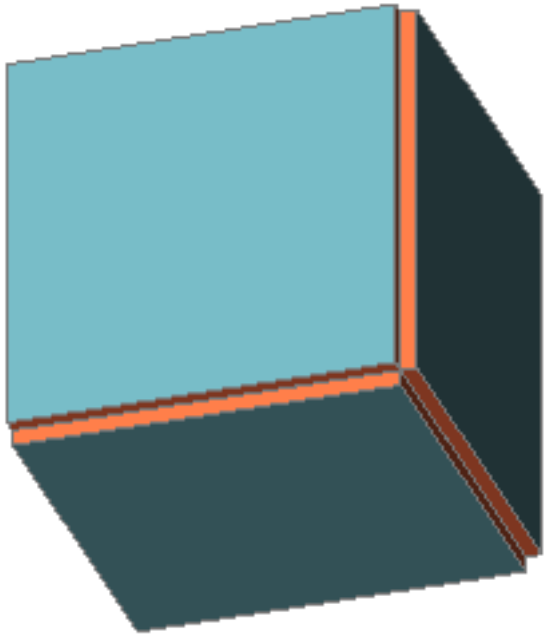


Figure 3.390



3.8.12 Creating and Splitting Lofted Bends

BricsCAD allows you to create or recognize (3.8.14) lofted sheet metal parts with curvilinear geometry, which can be manufactured using multiple bending.

General Procedure to Create a Sheet Metal Part with a Lofted Bend

1. Launch the SmLoft (6.28.20) command. Do one of the following:

   - Click the **Create Lofted Bend** tool button ( ) on the **Sheet Metal**toolbar.
   - Click the **Create Lofted Bend** tool button ( ) on the **Sheet Metal** ribbon bar.
   - Choose **Create Sheet Metal Loft** in the **Sheet Metal** menu.

   You are prompted: Select two cross sections:

2. Select two profile entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Dra
   You are prompted: Set fillets Radius/Thickness/Single bend/<Flanges with bends>:

3. Press Enter to accept the default settings. A preview of the sheet metal part is created. **You are prompted:** Set fillets Radius/Thickness/Single bend/Flanges with bends or make thicken Both sides/Inside/<Outside>: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingS

4. Hit the CTRL key to choose the position of the sheet metal part thickness with respect to the source profiles: **Left**, **Center** or **Right**. If TIPS (1.12.2)is ON the tips widget displays.

5. Press Enter to create the sheet metal part. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG
   If FEATURECOLORS (1.13.11) = ON, you can distinguish between the two flanges ( ) and the lofted bend ( ).

The properties of the lofted bend display in the Mechanical Browser (3.2.2): (\vref {fig:C:/Dev/Task-2448/images/staticbricsys
Optionally select the lofted bend feature to edit its properties:

- **Number of subdivisions**: the number of flat regions in the unfolded representation of a lofted bend feature (the greater the number, the more bend lines will be added to the unfolded view for the lofted bend).
- **Force triangulation**: most of lofted bends can be unfolded using an intelligent algorithm of quad tessellation; however sometimes it might produce non-optimal results. In such cases set this property to **Yes**to switch to more straight-forward algorithm which creates triangle-like quads.

Tutorial: Lofted bends in BricsCAD Sheet Metal Click here (3.7.2) to watch

Figure 3.391

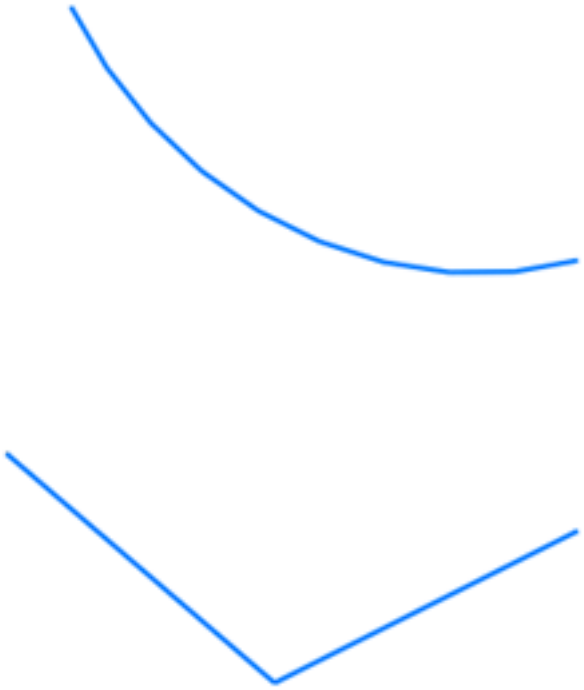


Figure 3.392

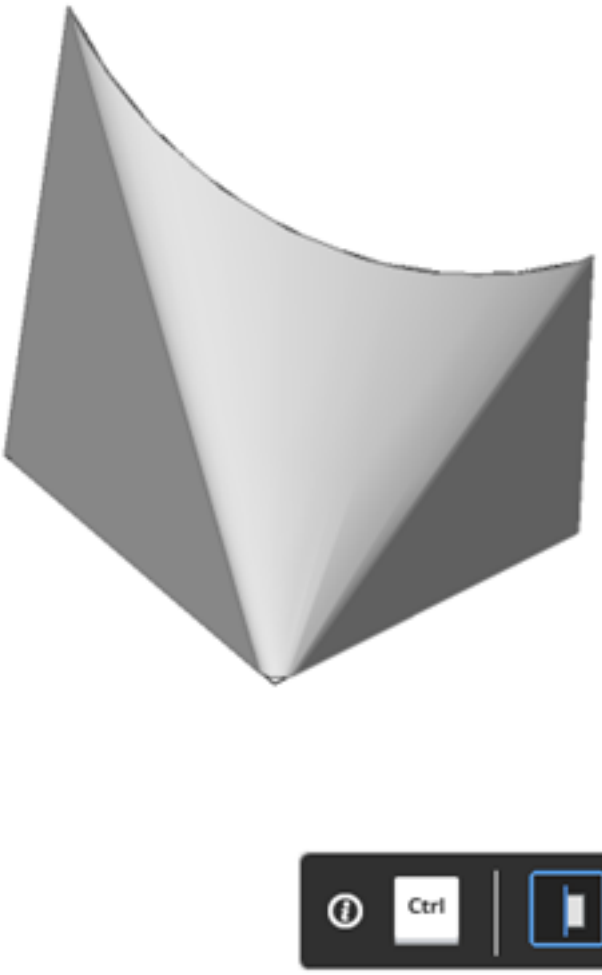


Figure 3.393

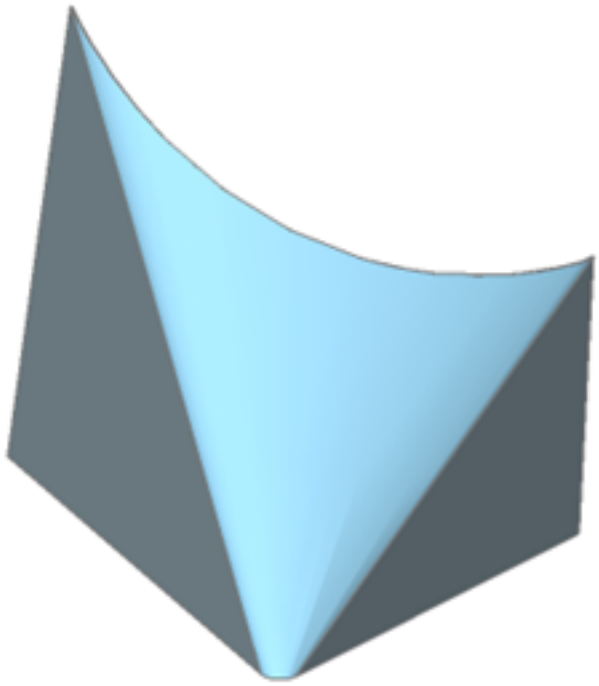
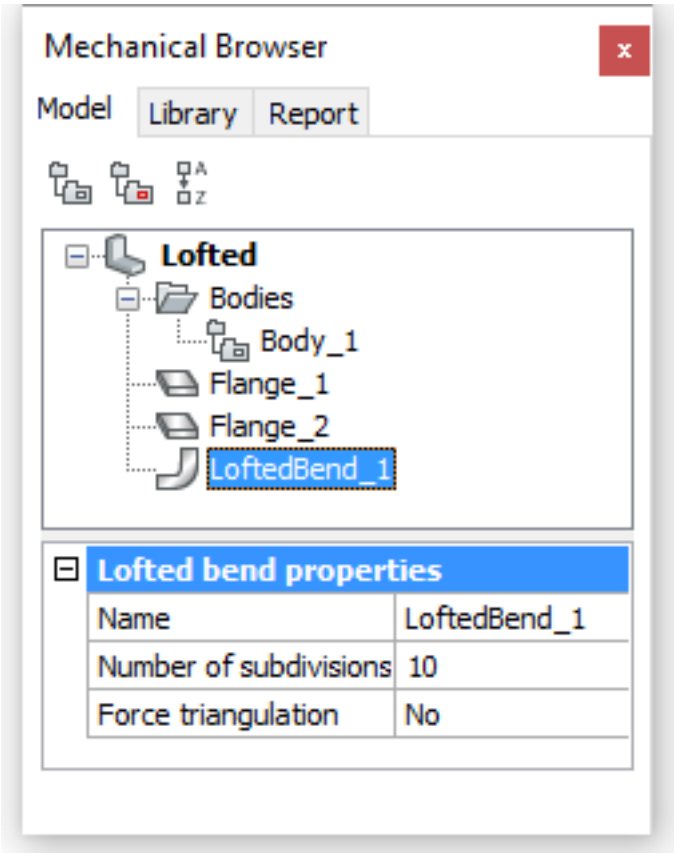


Figure 3.394



To Split a Lofted Bend

Lofted bends can split along a line on the lofted surface at an arbitrary point:

1. Do one of the following:

   - Click the **Split** tool () on the **Sheet Metal** ribbon tab;
   - Choose **Split** the **Sheet Metal** menu;
   - Type **smsplit** in the command bar

   You are prompted: Select flange or lofted bend face:

2. Select a lofted bend face. **You are prompted:** Select point on lofted bend face:

3. Select a point on the lofted bend surface. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mecha

The lofted bend is split. If the connecting junction is dissolved, you can freely move the separated part. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/lofted2.png}) (\vref {fig:C:/Dev/Tas 2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/lofted1.png})

Figure 3.395

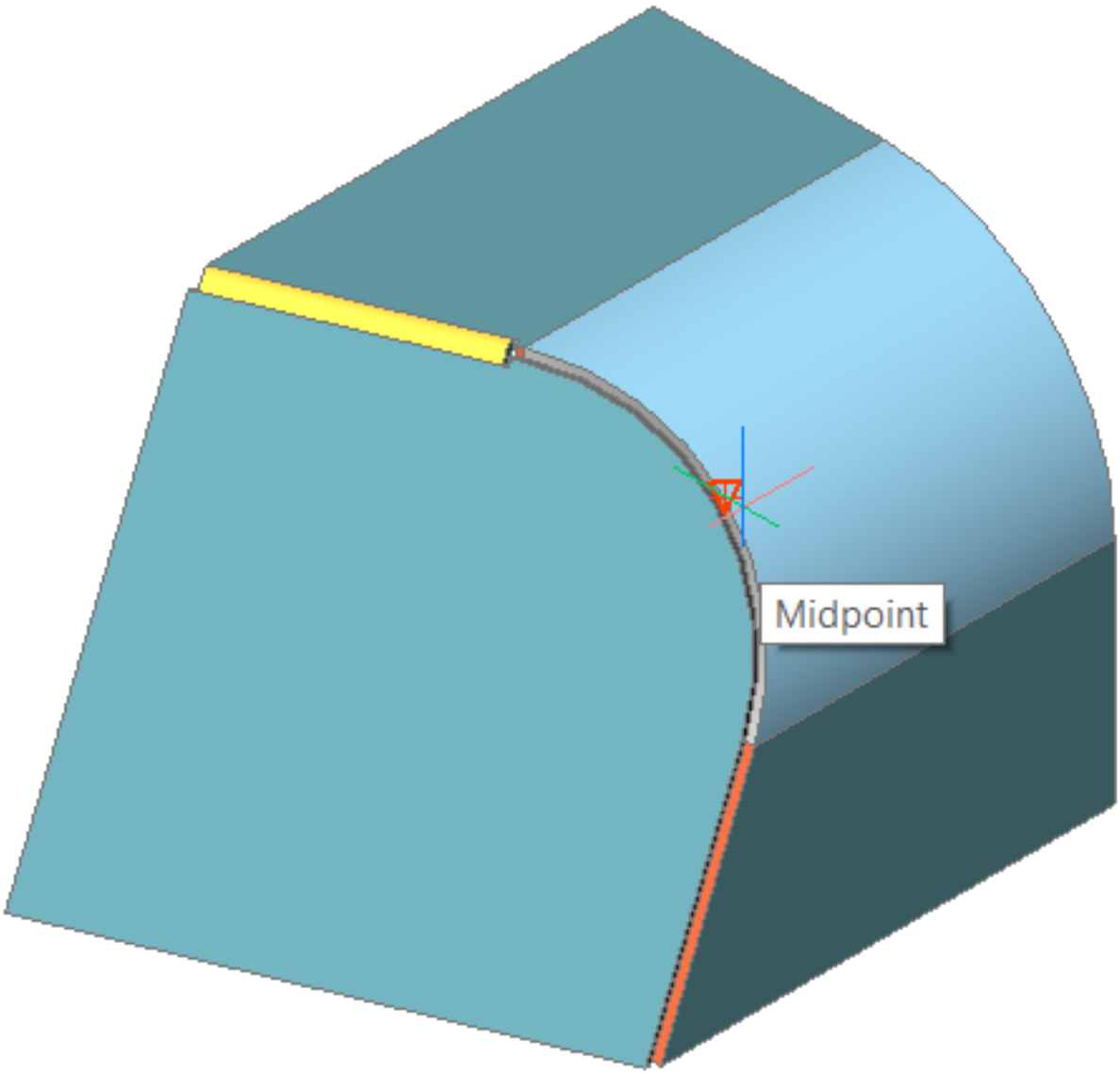


Figure 3.396

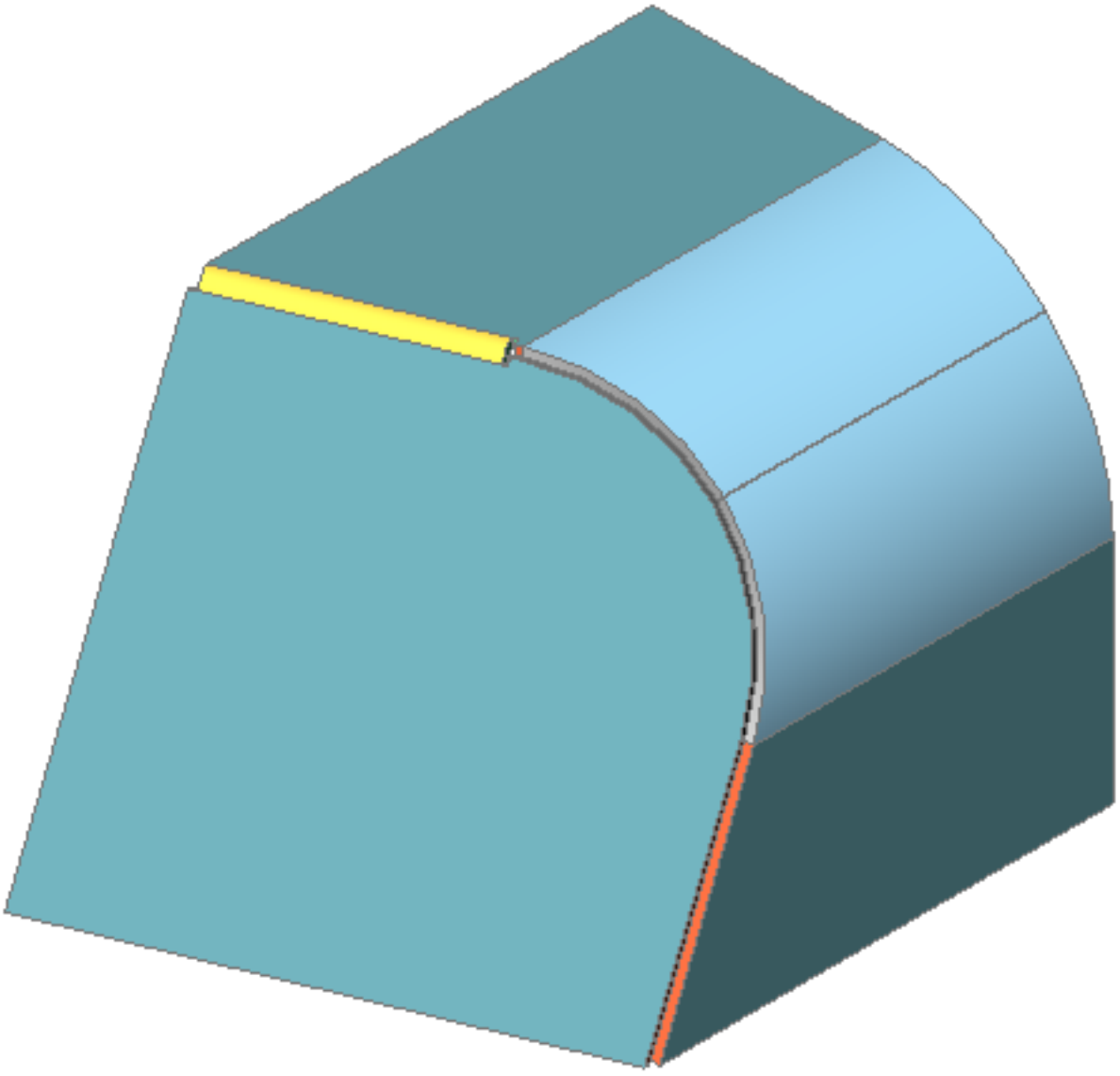
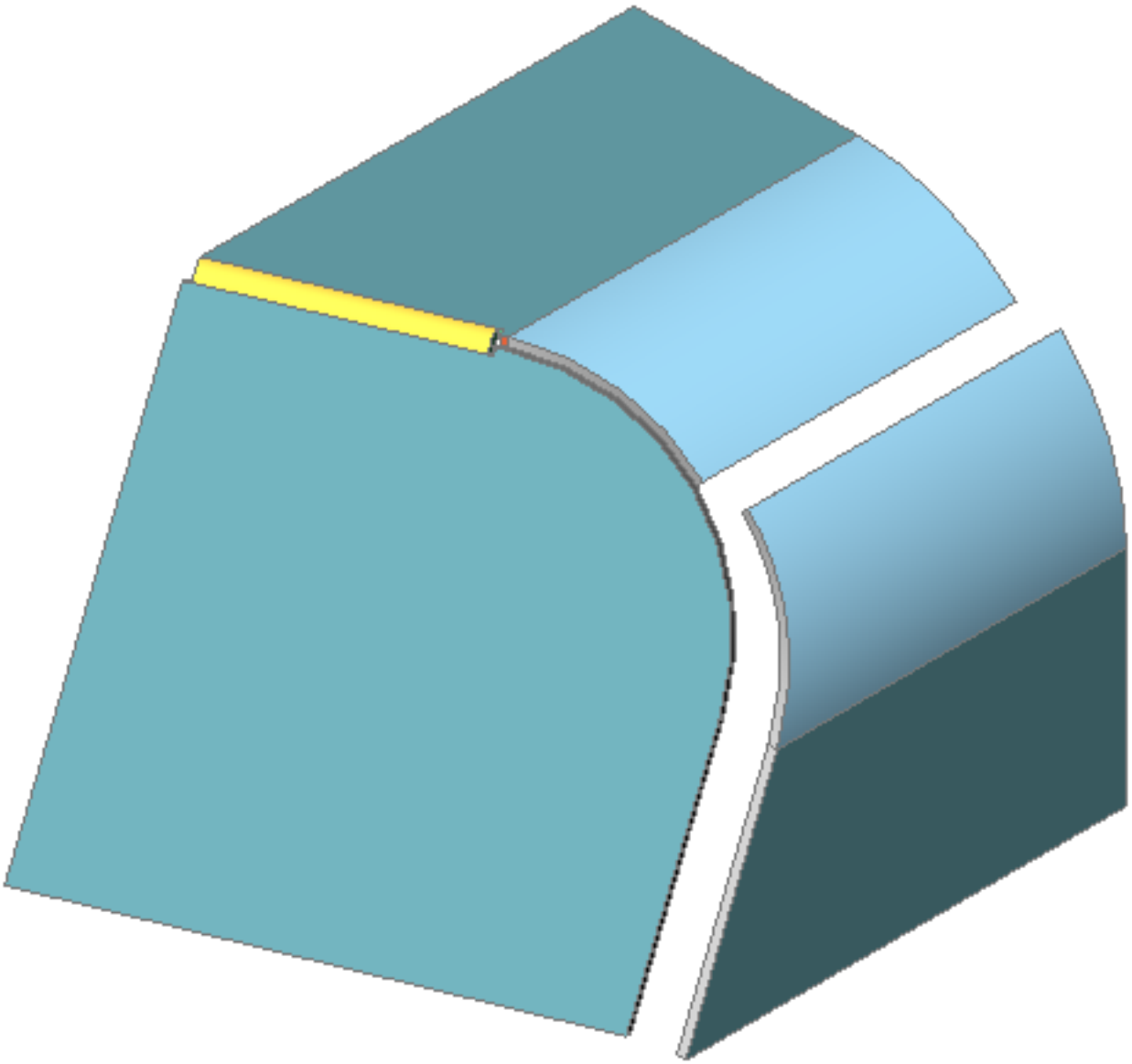


Figure 3.397



3.8.13 Bend Relief and Corner Relief Feature Operations

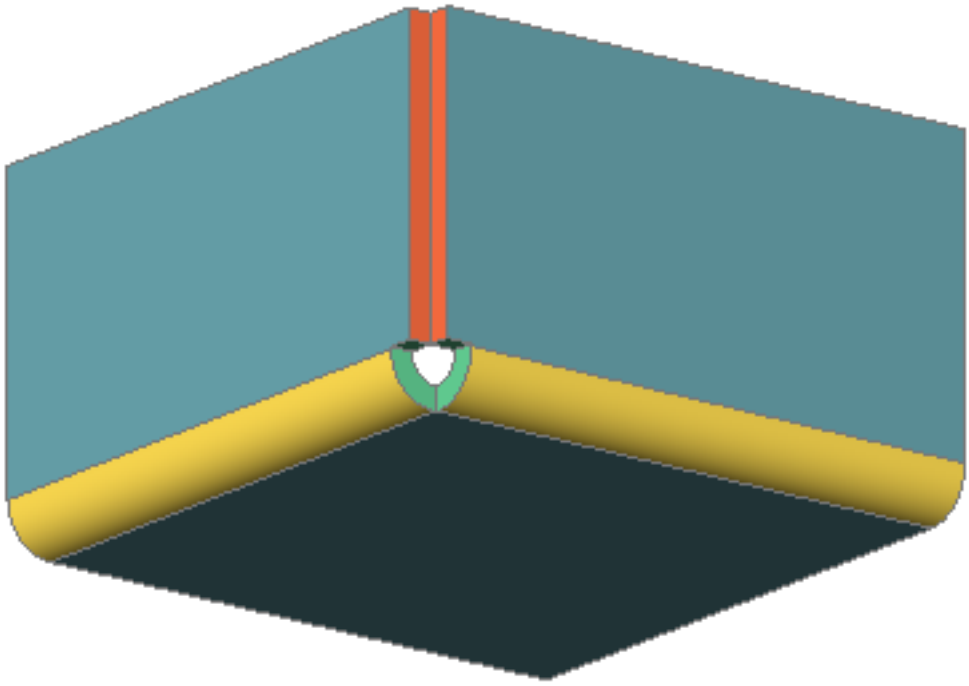
Commands: SMFLANGEEDGE (6.28.13), SMRELIEFCREATE (6.28.22), SMJUNCTIONSWITCH (6.28.19), SMRELIEFSWITCH (6.28.23) Reliefs are needed in sheet metal parts to avoid material tear when bent on press breaks.Basically you should not care about reliefs, because BricsCAD automatically creates all needed reliefs when you create a bend (implicitly or explicitly). However if you rework poor sheet metal parts, you may find it useful to create corner and bend reliefs with the dedicated tools.

Corner Reliefs and Junctions

When you create a flange using theSmFlangeEdge (6.28.13) command, adjacent to a bend edgea corner relief is created automatically in the corner where the three flanges meet. In additiona junction is created between the two flanges, which are not connected with a bend. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechar

Tutorial: Designing from scratch in BricsCAD Sheet Metal Click here (3.7.8) to watch

Figure 3.398



Creating Missing Reliefs

1. Do one of the following:

   - Click the **Create Relief** tool button ( ) on the **Sheet Metal** ribbon tab.
   - Choose **Create Relief** in the **Sheet Metal** menu.
   - Type **smreliefcreate** in the command bar.

   You are prompted: Select a hard edge, bend face, flange face or 3D solid [Entire model]:

2. Select an item (see the note below). Enter relief size by bend radius ratio or [force Bend reliefs/Auto] <Auto>:

3. Do one of the following:

   - Press Enter or right click to accept the default bend radius ratio.
   - Type a value at the command prompt, then press Enter.

   The relief is created.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Sheetmetals/Bend&Corner_Relief/smbrelief1/32.png)) Selected bend faces	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Sheetmetals/Bend&Corner_Relief/smbrelief1/32.png)) Created reliefs

NOTE

- The selected items can be a 3D solid with sheet metal features, a face or a hard edge of the solid.
- To create a relief locally, you can select one or two hard edges, one or two bends or a bend and a hard edge.
- If a 3D solid is selected the command recognizes the places where corner or bend reliefs are needed and creates them automatically.
- If two hard edges or two bends or a bend and a hard edge at a corner are selected, the command tries to create a corner relief.

NOTES

- A **Rip** relief has a very small width of 0.001 * thickness. If you need to remove the relief completely, change its width to 0. This operation removes both the bend relief geometry and the bend relief feature.
- **Smooth**, **Round** and **Rip** bend relief types can be applied only if the current bend relief type is **Rectangular**.

Using the ribbon, menu or toolbar

1. To choose the desired relief type, do one of the following:
   - Click the **Relief** drop-down button in **Sheet Metal / Modify** panel on the ribbon and select **Switch to Rectangular**, **Switch to Smooth**, **Switch to Round** or **Switch to Rip**.
   - Click a tool button on the **SM Relief** toolbar: **Switch to Rectangular Relief** (), **Switch to Smooth Relief** (), **Switch to Round Relief** () or **Switch to Rip Relief** ().
   - Choose the relief type in the **Sheet Metal / Relief** menu.

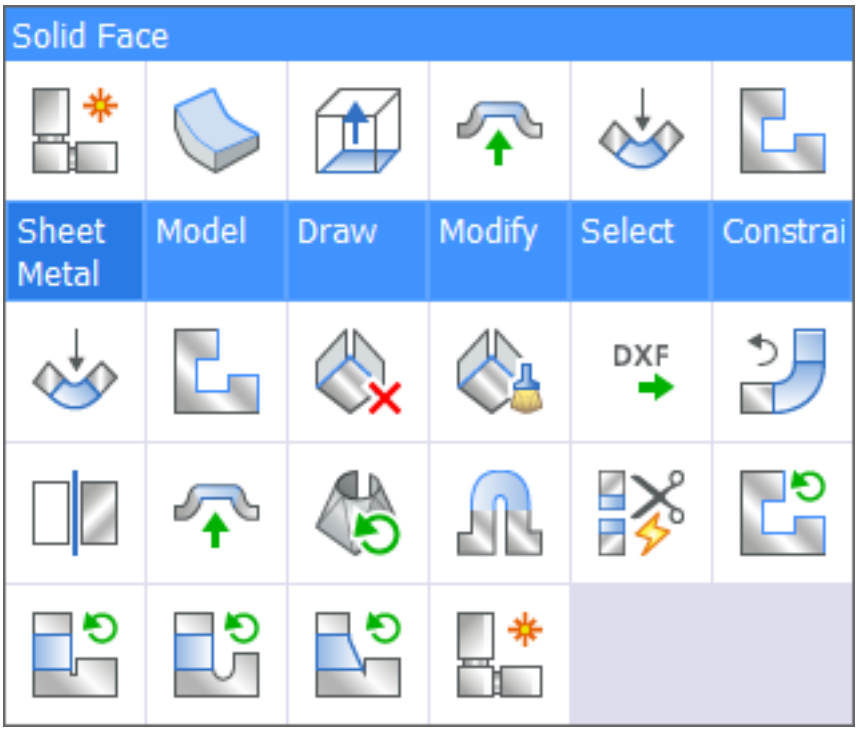
   You are prompted: Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/re EXtension/Entire model] <Entire model>:
2. Do one of the following:
   - Press Enter to apply the selected relief type to the entire model.
   - Click a neighboring flange or bend or a face of the bend relief.

   The bend relief is converted.
3. If the command has been called from the Ribbon, the command repeats the main prompt until you press the ESC key to stop.

Using the Quad

1. Hover over a neighboring flange or bend or a face of the bend relief. If the **Select Faces** option of the SELECTIONMODES (1.13.11) system variable is selected, the flange or face highlights; otherwise hold the CTRL key down when hovering until the face highlights.
2. Select the desired relief type in the**Sheet Metal** tab in the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsys

Figure 3.399



Switching the Corner Relief Type

The corner relief type can be switched between three types:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e2448/images/staticbricsyscom/help/Sheet Metal/Bend2448/images/staticbricsyscom/help/Sheet Metal/Bend type.png})
Rectangular	Circular	V-type

Using the ribbon, menu or toolbar:

1. To choose the desired relief type, do one of the following:
   - Click the **Relief** drop-down button in **Sheet Metal / Modify** panel on the ribbon and select a relief type.
   - Click a tool button on the **SM Relief** toolbar.
   - Choose the relief type in the **Sheet Metal / Relief** menu.

   You are prompted: Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/re EXtension/Entire model] <Entire model>:
2. Do one of the following:
   - Press Enter to apply the selected relief type to the entire model.
   - Click a neighboring flange or bend or a face of the corner relief.

   The corner relief is converted.
3. If the command has been called from the Ribbon, the command repeats the main prompt until you press the ESC key to stop.

Using the Quad:

1. Make sure the **Select Faces** option of the SELECTIONMODES (1.13.11) system variable is selected.
2. Hover over a neighboring flange or bend or a face of the corner relief.
3. Select the desired relief type in **Sheet Metal** command group in the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360018233774/mceclip7.png})

Figure 3.400

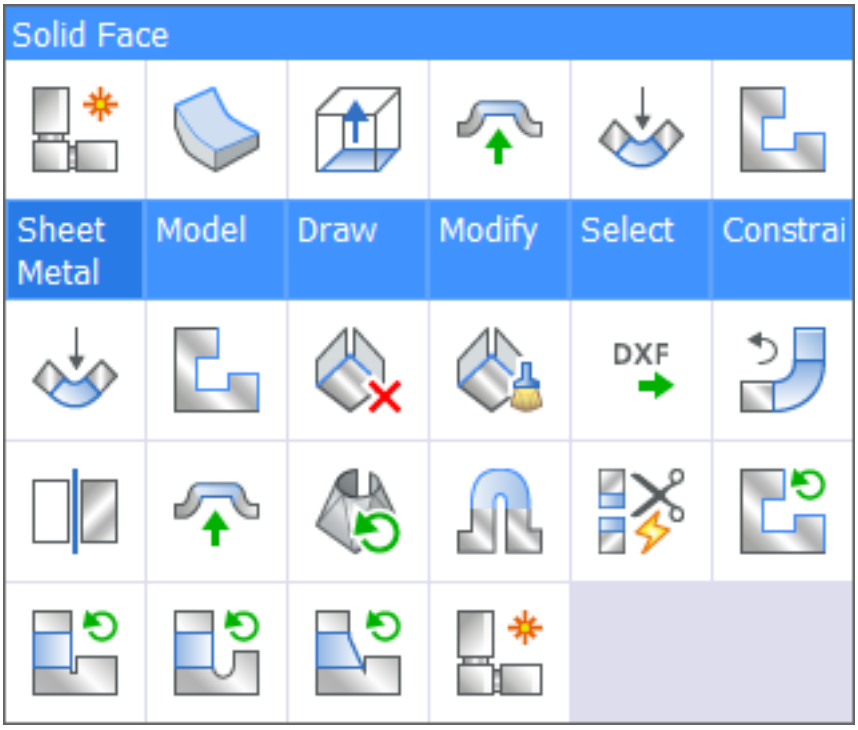


Figure 3.401

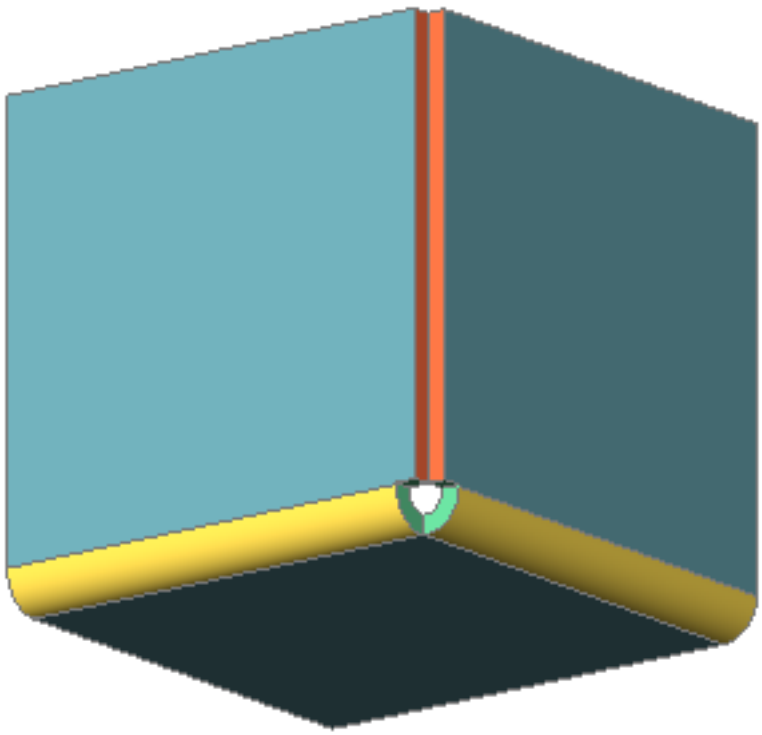


Figure 3.402

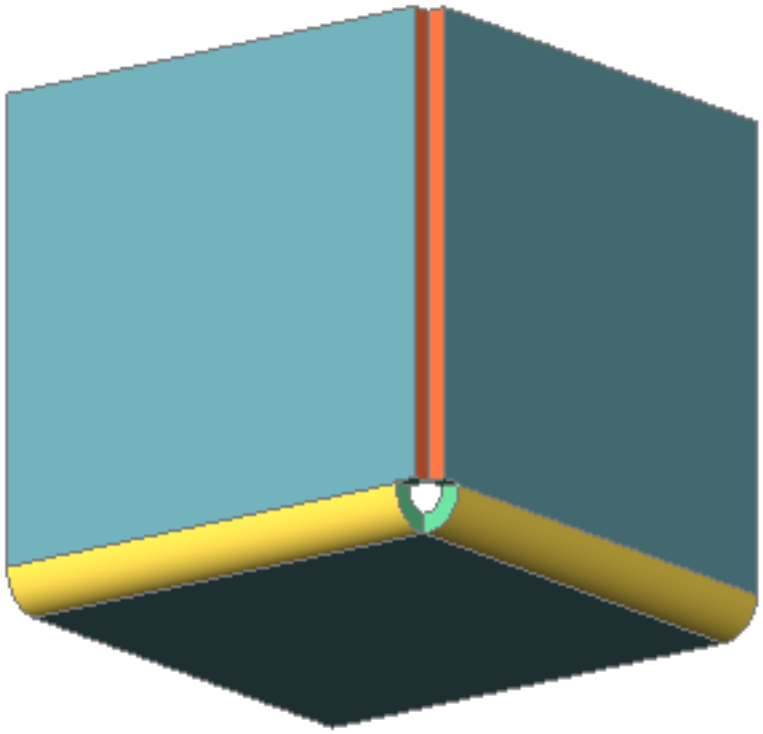


Figure 3.403

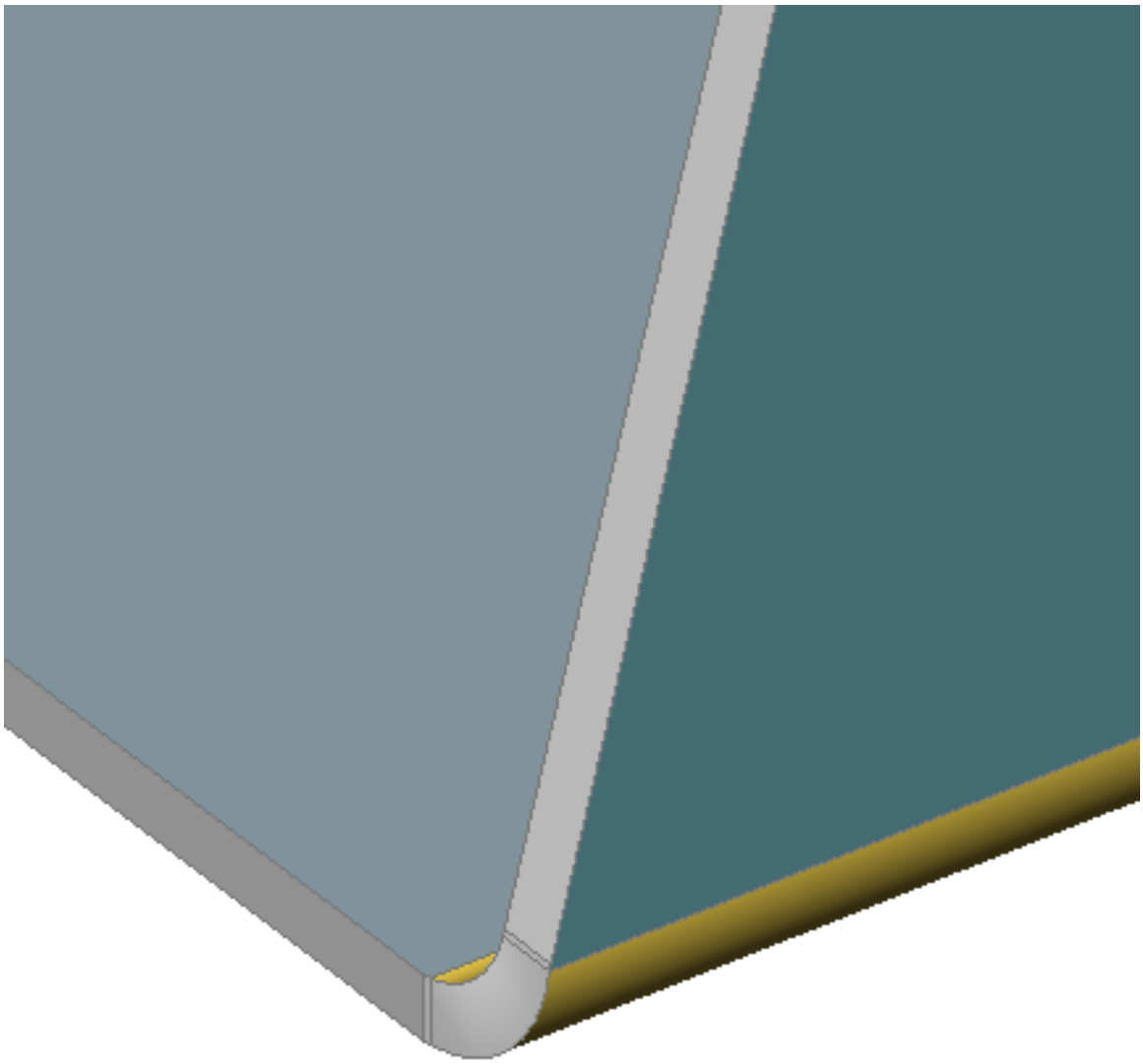


Figure 3.404

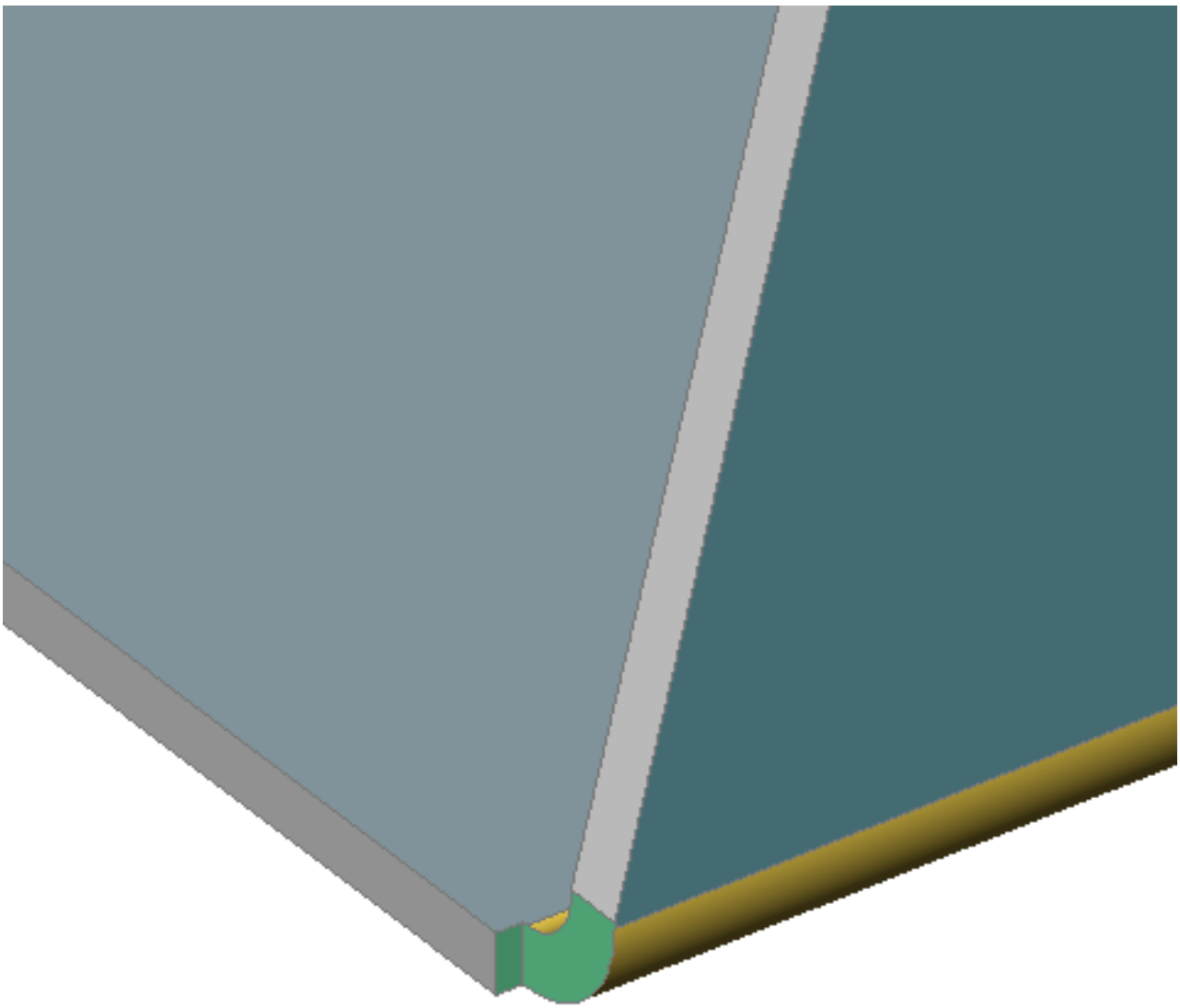


Figure 3.405



Figure 3.406



Figure 3.407

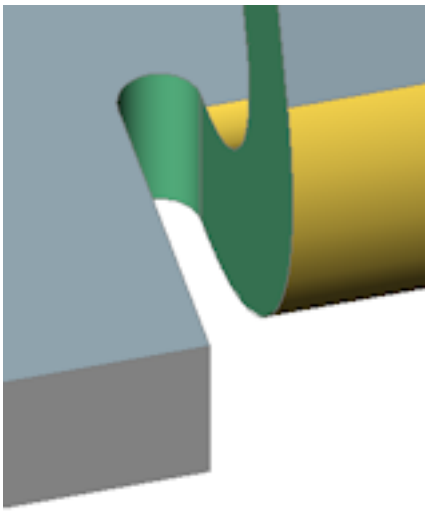


Figure 3.408

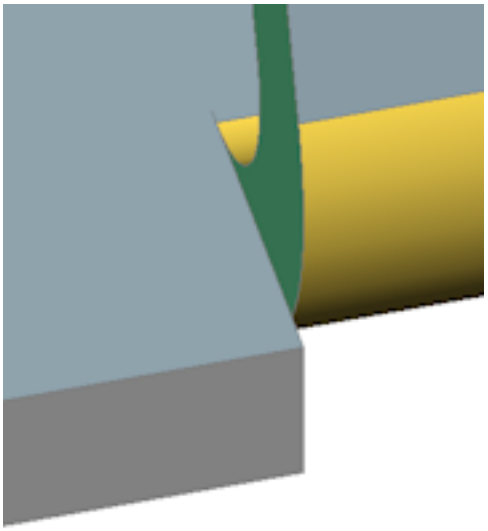


Figure 3.409

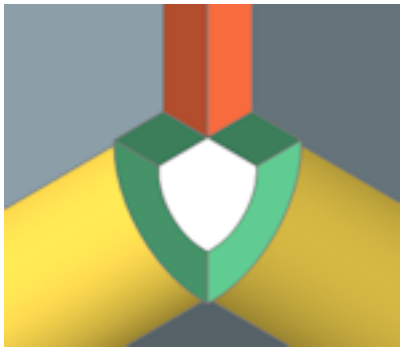


Figure 3.410

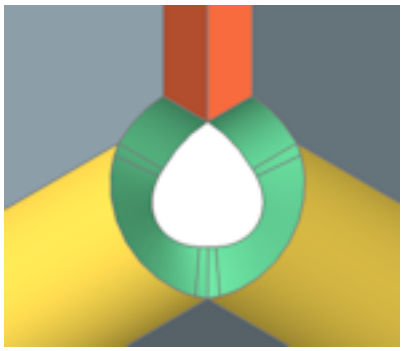
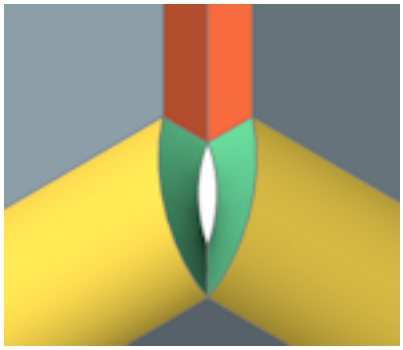


Figure 3.411



3.8.14 Converting to Sheet Metal and Repairing

Command: SMCONVERT (6.28.4), SMREPAIR (6.28.24)

To convert 3D geometry to a valid Sheet Metal part

A sheet metal part can be created from any 3D solid geometry, whether created in BricsCAD or imported from another CAD system. However, the geometry might not be a valid sheet metal model: missing bends and reliefs or thickness faces which are not orthogonal to the flange faces. BricsCAD Sheet Metal contains a dedicated toolset to convert such geometry to a valid sheet metal part and add all missing sheet metal features.

1. Do one of the following:

   - Click the **Convert to Sheet Metal** tool button ( ) on the **Sheet Metal**toolbar.
   - Click the **Convert to Sheet Metal** tool button ( ) on the **Sheet Metal** ribbon bar.
   - Choose **Convert to Sheet Metal** in the **Sheet Metal** menu.

   You are prompted: Select 3D solids or [Entire model] <Entire model>:

2. Select the 3D solid to convert.
3. **You are prompted:** Select 3D solids or [Entire model] <Entire model>:
4. Press Enter to convert the 3D solid to a sheet metal part.

The properties of the sheet metal part display in the Mechanical Browser for Sheet Metal (3.8.5).

NOTE

Converted bodies can be unfolded or exported to .dxf or .osm file formats for processing by a CAM system. In some cases it might be necessary to fix converted bodies before unfolding/exporting:

- Create missing reliefs:SmReliefCreate (6.28.22)command.
- Convert hard edges to bends or junctions:SmBendCreate (6.28.2)andSmJunctionCreate (6.28.18)commands.
- Split flanges and lofted bends:SmSplit (6.28.28)command.
- Repair
- sheet metal bodies: SmRepair (6.28.24) command.

To repair a Sheet Metal part

A sheet metal part may have inaccuracies in geometry and topology or sheet metal design issues, which is often the case for imported parts which are designed in geometrical kernels different from ACIS. Examples of such problems are:

- thickness faces, which are not perpendicular to the flange/bend faces.
- lofted bends are not-tangential to the adjacent flanges/bends.
- wrong bends.
- general ACIS issues.

Such problems can prevent editing operations from being executed successfully. The recommended workflow to fix such problems is:

- Execute the DmAudit (6.12.2)command to fix general modeling issues.
- Execute the SmRepair (6.28.24)command.

To repair a Sheet Metal body

Sheet metal bodies with thickness faces, which are not perpendicular to the flange/bend faces, cannot be correctly manufactured using traditional tools.

1. Do one of the following:

   - Click the**Repair Sheet Metal**tool ( ) on the**Heal** panel of the **Sheet Metal** ribbon tab;
   - Click the**Repair Sheet Metal**tool ( ) on the **Sheet Metal** toolbar;
   - Choose**Repair Sheet Metal** in the**Sheet Metal** menu;
   - Type**smrepair** in the command bar.

   You are prompted: Select flange face or lofted bend face to repair or [switch on Local repairing/Scan for lost details]:

2. Select the face to be repaired.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/images/staticbricsyscom/help/enUS/V18/UsrGui/source/Sr The same model before ...	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/images/staticbricsyscom/help/enUS/V18/UsrGui/source/Sr ... and after applying a repair operation:

When the model has wrong bends SmRepair transforms them to regular bends:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/images/staticbricsyscom/help/enUS/V18/UsrGui/source/Sr Wrong bends, which have a non-tangential connection with the flanges.	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/Sr Correct bends after repairing.

Form features are translated to the new repaired model:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/images/staticbricsyscom/help/enUS/V18/UsrGui/source/Sr Model before repairing	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/images/staticbricsyscom/help/enUS/V18/UsrGui/source/Sr The form feature has not been changed while a hole, which is not a form feature, is orthogonalized.

To repair a Sheet Metal body with lofted bends

1. Do one of the following:

   - Click the**Repair Sheet Metal**tool ( ) on the**Heal** panel of the **Sheet Metal** ribbon tab.
   - Click the**Repair Sheet Metal**tool ( ) on the **Sheet Metal** toolbar.
   - Choose**Repair Sheet Metal** in the**Sheet Metal** menu.
   - Type**smrepair** in the command bar.

   You are prompted: Select flange face or lofted bend face to repair or [switch on Local repairing/Scan for lost details]:

2. Select a face of a flange or a lofted bend. The selected face provides the surface to be thickened.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/images/staticbricsyscom/help/enUS/V18/UsrGui/source/Sr Before: the lofted bend is not tangent to the flange. Unfolding is impossible.	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/images/staticbricsyscom/help/enUS/V18/UsrGui/source/Sr After: notice the difference in the connection of the blue lofted bend with the top flange, which is smooth now. Unfolding is possible.

Option: If the**SmRepairLoftedBendMerge**user preference is ON, adjacent lofted bends with a tangential connection are merged to a single lofted bend.

Figure 3.412

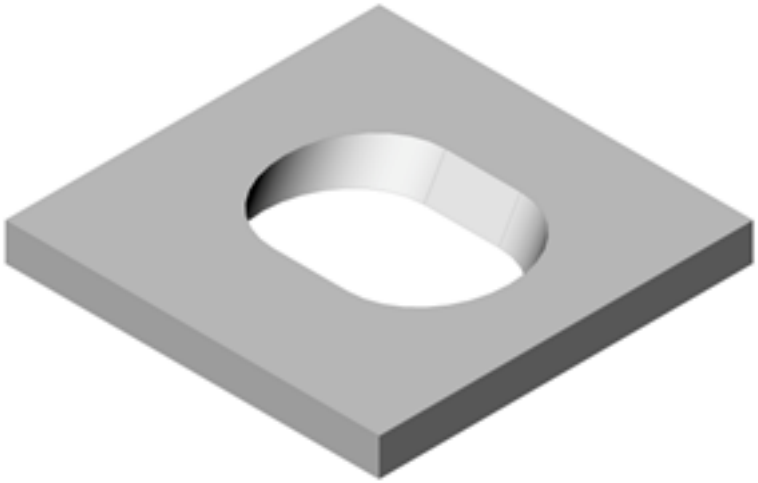


Figure 3.413

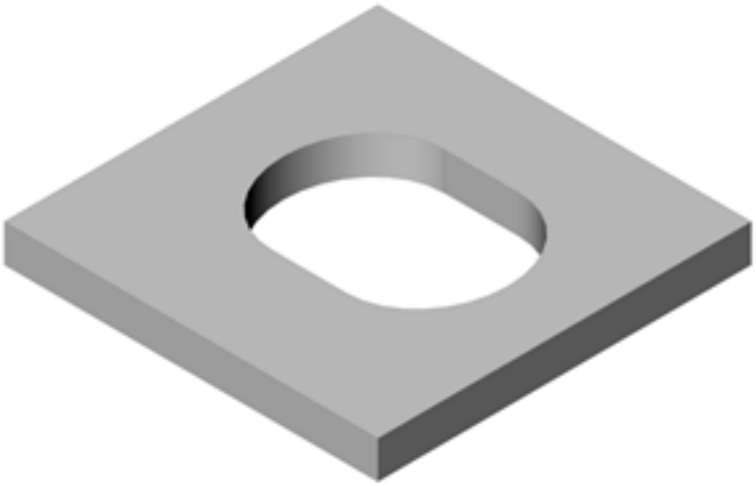


Figure 3.414

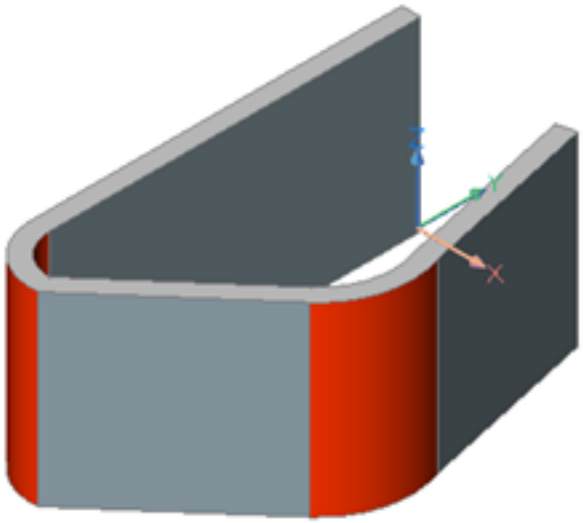


Figure 3.415

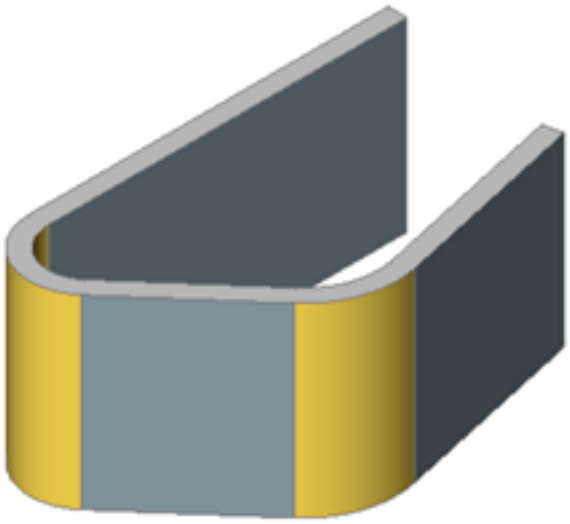


Figure 3.416

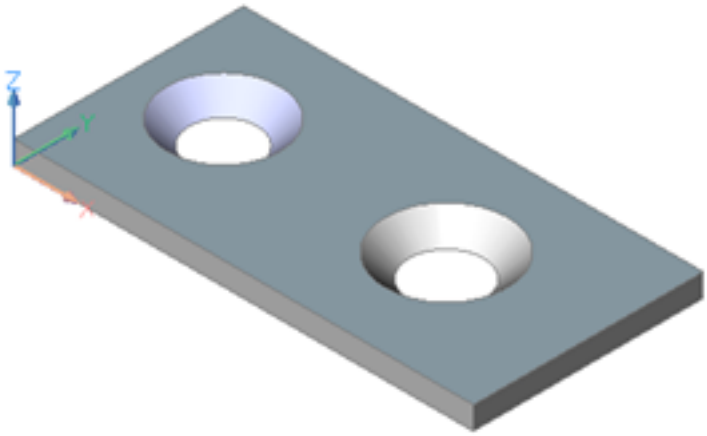


Figure 3.417

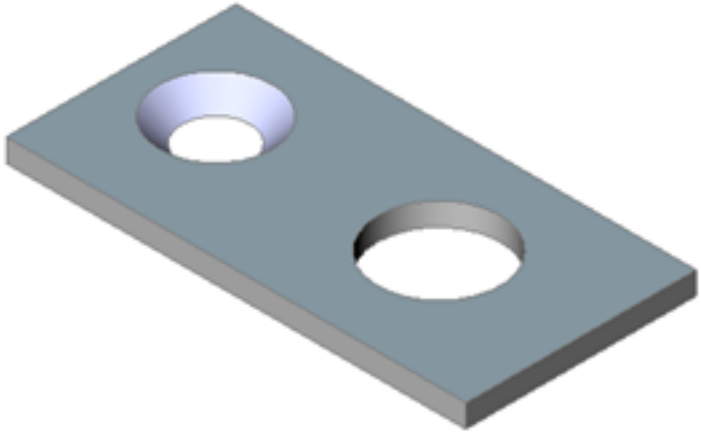


Figure 3.418

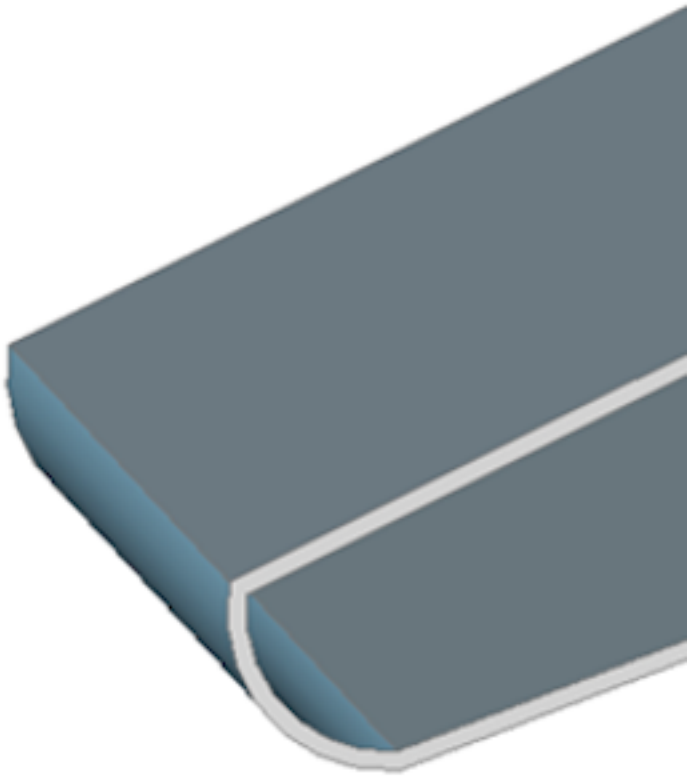
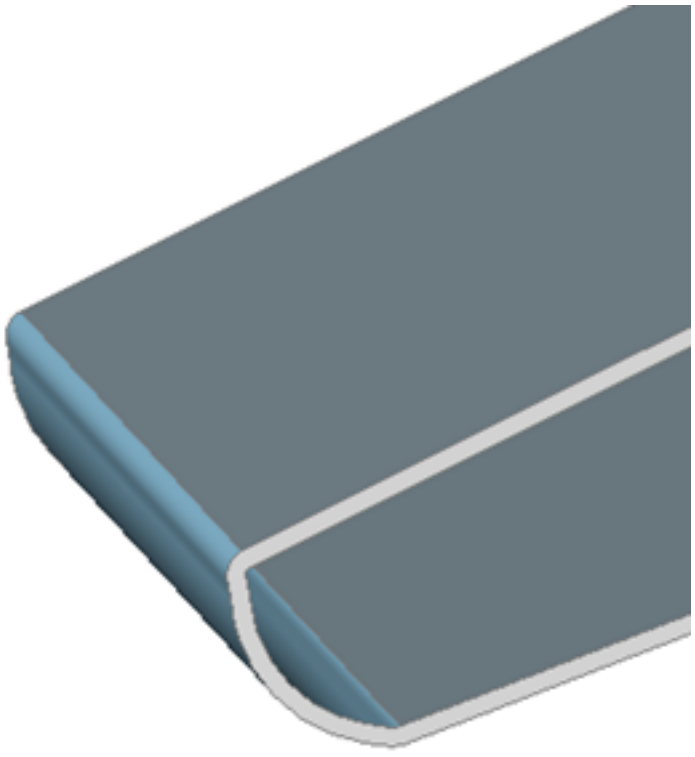


Figure 3.419



3.8.15 Unfolding Sheet Metal Parts and Exporting to CAM

Command: SMUNFOLD (6.28.29) Sheet metal parts can be designed in manydifferent ways (3.8.2). To manufacture the sheet metal part a flat pattern with bend annotations is needed, which is then used in a CAM system to program CNC machines, such as laser cutters and a press breaks. This flattening operation is called unfolding.

Unfolding a sheet metal body

TheSmUnfold (6.28.29) command unfolds sheet metal parts. BricsCAD automatically creates a solid body corresponding to the flat metal sheet needed to manufacture the sheet metal part using bending techniques. This sheet is placed on the XY-plane and is oriented along the coordinate axes similarly to the orientation of the initial body in 3D space. To change the orientation of the unfolding in OZ axis, run theSmUnfold (6.28.29) command again and select the opposite face of the initial body. Form features are unfolded according to the value of the SMDEFAULTFORMFEATUREUNFOLDMODE (1.13.11) system variable. The options are:

- **Keep** (0): allows to keep form features geometry on the unfolded representation.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/SmUnfoldFeatures3DKeep.png}) 3D model and unfolded body	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/SmUnfoldFeatures2DKeep.png}) 2D

- **Remove** (1): removes the form feature geometry from the unfolded representation.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/SmUnfoldFeatures3DRemove.png}) 3D model and unfolded body	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/SmUnfoldFeatures2DRemove.png}) 2D

- **Project** (2): generates a form feature symbol by projecting the form feature edges onto the flange plane.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/SmUnfoldFeatures3DProject.png}) 3D model and unfolded body	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/SmUnfoldFeatures2DProject.png}) 2D

- **Contour** (3): generates a form feature symbol by projecting the external contour and holes of the form feature onto the flange plane.

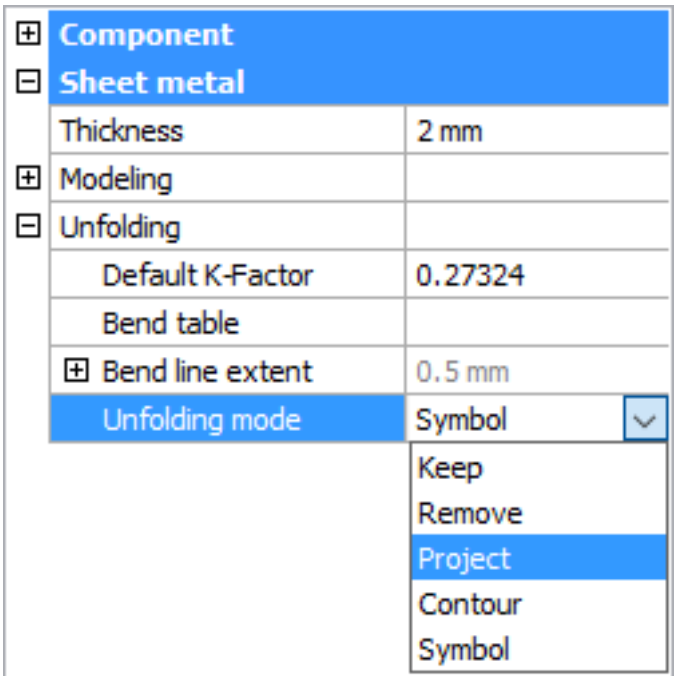
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/SmUnfoldFeatures3DContour.png}) 3D model and unfolded body	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/SmUnfoldFeatures2DContour.png}) 2D

- **Symbol** (4): creates the unfold symbol on the**UNFOLD_SYMBOL_UP** or**UNFOLD_SYMBOL_DOWN**layer depending on the direction of the form feature.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/SmUnfoldFeatures3DSymbol.png}) 3D model and unfolded body	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/SmUnfoldFeatures2DSymbol.png}) 2D

The unfold mode for the current sheet metal part can be set in Mechanical Browser (3.2.2). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmUnfoldModeOptions.png})

Figure 3.420



To unfold a sheet metal body

1. Launch the SmUnfold (6.28.29) command. **You are prompted:**Select a flange or lofted bend face to start unfolding [Associative]:
2. Select a face on sheet metal part. **You are prompted:**Select position of the unfolded body:
3. Specify a point. **You are prompted:**Validate the unfolded body and select an option [save 2D geometry/save 3D geometry/Optimize bend annotations/Keep] <Keep>
4. Do one of the following:
   - Save the unfolded 3D solid in a separate drawing file.
   - Convert the unfolded 3D solid to a 2D drawing file (*.dwg or *.dxf)
   - Rearrange the bend annotations.
   - Keep the block with the unfolded part in the sheet metal part model.

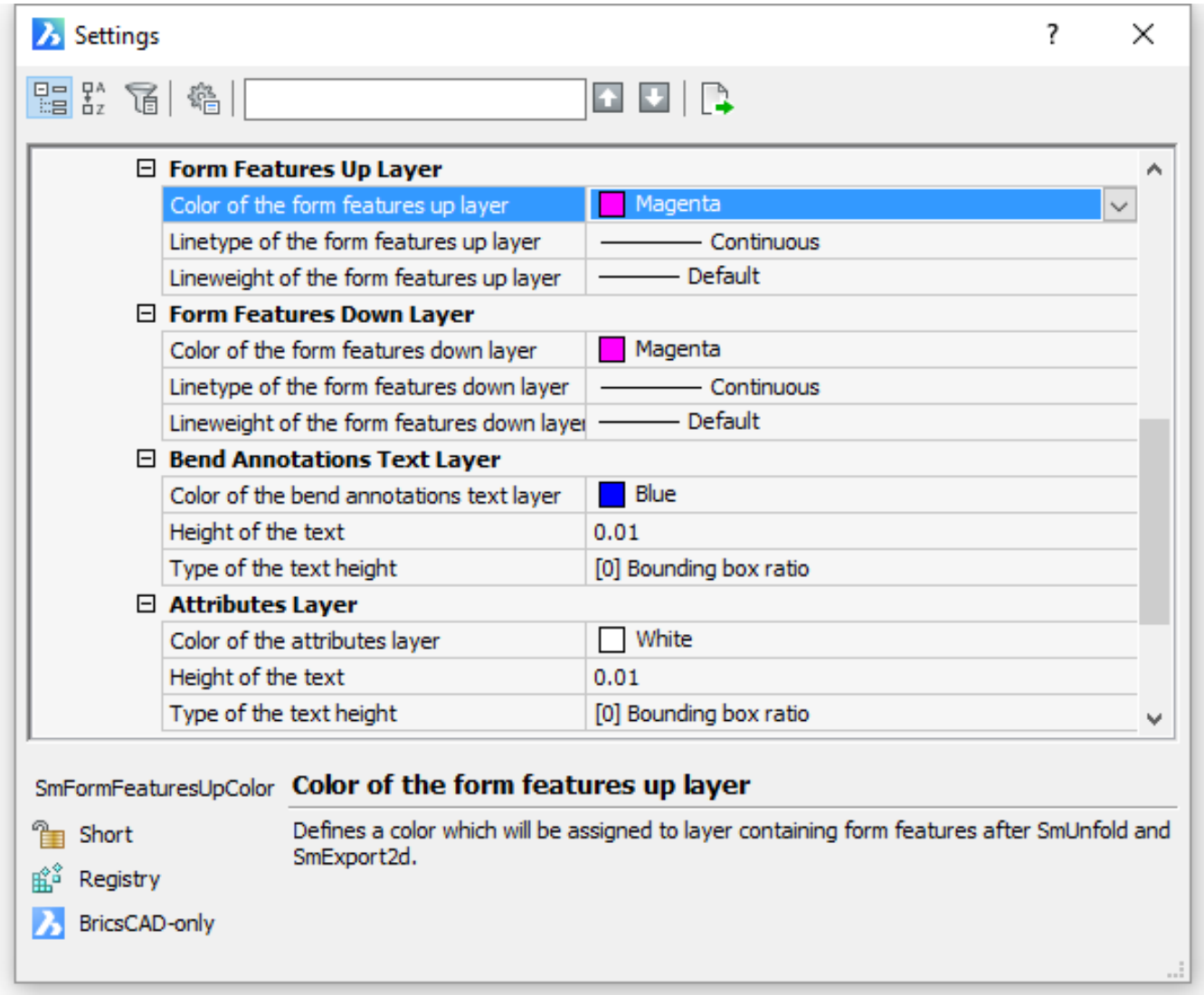
The SmUnfold command takes the deformation of the sheet metal material during bending into account. When a flat sheet of metal is bent into a 3D part with a bending tool (e.g. a press break), the material is plastically deformed, which means it is compressed at the inside of the bend and stretched at the outside. As a result the length of the part measured along its surface is different in the flat and bend states. BricsCAD is able to automatically compute the proper unfolded length of your part based on the material deformation properties. These properties can be defined by setting the value of K-Factor (3.8.7)parameter or by attaching a Bend Table (3.8.8).

Tutorial: Rework an imported model in BricsCAD Sheet Metal Click here (3.7.7) to watch

Managing layer properties of the 2D geometry

The 2D geometry generated by theSmUnfold (6.28.29),SmExport2d (6.28.7), andSmAssemblyExport (6.28.1) commands has different colors for contours, bend lines up, bend lines down, overall dimensions, form features up, form features down, bend annotations and attributes. You can set the colors differently in the **Sheet Metal/Commands/Layers for unfolding** settings group on the Settings dialog (1.13.11).Each entity type is placed on a dedicated layer of which the color is controlled through a series of user preferences: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
If the 2D geometry of a form feature was taken from alibrary (3.8.18) component, its properties are controlled by the properties of this component at the moment of insertion. There are two other modes of**Unfolding Mode** which lead to generation of 2D geometry for form features:**Project** or **Contour**, in these modes 2D geometry will be generated automatically from 3D geometry and its appearance will be controlled by **Form Features Up Layer** and **Form Features Down Layer** settings groups.

Figure 3.421


Settings
Form Features Up Layer
Color of the form features up layer
Magenta
Linetype of the form features up layer
Continuous
Lineweight of the form features up layer
Default
Form Features Down Layer
Color of the form features down layer
Magenta
Linetype of the form features down layer
Continuous
Lineweight of the form features down layer
Default
Bend Annotations Text Layer
Color of the bend annotations text layer
Blue
Height of the text
0.01
Type of the text height
[0] Bounding box ratio
Attributes Layer
Color of the attributes layer
White
Height of the text
0.01
Type of the text height
[0] Bounding box ratio
SmFormFeaturesUpColor
Color of the form features up layer
Short
Registry
BricsCAD-only
Defines a color which will be assigned to layer containing form features after SmUnfold and SmExport2d.

To export a sheet metal body

Sheet metal parts created or edited in BricsCAD can be then processed by different CAM systems (such as JETCAM or CADMAN-B) to generate NC code for cutting and bending machines. The user preference controls the **save 2D geometry** optionof the SmUnfold (6.28.29) command to customize the .dxf file to be compatible with the list of popular CAM systems. The SmExport2d (6.28.7) command exports the unfolded representation of a sheet metal body as a 2D profile in *.DXF or *.DWG file format. The SmExportOSM (6.28.8) command exports a sheet metal solid to the *.OSM (Open Sheet Metal) file format (native for the CADMAN-B CAM system).

To export an assembly with sheet metal parts to .dxf

The SmAssemblyExport (6.28.1) command searches the assembly tree for sheet metal design and exports it to a *.DXF file. As a result, a set of *.DXF files, sorted by thickness in different subfolders is created in the output folder. The command generates a HTML report which allows to control the output. Most of the sheet metal parts will be processed automatically. Poor sheet metal parts will be reported by the command to be reworked manually.

Figure 3.422

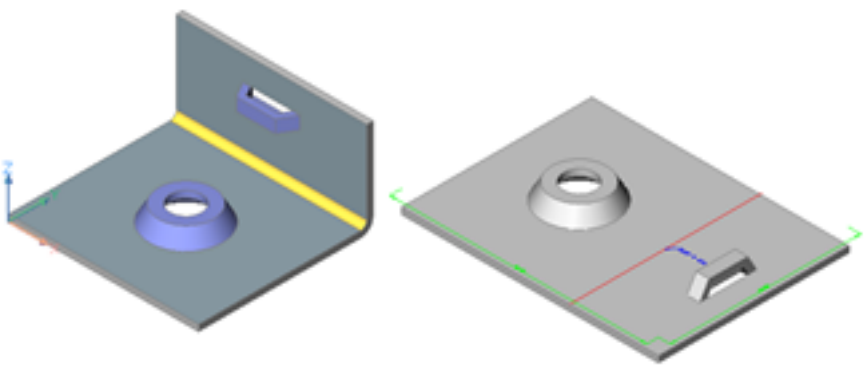


Figure 3.423

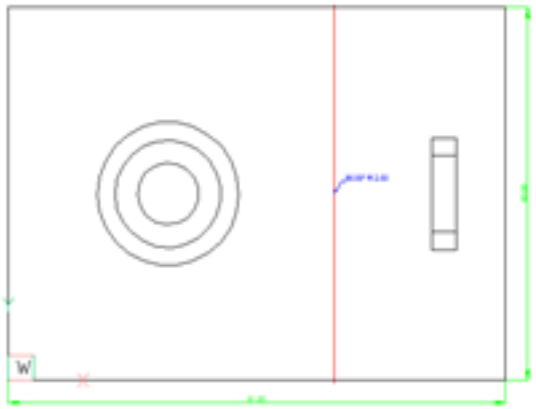


Figure 3.424

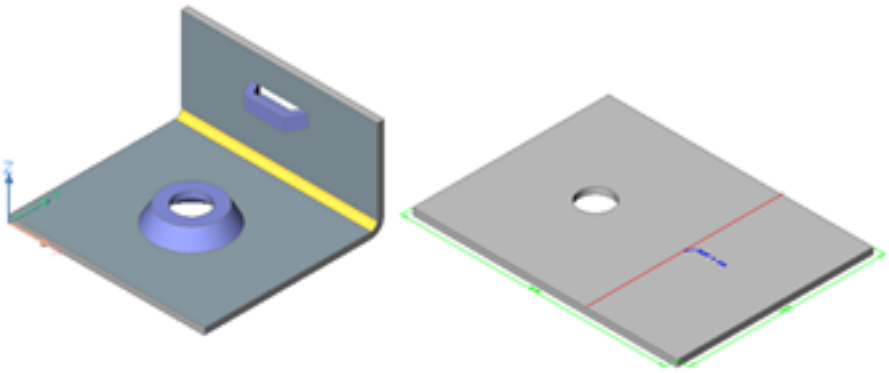


Figure 3.425

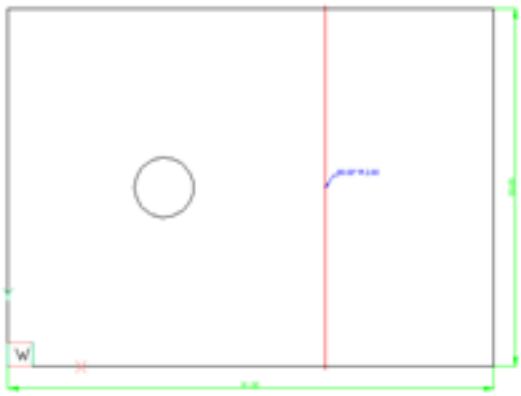


Figure 3.426

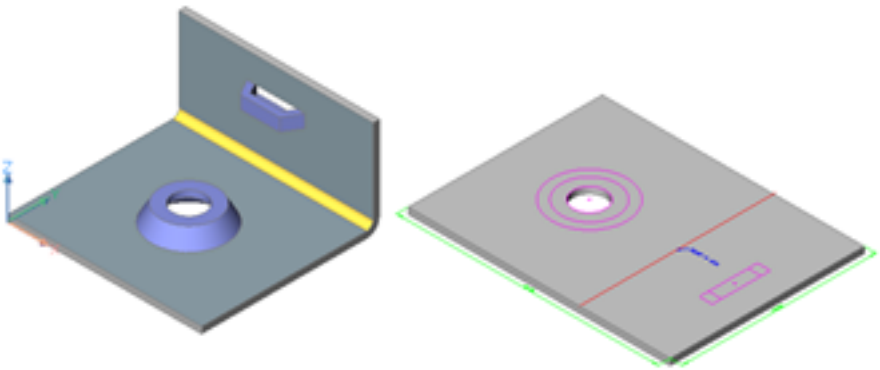


Figure 3.427

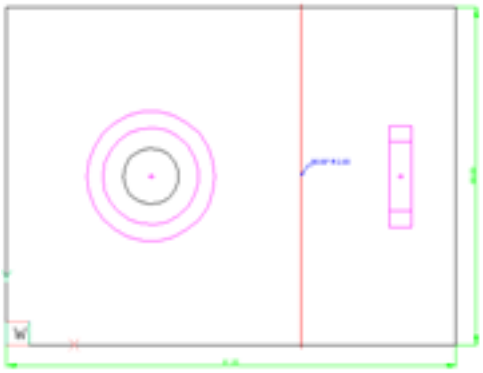


Figure 3.428

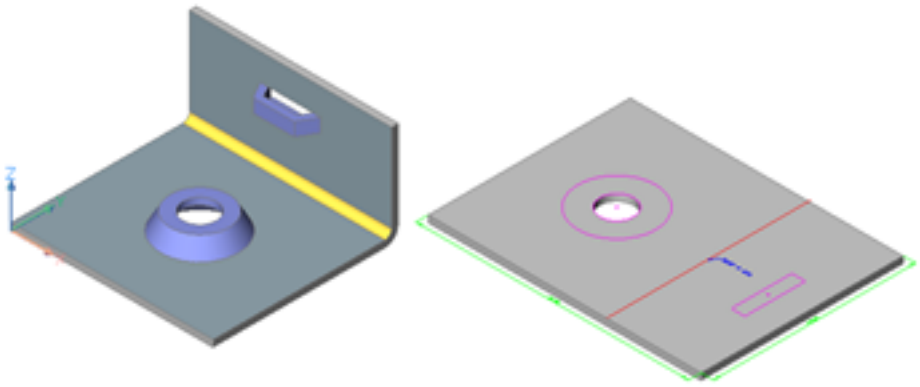


Figure 3.429

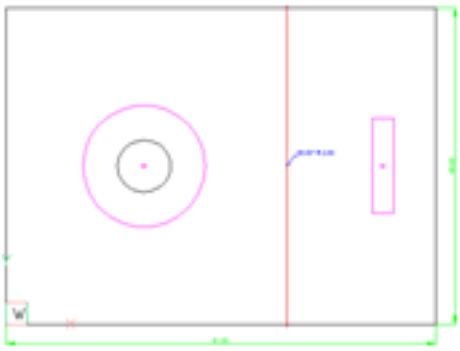


Figure 3.430

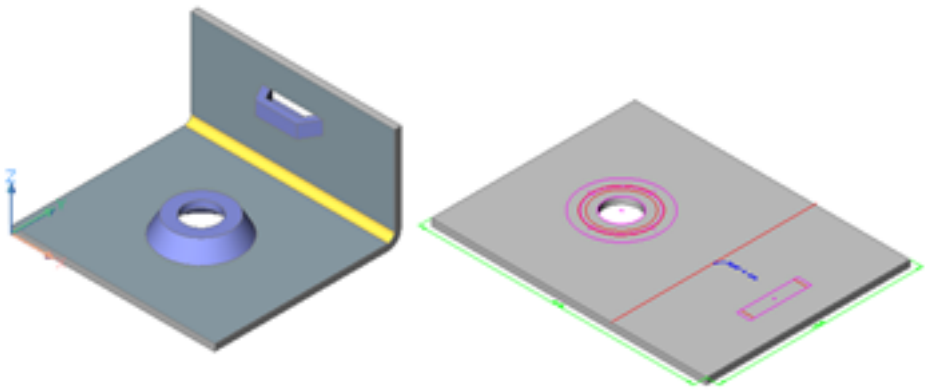


Figure 3.431

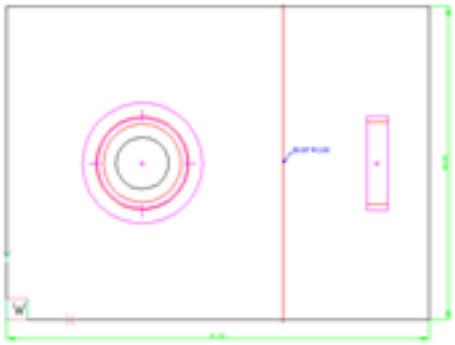


Figure 3.432



3.8.16 Smart Features

The Smart Features technology manages local intelligent update of the geometry after Sheet Metal commands are executed. The behavior is controlled through the**SmSmartFeatures** user preference. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmartFeaturesModelingMenu.png}) Two options are available (more to come in the future):

- Allow rebuilding sheet metal features:
  - corner relief is transformed into two bend relief after executingSmSplit (6.28.28) on a flange
  - unnecessary reliefs are removed
- Allow automatic edges imprint after rebuild:automatic imprinting edges for thickness faces.

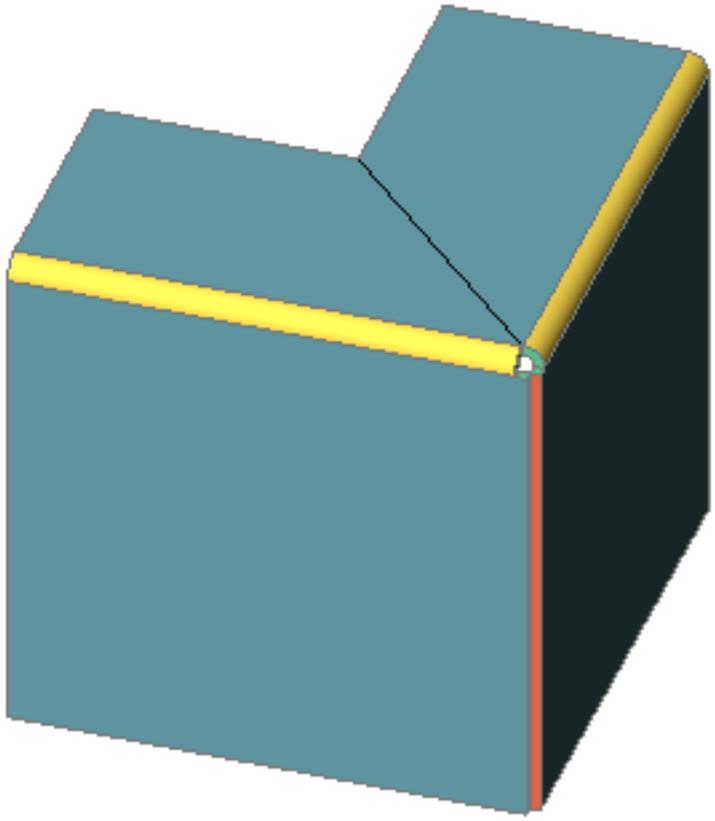
Note: Always keep Smart Features options ON, unless inconsistent behavior occurs.

Transformation of a corner relief into bend reliefs

When splitting a top flange by a line into two separate parts, the corner relief is transformed into two bend reliefs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmartFeaturesTransformati

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmartFeaturesTransformati	
Smart features create bend reliefs automatically, which can then be switched to smooth.	Without smart features the corner relief is kept and cannot be switched.

Figure 3.433

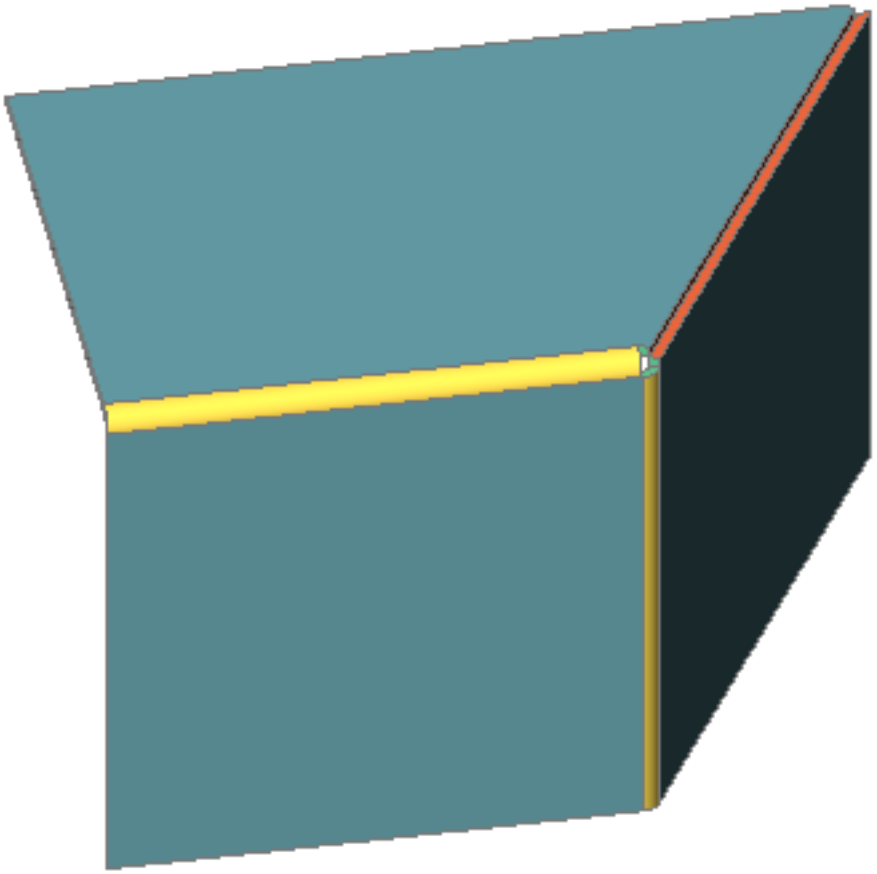


Removing unnecessary cuts

In the model belowexists a corner relief cut, which is needed to prevent the material from being stretched. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmartFeaturesRemoving1.png}) When the top bend is converted into a junction, we get to separate parts. The smart features automatically remove the now unnecessary relief.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmartFeaturesRemoving2.png}) Smart features automatically fill unnecessary cuts, thus simplifying the geometry.	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmartFeaturesRemoving2.png}) Without smart features the corner relief still exists.

Figure 3.434



Automatic imprinting of edges

Imprinted edges on thickness faces separate them into areas which can be reworked automatically, i.e. localized junctions can be converted into bends, seeBend, Lofted Bend and Junction Feature Operations (3.8.11) for more details. Without automatically imprinted edges the SmImprint (6.28.17) command is needed here. In the example below we will separate a flange using junctions.We turn 'Rebuilding sheet metal features' ON, and then use 'Automatic edges imprint after rebuild'. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmartFeatures/help/enUS/V19/Mechanical/Shee	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmartFeatures/help/enUS/V19/Mechanical/Shee
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmartFeatures/help/enUS/V19/Mechanical/Shee	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/SmartFeatures/help/enUS/V19/Mechanical/Shee
Allow automatic edges imprint after rebuild = ON Junction faces are split correctly.	Allow automatic edges imprint after rebuild= OFF Junction features overlap each other.

Figure 3.435

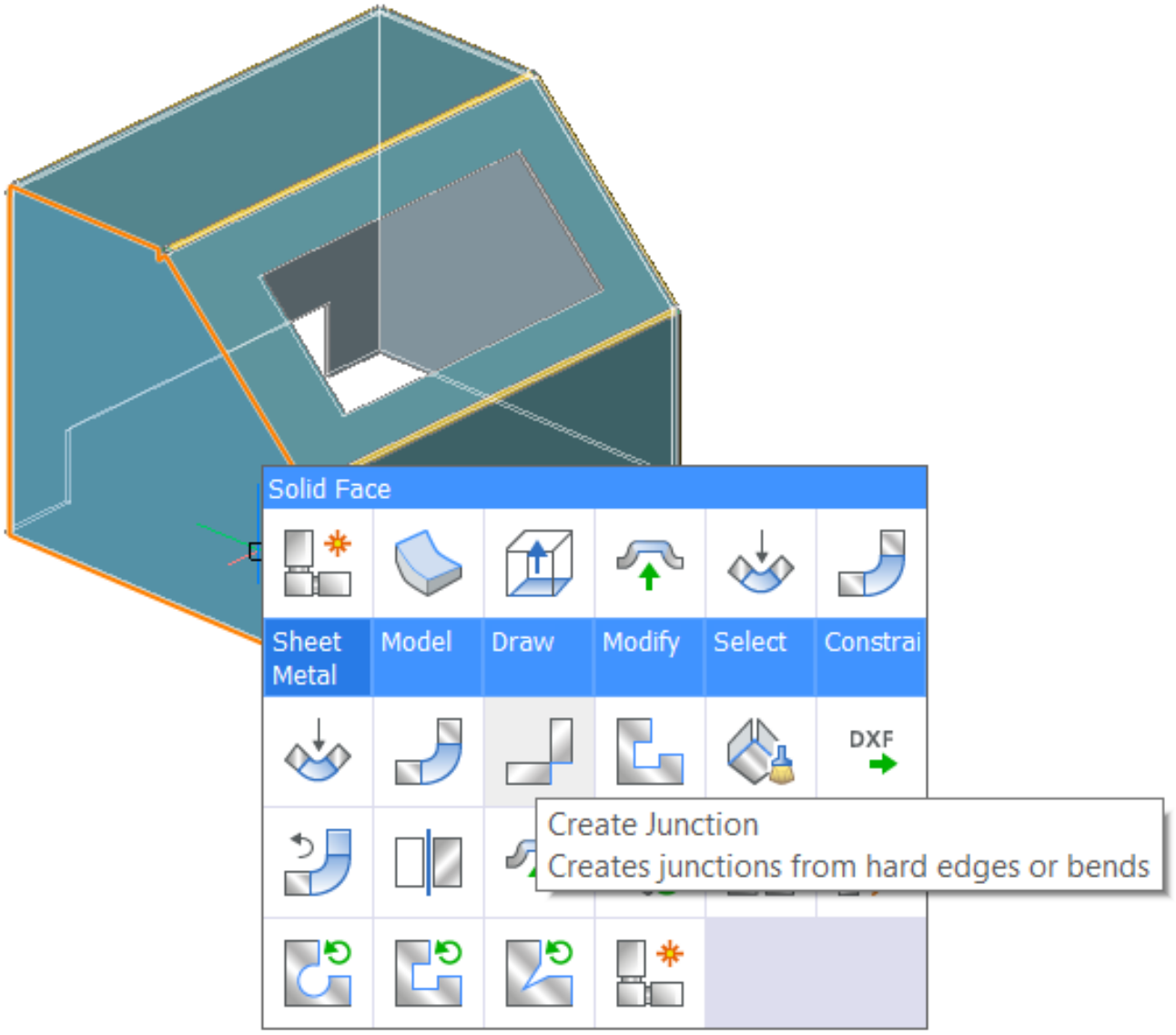

Solid Face
Sheet Metal
Model
Draw
Modify
Select
Constrai
DXF
Create Junction
Creates junctions from hard edges or bends

Figure 3.436

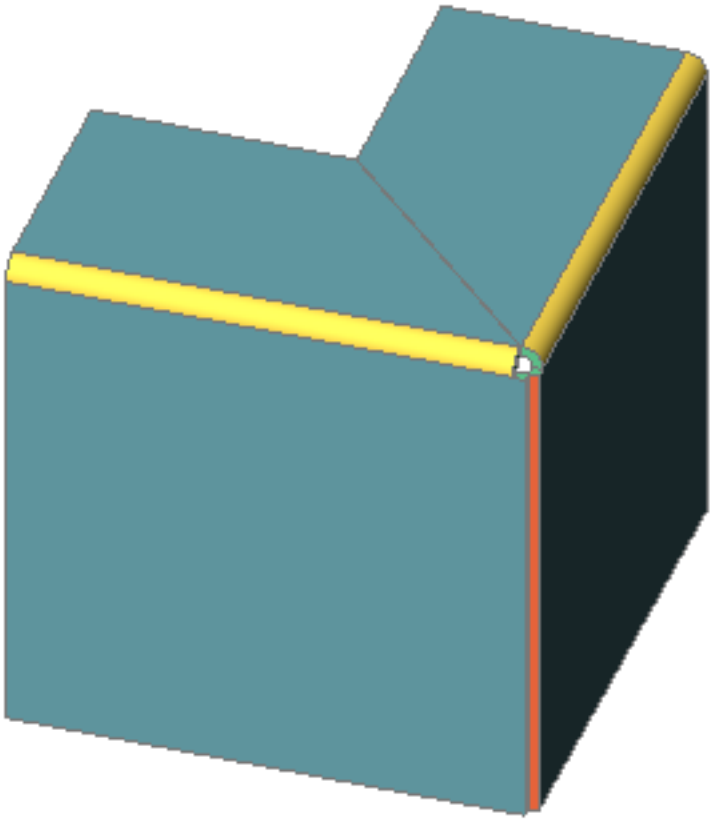


Figure 3.437

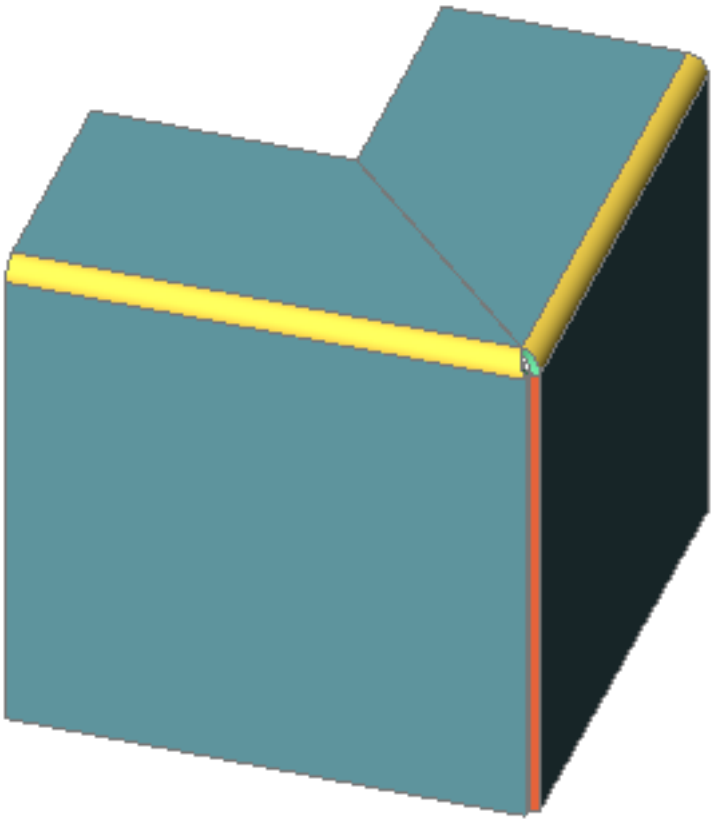


Figure 3.438

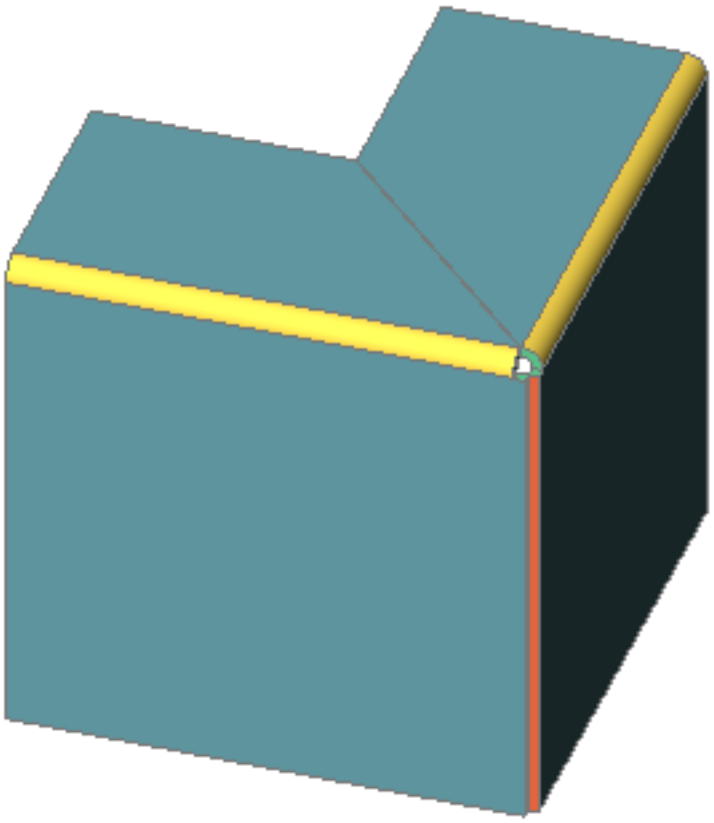


Figure 3.439

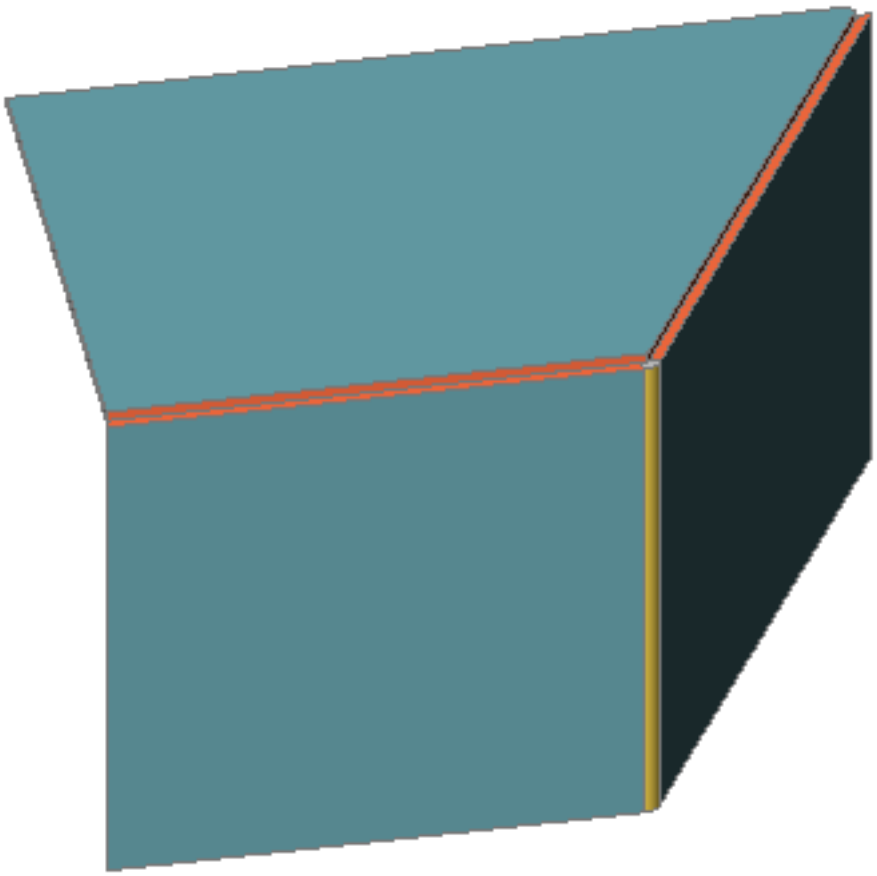


Figure 3.440

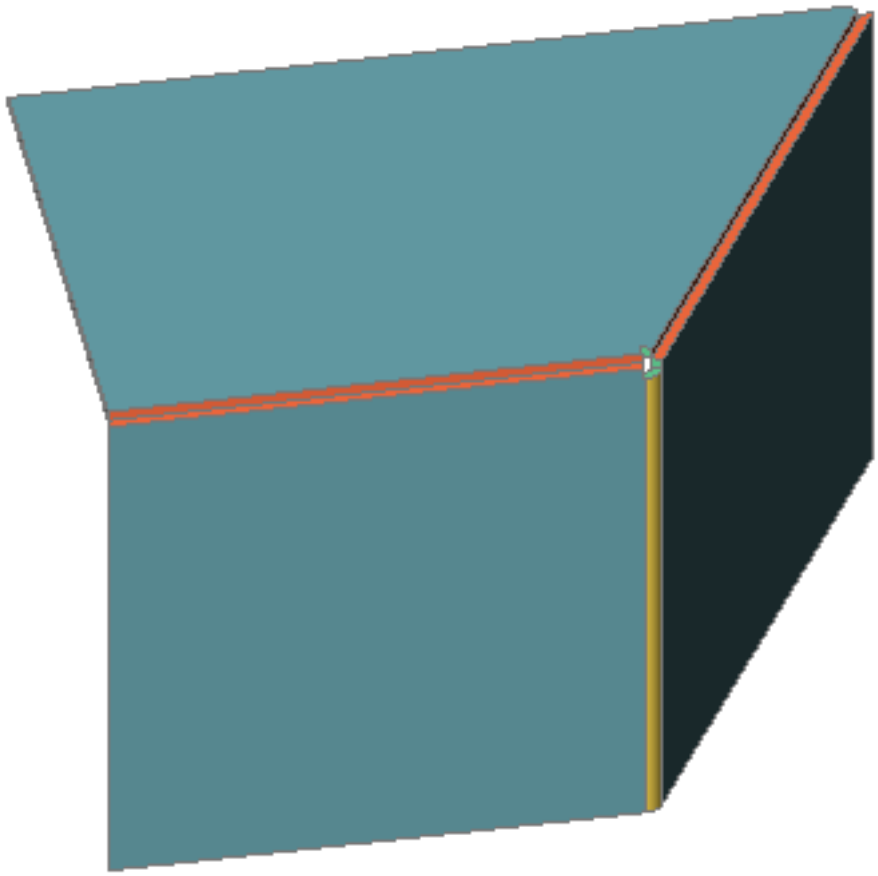


Figure 3.441

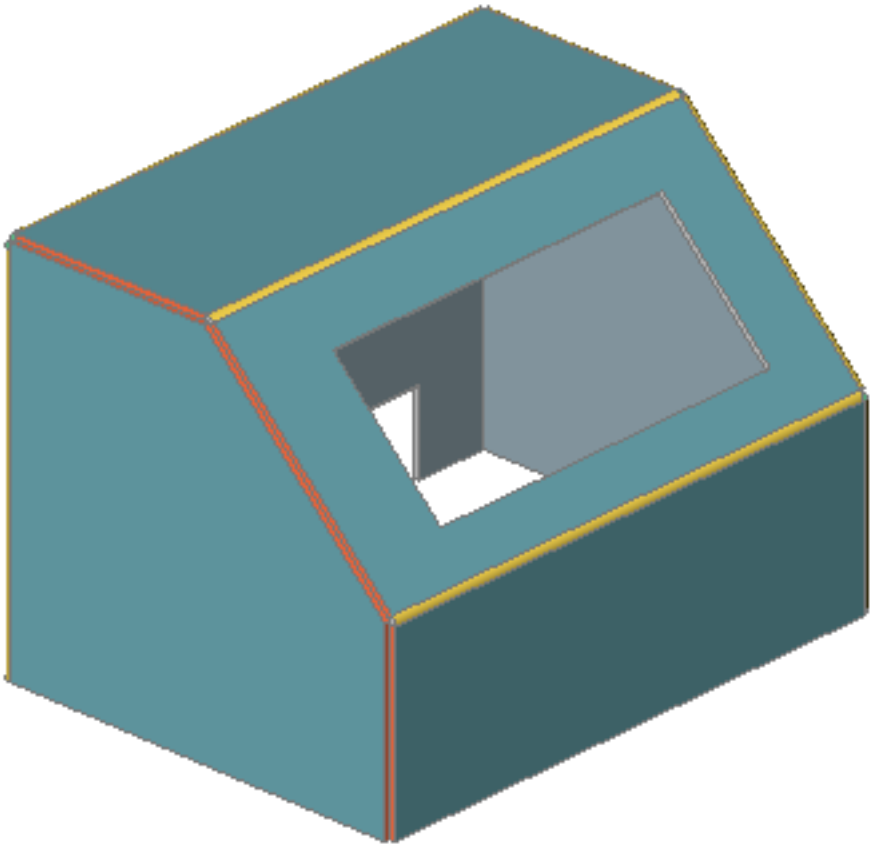


Figure 3.442

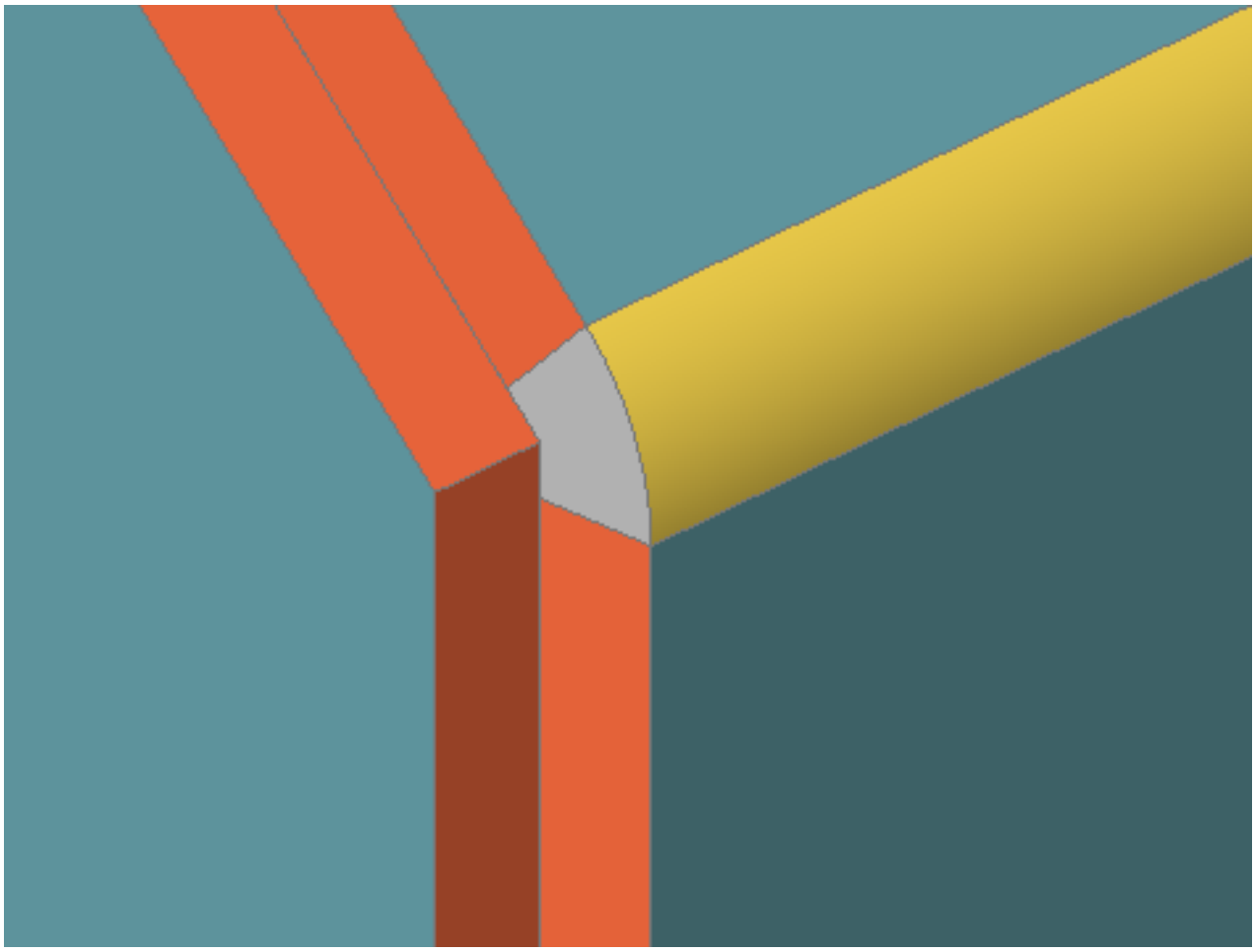


Figure 3.443

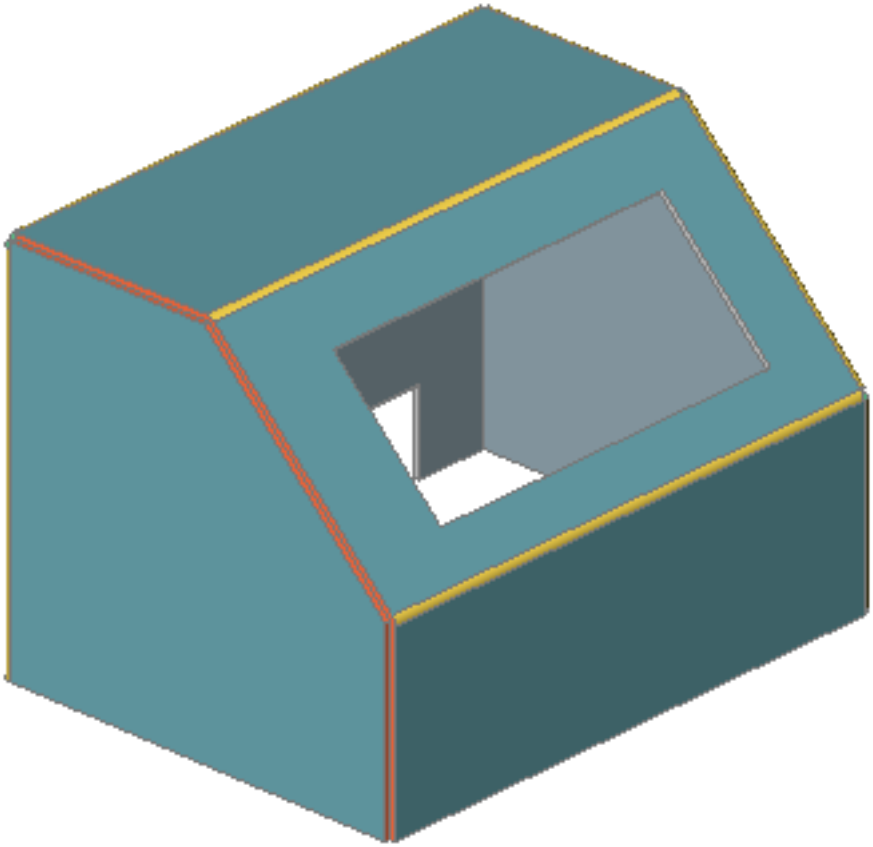
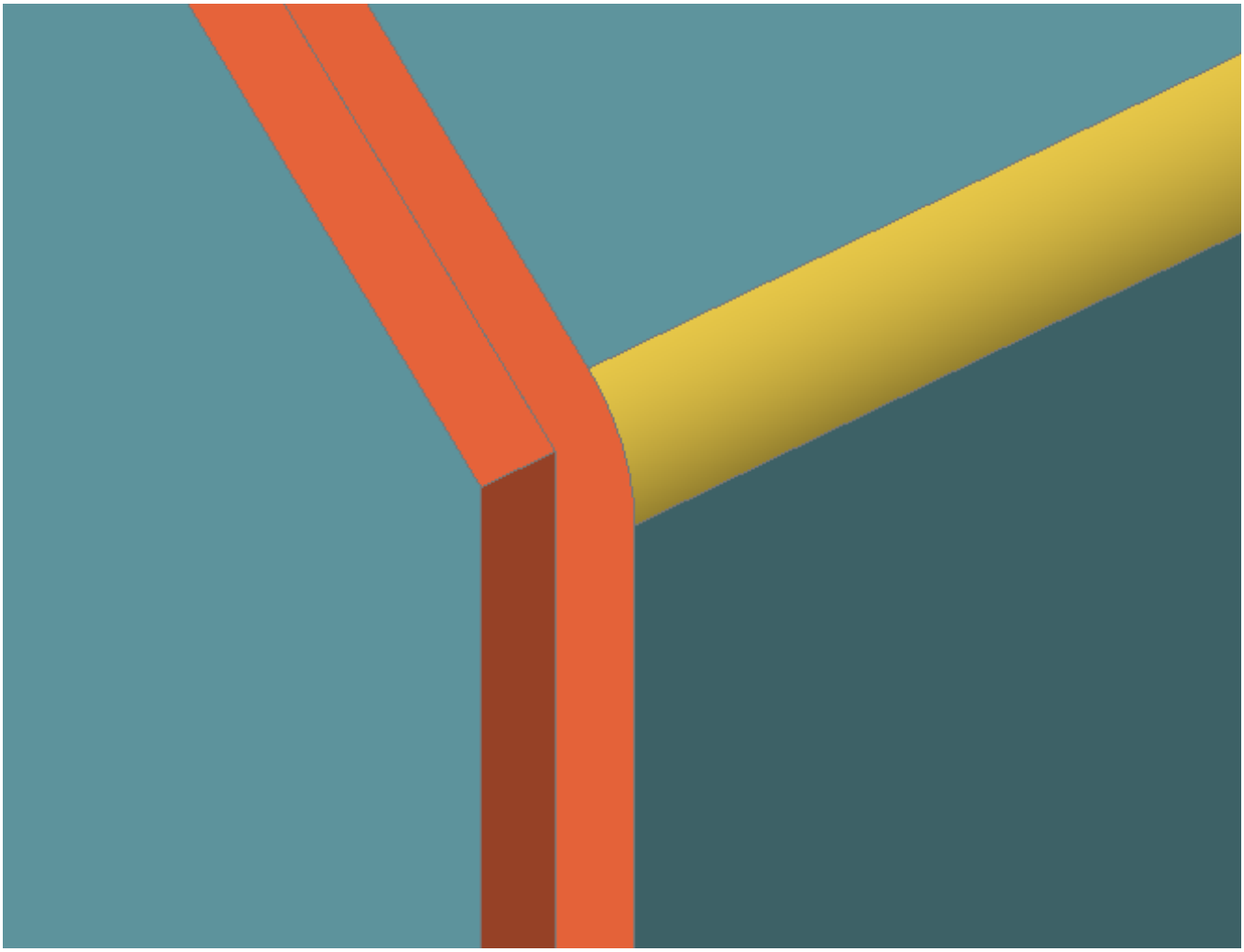


Figure 3.444



3.8.17 Associative Unfolded Representations of 3D Solids

BricsCAD offers bidirectional associativity between 3D Sheet Metal part and its flattened appearance. In this mode bothrepresentations exist in separate documents: **Original part** is stored in its native document and the **Flattened part**is stored in temporary document. Associativity is temporary and exists while both documents are opened. In order to work in associativity mode in next BricsCAD session or once the document were closed, one should repeat steps again.

What Does Associativity Mean?

Associative Unfolded allows you to the make cuts on flattened geometry and propagate the changes back to the 3D model. This helps simplify two scenarios:

- Creating complex, non-standard Bends and Corner Relief cuts.
- Avoid self-intersections in unfolding.

In these two scenarios both are easy to do in 2D rather than in 3D. In flattened mode Sheet Metal part is represented as set of separate flat solids (per-feature).Use the SmDelete (6.28.5)command to delete solid flanges and bends on flat patterns. Associativity is bidirectional - changes can be propagated from flattened parts back to 3D models and vice versa. Changes made on one of representations are propagated to the document with other representation once it becomes active.

Switching to Associativity Mode The associativity mode is turned on using theSmUnfold (6.28.29)command. Make sure there are no faces selected. From**Sheet Metal**section of theRibbon (6.26.31)select **AssociativelyUnfold Body** icon and select **Flange** face. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/She
Application will compute flattening of the given part and switch it to associative mode. Condition of success of given operation is the same as for regular unfolding.Continue working in associativity mode only if no errors or warnings appeared during the operation. Anew temporary document containing the flattened part will be created. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/AssociativeUnfoldedRepresentati
Note: there is no need to save this document, it's used to propagate editing operations from the flattened solid back to the 3D model. Use Vertical Tile to view both the 3D model and associative unfolding documents. Click **Vertical Tile** button on Toolbar (6.29.18) or Ribbon. (6.26.31) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Note: In the Mechanical Browser (3.2.2) when flattened geometry is selected, it lists features are disabled.

Figure 3.445

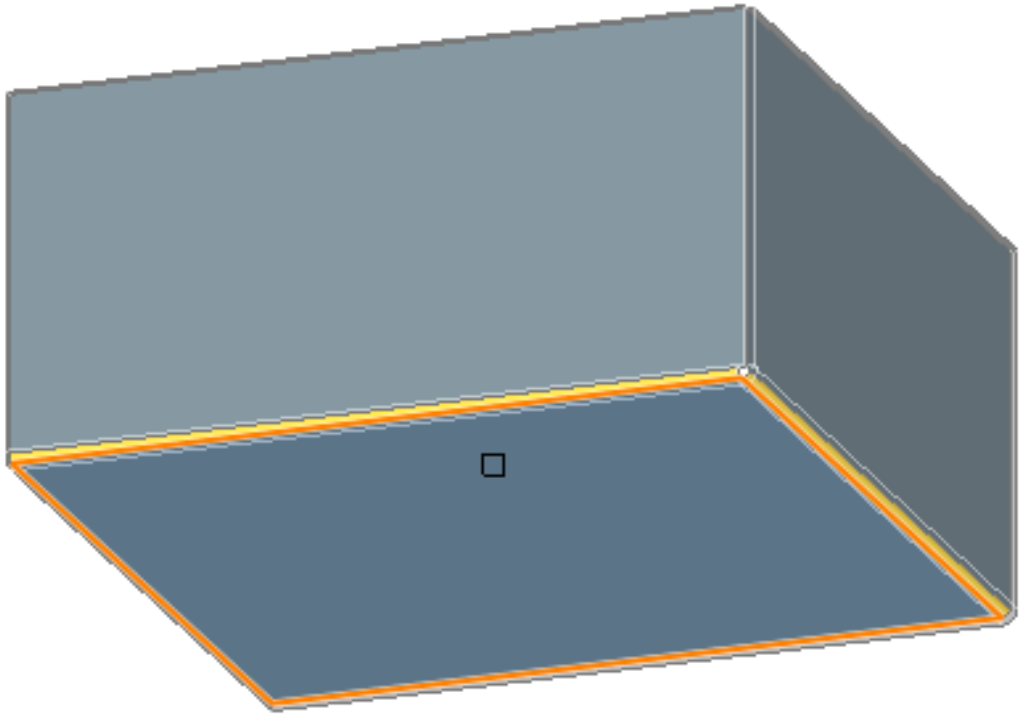


Figure 3.446

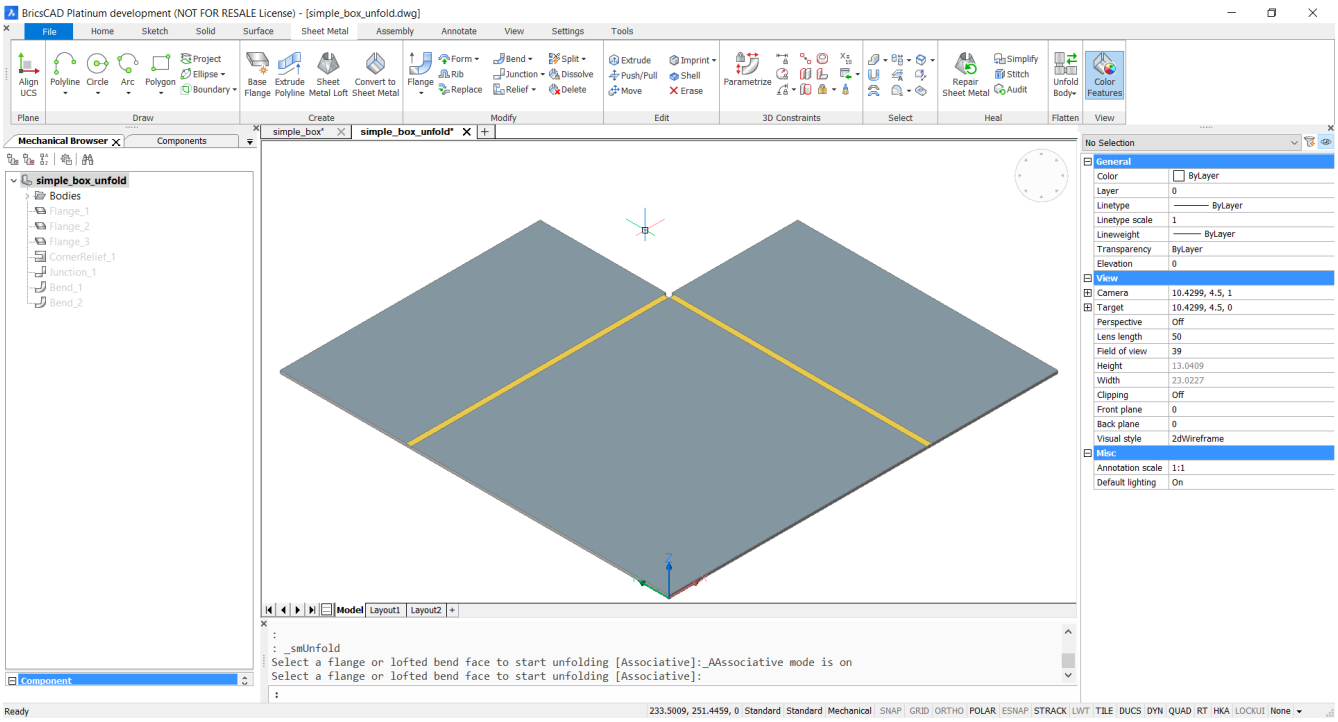
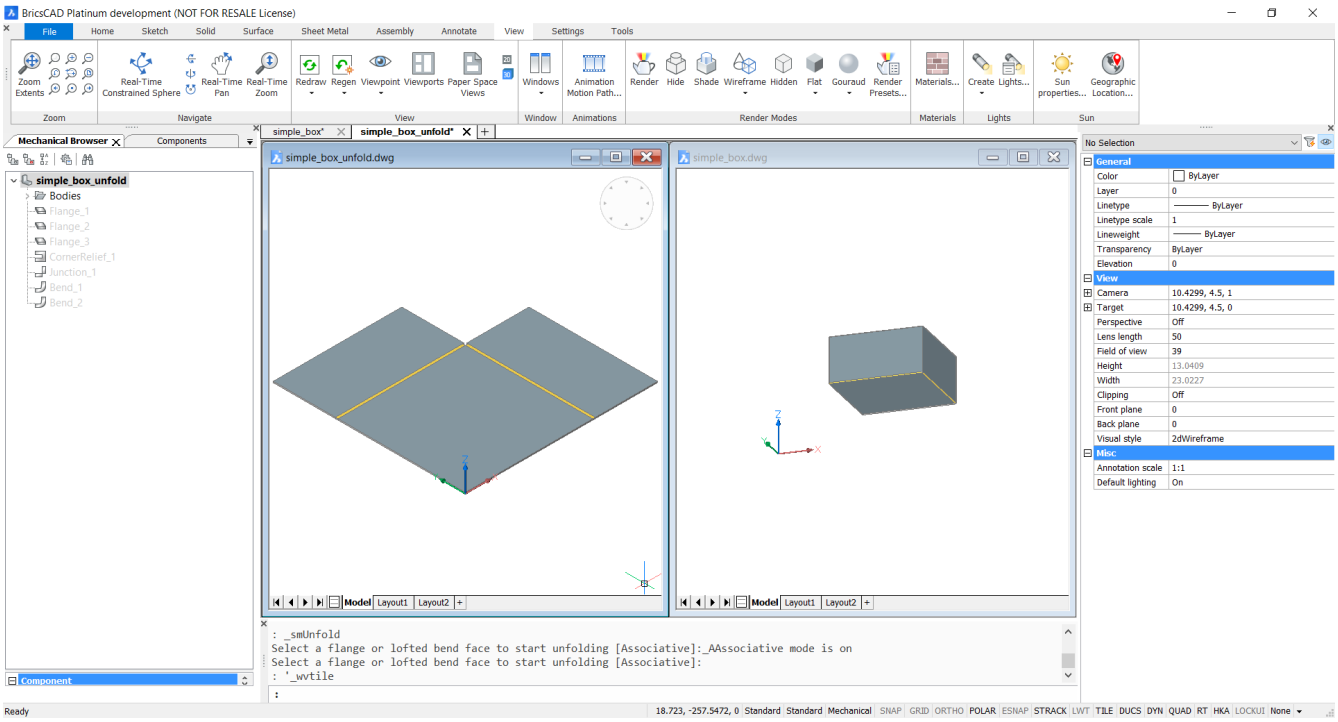


Figure 3.447



Editing the Geometry

1. Working with flattened design document, draw a circle in the corner (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco
2. Apply DmExtrude (6.12.13) command to the circle, cutting the volume. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys
3. Click on original document again to make it active. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
4. Model will be updated and the custom corner relief appears. Application writes to command line: Changes were successfully transferred to source model

Figure 3.448

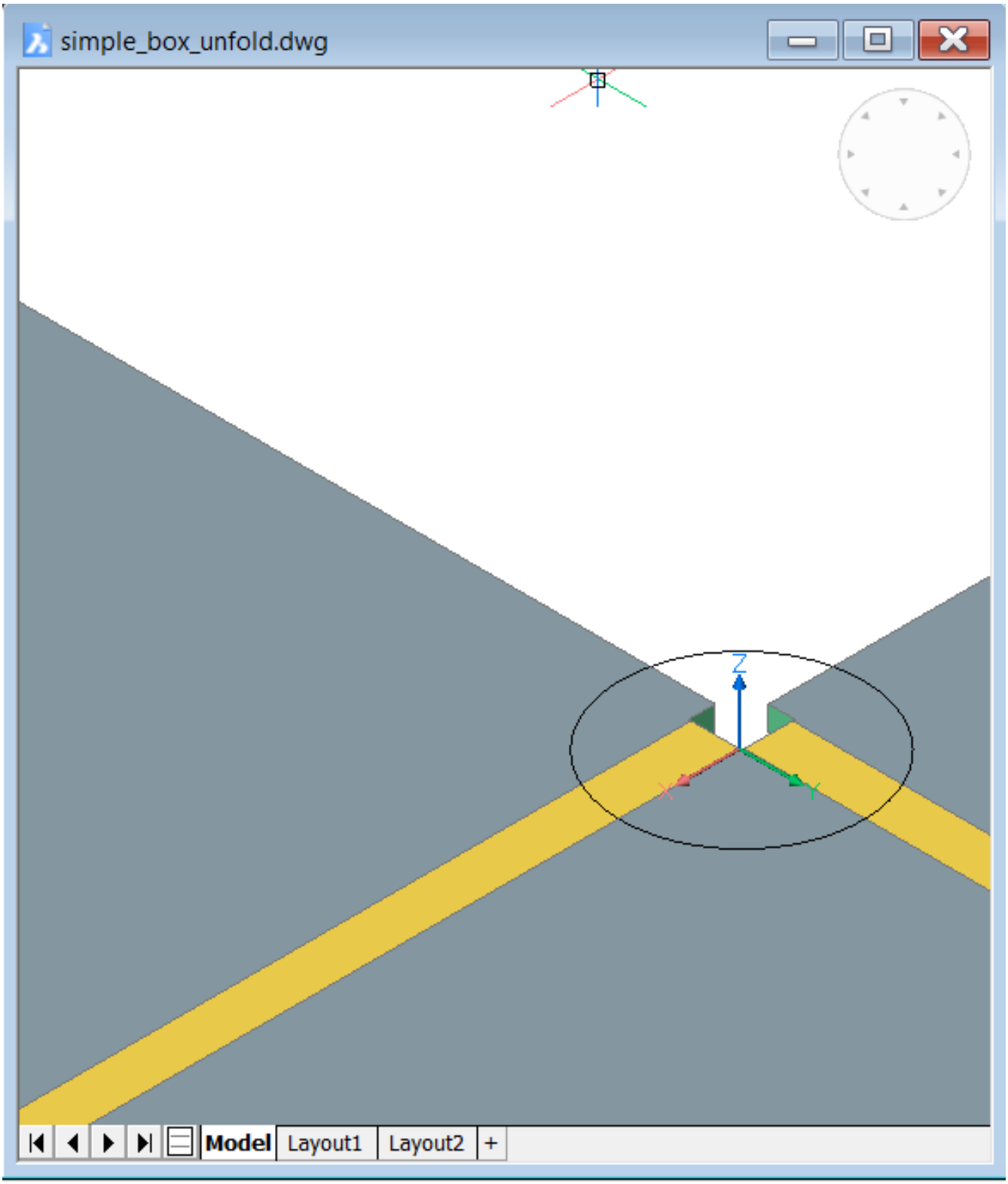


Figure 3.449

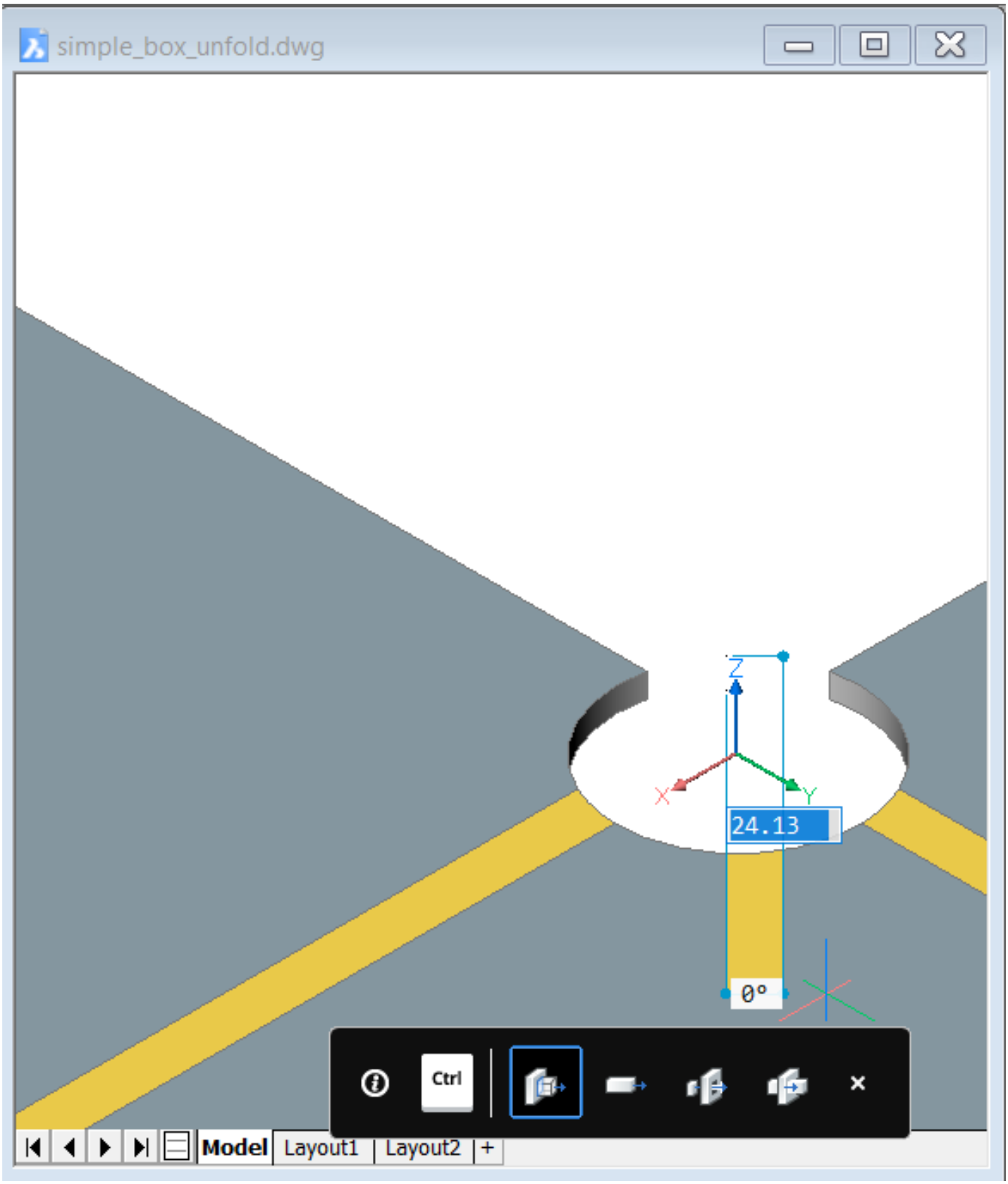
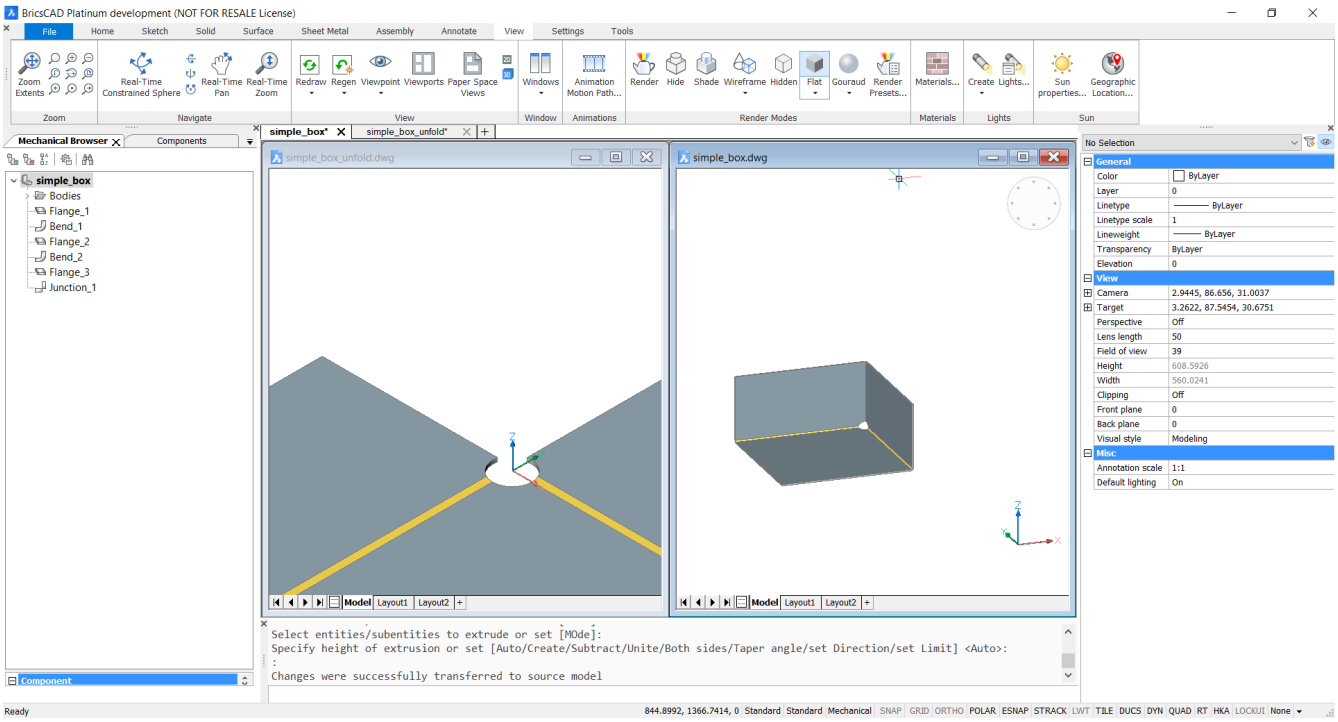


Figure 3.450



Avoiding Collisions in Flat Pattern

Another useful application of associative unfolding is fixing the so not obvious material collisions. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/AssociativeUnfoldedRepresentationAvoid1.png}) In this example, we don't know what changes to apply to the 3D model to avoid the collision when flattened. Let's rework the model with using Associative Unfolding.

1. From**Sheet Metal**section of theRibbon (6.26.31)select **AssociativelyUnfold Body** icon , select **Flange** face, unfold andclick **Vertical Tile** button on Toolbar (6.29.18) or Ribbon (6.26.31)to tile the opened documents vertically and zoom to the collision area. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mecha

2. Select boundary circular edge and offset it by 1. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/

3. Draw extra straight lines adjacent to the edge and join them. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

Figure 3.451

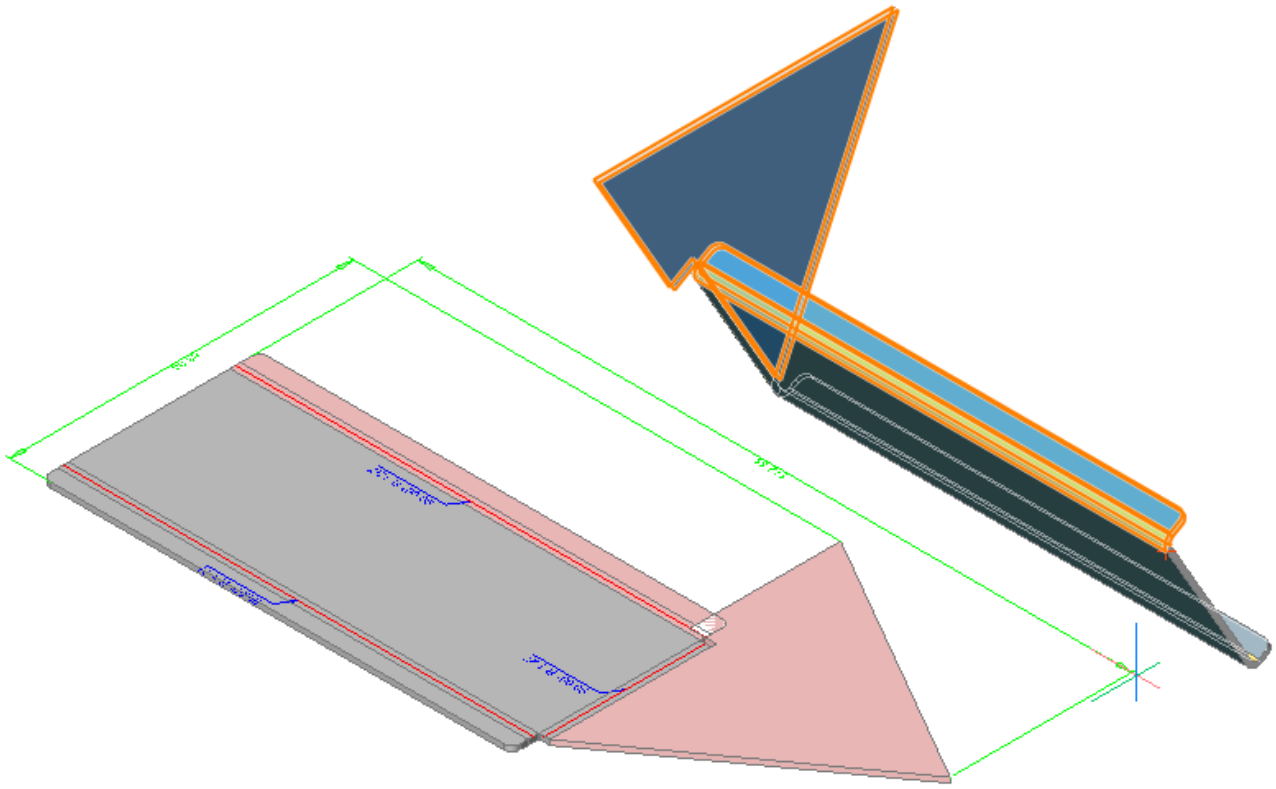


Figure 3.452

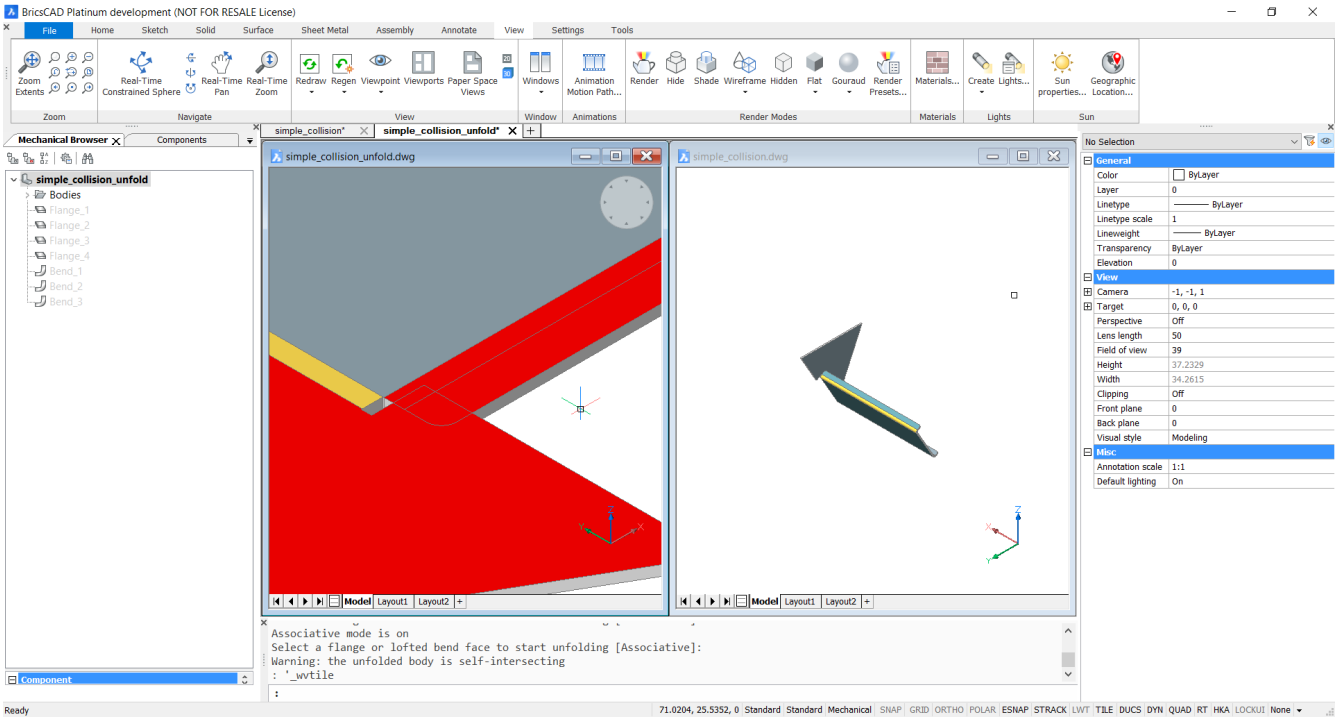
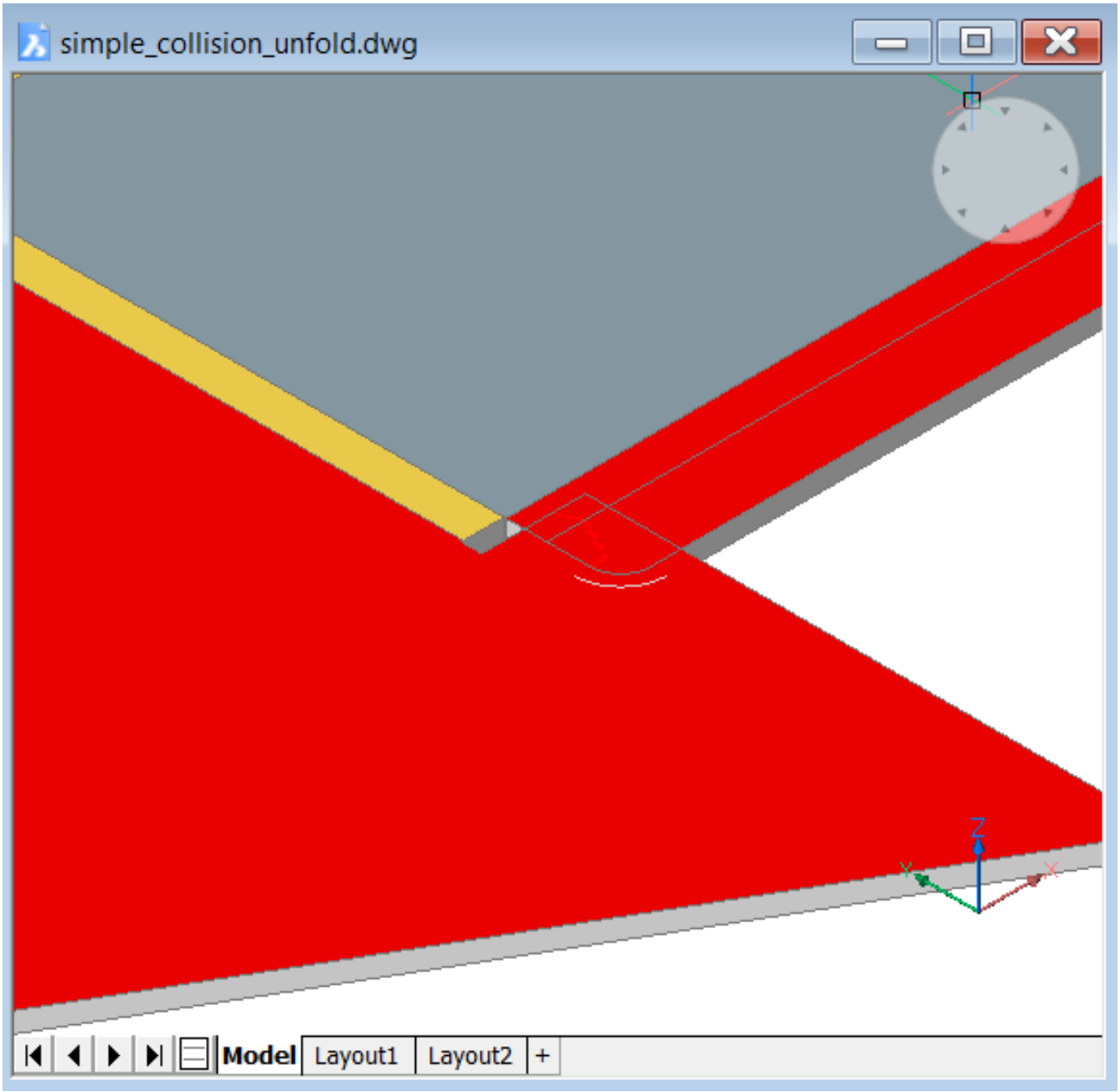


Figure 3.453



4. DmExtrude (6.12.13)the polyline on both sides. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/
5. DmThicken (6.12.31) the surface by a bigger value. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
6. Subtract (6.27.56) thickened body from unfolded pattern. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
7. Erase (6.13.8) the leftover pieces. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/S
8. Switch back to the document with the 3D model. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/
9. Make sure the that changes have been propagated. Changes were successfully transferred to source model.
   Zoom to the corner that updated. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Sl
10. To test the 2D unfold, export the file as a .dxf. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Me

Figure 3.454

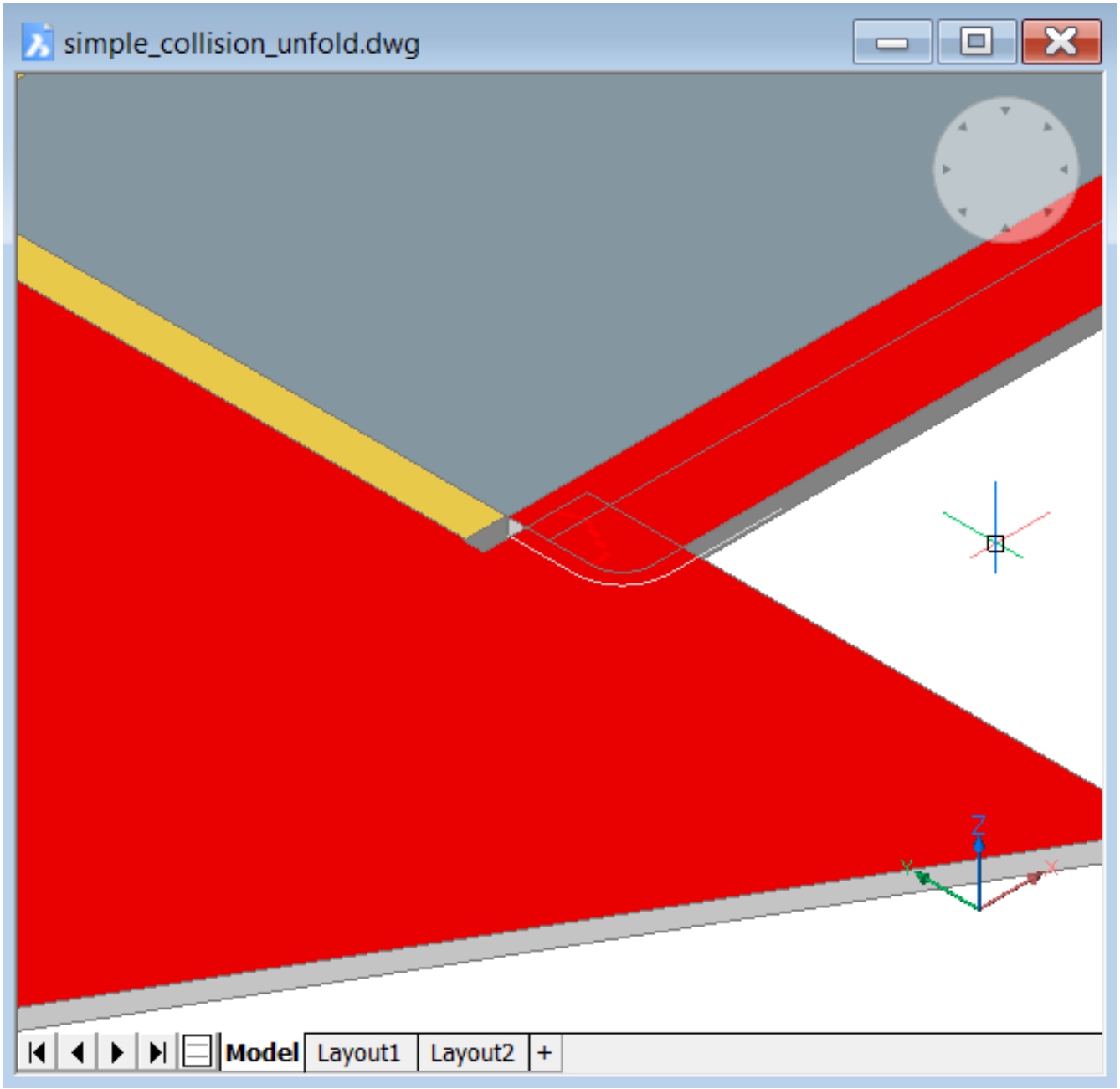


Figure 3.455

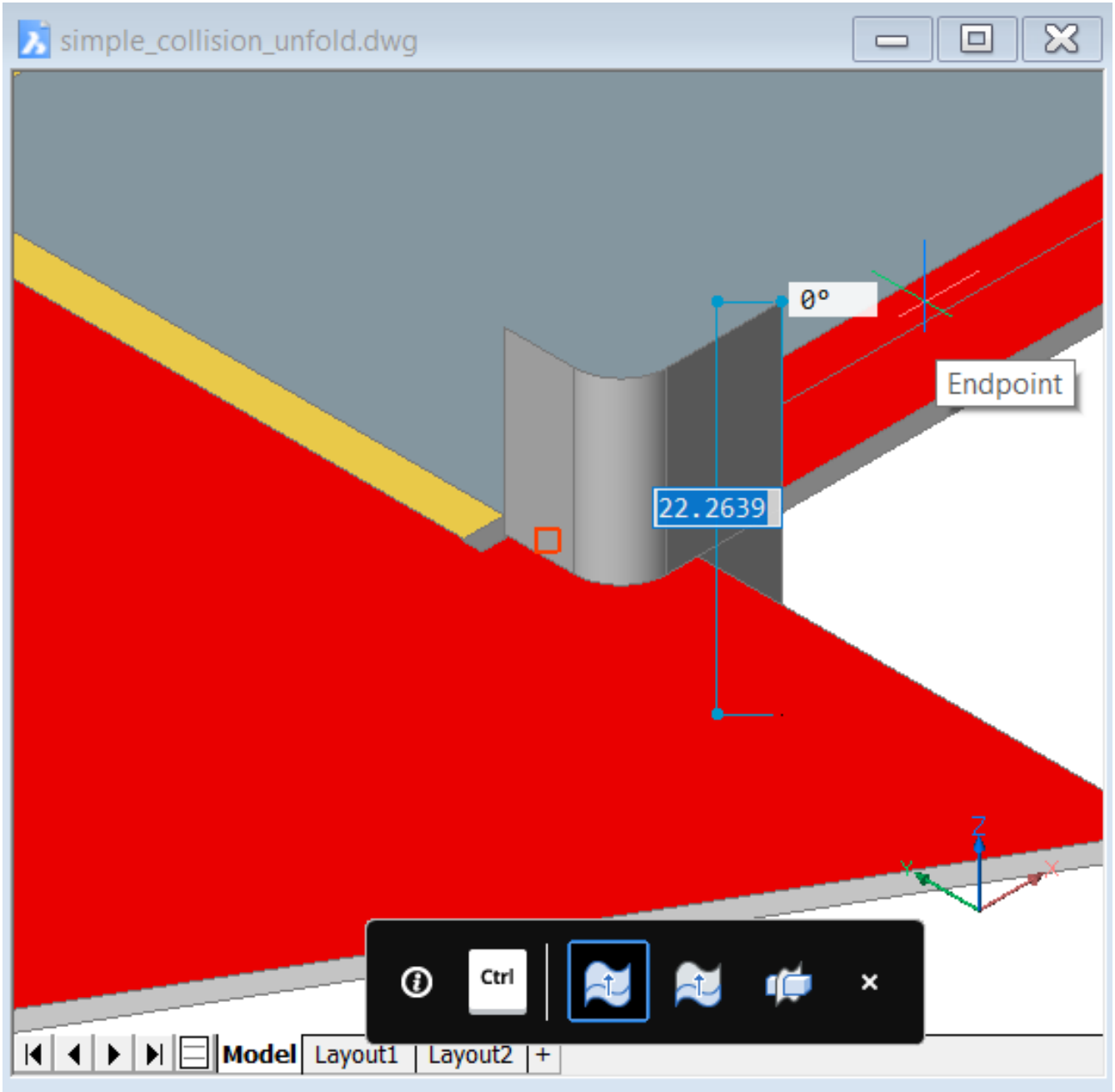

simple_collision_unfold.dwg
0°
Endpoint
22.2639
Ctrl
Model
Layout1
Layout2
+

Figure 3.456

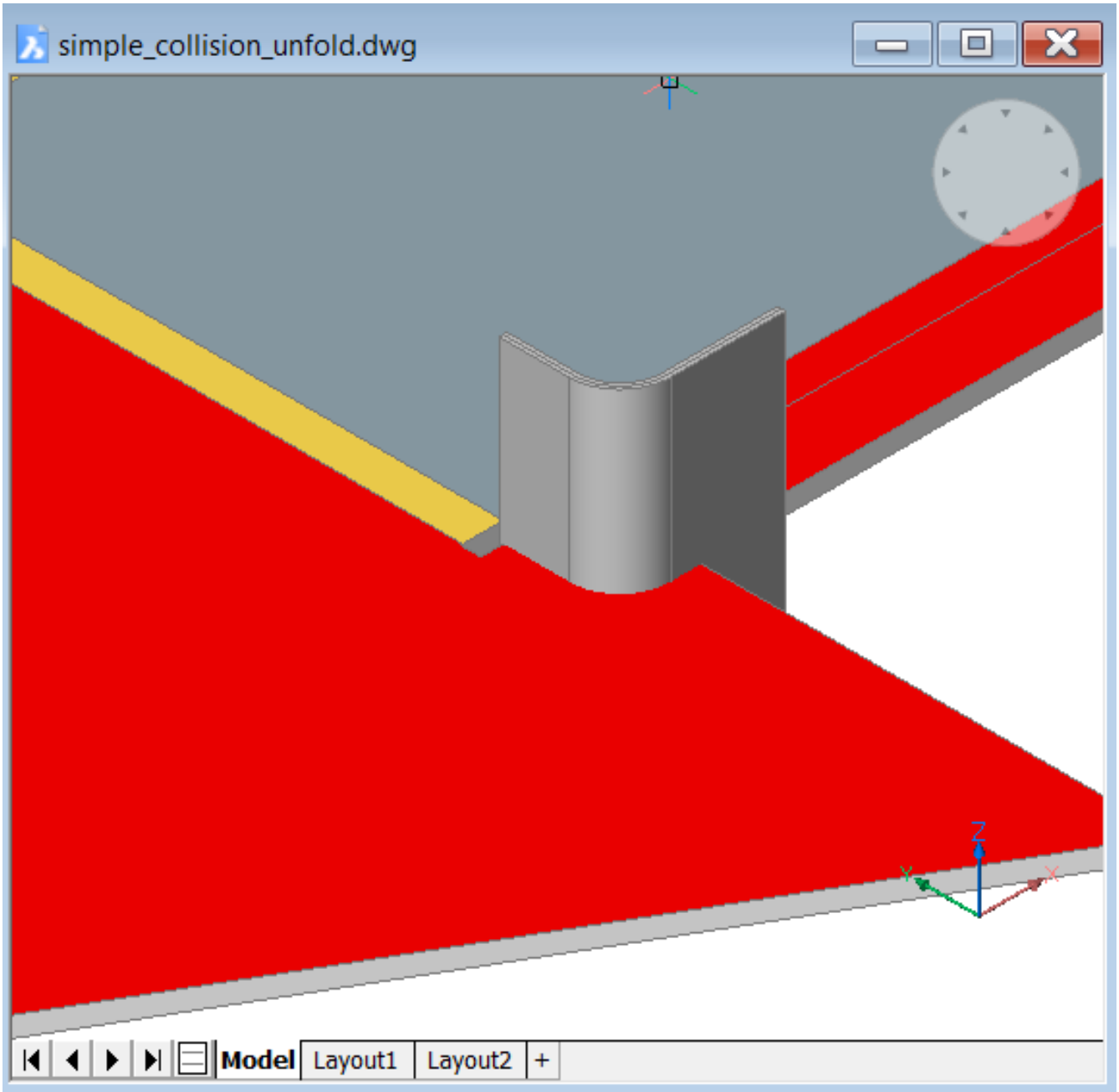


Figure 3.457

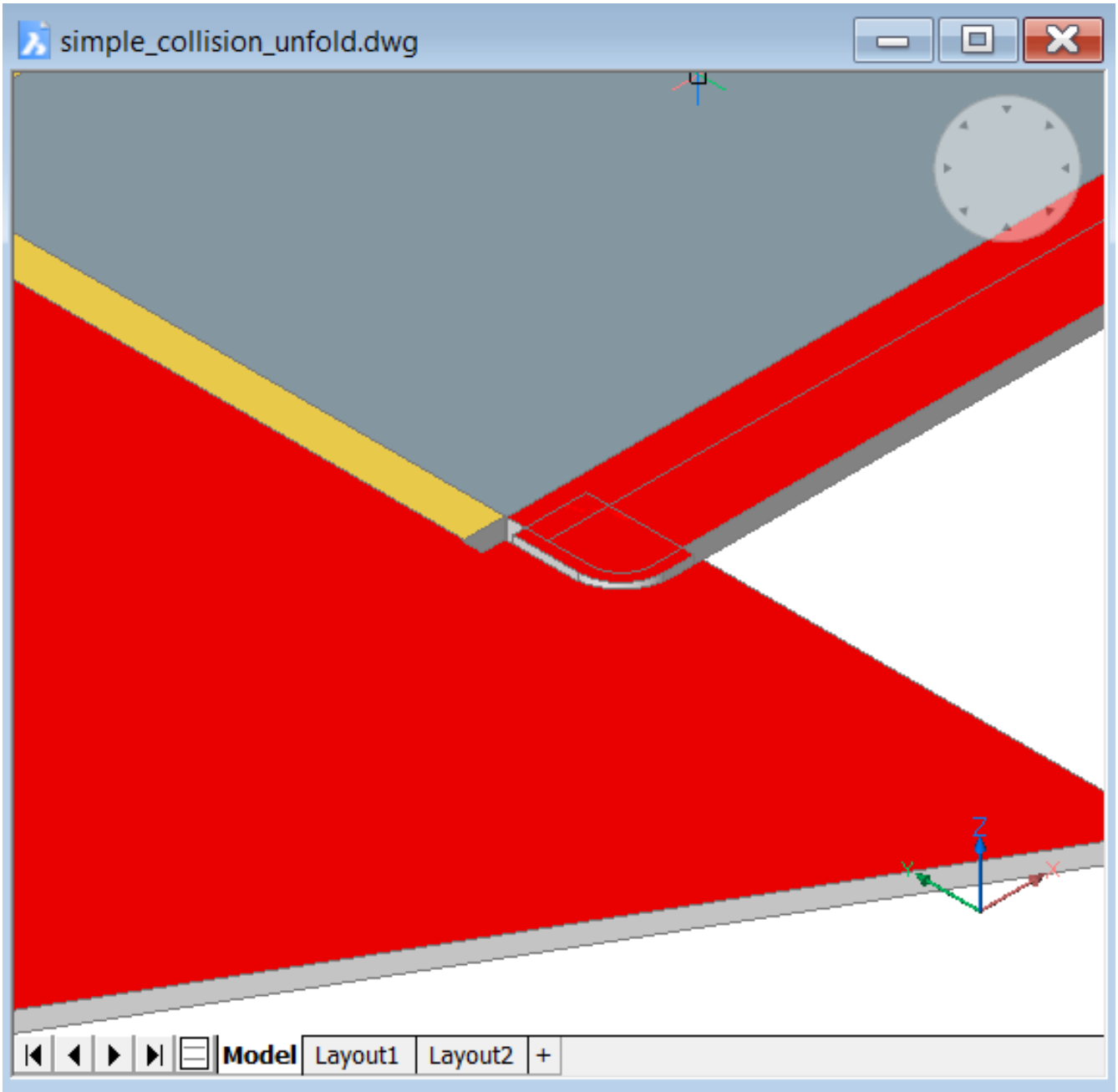


Figure 3.458

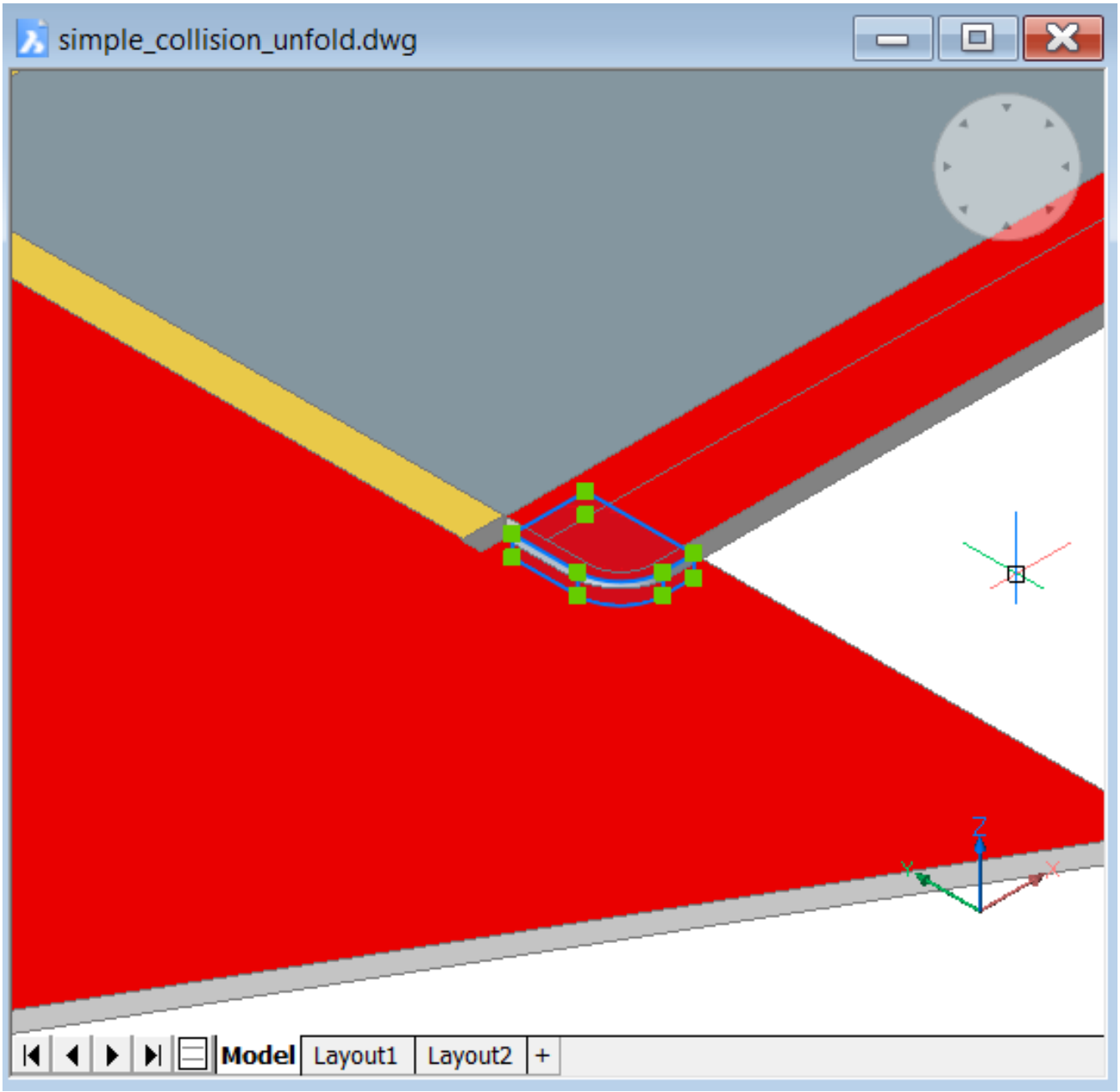


Figure 3.459

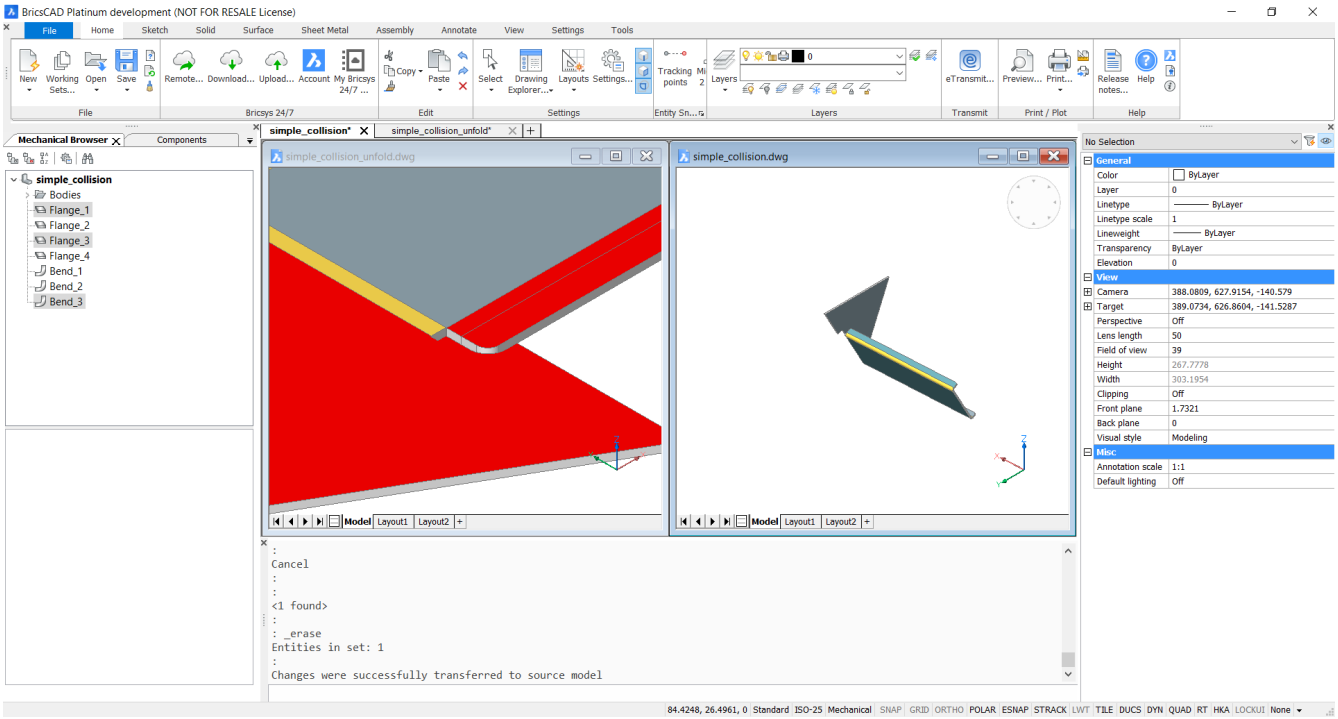

BricsCAD Platinum development (NOT FOR RESALE License)
Mechanical Browser
simple_collision
Bodies
Flange_1
Flange_2
Flange_3
Flange_4
Bend_1
Bend_2
Bend_3
simple_collision_unfold.dwg
simple_collision.dwg
Cancel
<1 found>
: _erase
Entities in set: 1
Changes were successfully transferred to source model
No Selection
General
View
Misc

Figure 3.460

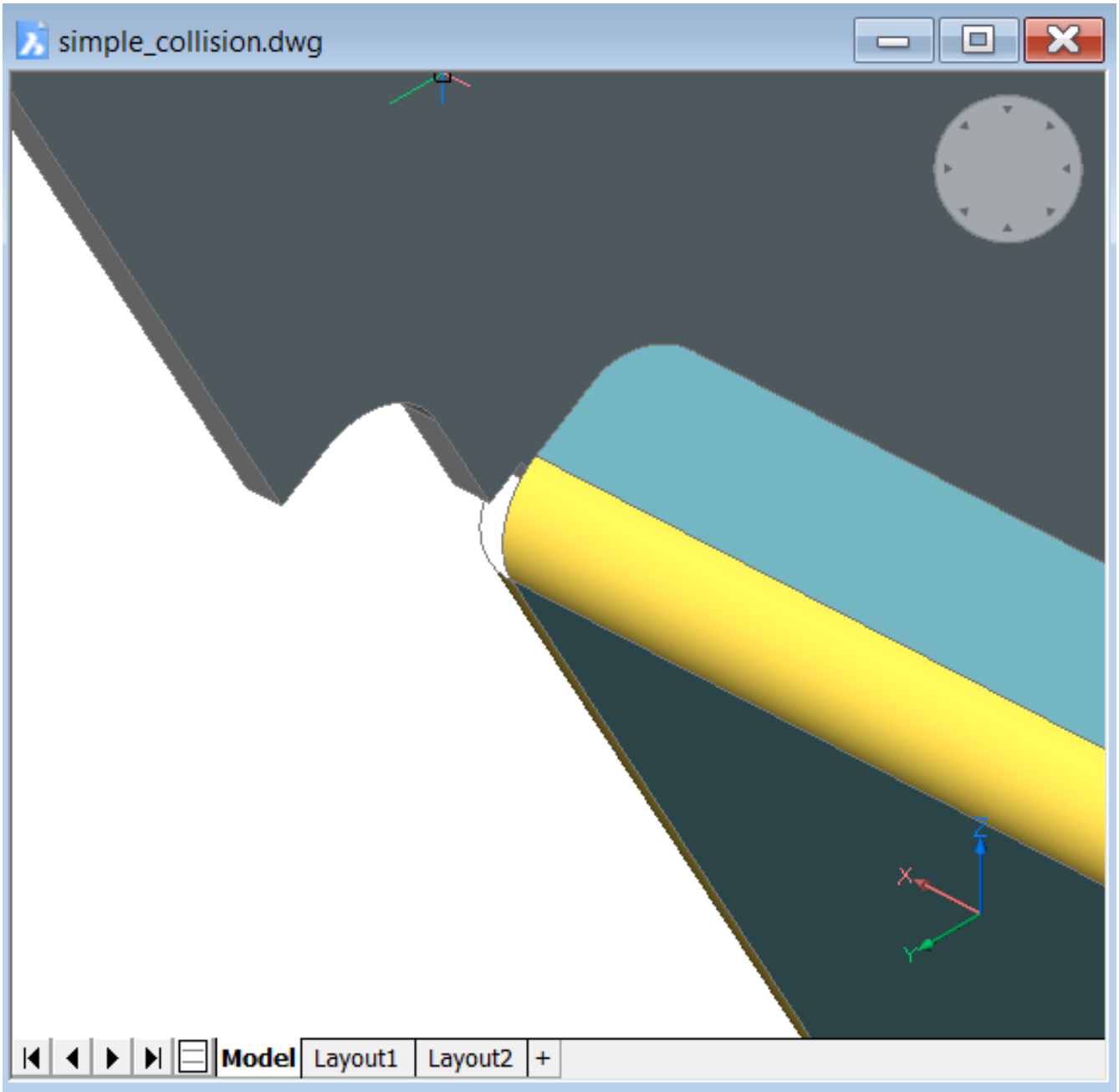
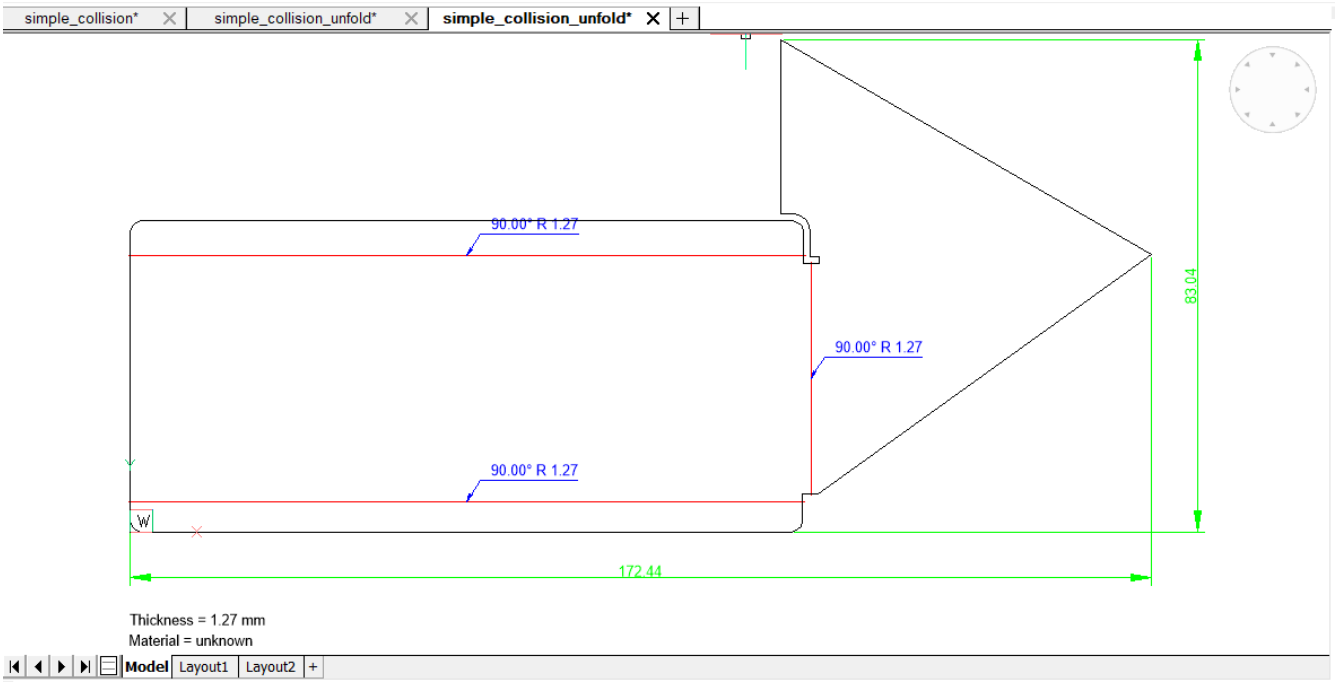


Figure 3.461


simple_collision*
simple_collision_unfold*
simple_collision_unfold*
90.00° R 1.27
90.00° R 1.27
90.00° R 1.27
83.04
172.44
Thickness = 1.27 mm
Material = unknown
Model
Layout1
Layout2

3.8.18 Working with Form and Rib Features

Form and Rib features are created by a forming tool. Rib features are produced by imprinting a wheel rolling over the flange following an arbitrary trajectory. We will use 'form' to refer to both form and rib features.Form features can be recognized in imported geometry or inserted from built-in or user-defined libraries. Form features can be edited directly or parametrically. Form features are listed in the Mechanical Browser (3.2.2) with their parameters. Parameters of form features are display and can be edited in the Properties Panel (1.12.4)when selected. The following commands apply to form features:

Icon	Command	Description
	SmForm (6.28.16)	Converts a selected set of faces to a form feature. A drawing file with a user-defined form feature can be saved and then inserted using theBmInsert (6.8.12)command.
	SmConvert (6.28.4)	Recognizes form and rib features in imported geometry.
	SmReplace (6.28.25)	Allows replacing form features (including recognized ones) in sheet metal parts with form features from the built-in or user library.
	SmRibCreate (6.28.25)	Creates associative Rib form features on sheet metal parts from 2D profiles. The feature is updated automatically when the defining profile has been modified.
	BmInsert (6.8.12)	Inserts an existing mechanical component or sheet metal form feature into the current drawing.
	SmSelect (6.28.27)	Selects comparable form features. The command works in two modes: • **Similar**:features which have the same set of faces, but possibly different sizes. • **Same**: identical features.

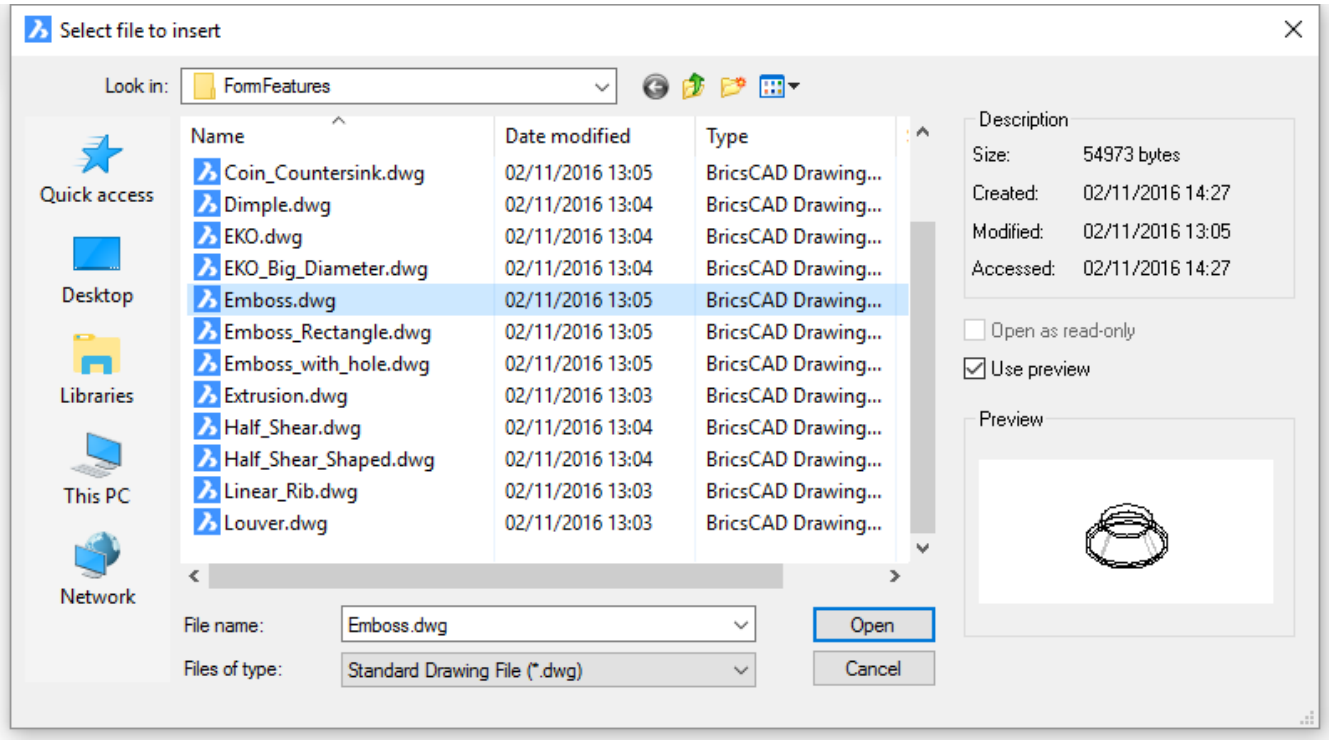
A library of parametrized sheet metal form features is available from the Form Features tab on the Tool Palettes (1.13.7) panel. The library contains the most commonly used form feature types. To insert a form feature to a sheet metal 3D solid, drag the form feature icon from the Tool Palettes (1.13.7) panel or theComponents (6.9.61) panelonto a flange of the solid. TheSMDEFAULTFORMFEATUREUNFOLDMODE (1.13.11) system variable controls the appearance of form features in 2D and 3D unfolded model representations.

To Insert a Form Feature

Form features are added to a sheet metal part by insertion on the existing flanges. The form features can be inserted by means of BmInsert (6.8.12) command or by drag-and-drop from the**Form Features** tab of Tool Palettes (1.13.7).

1. Make sure Dynamic Dimensions (1.15.9) (DYN) and Dynamic UCS (1.15.18) (DUCS) are active.
2. Do one of the following:
   - Click the **Form Feature** tool button ( ) on the **Sheet Metal** toolbar.
   - Click the **Form Feature** tool button ( ) on the **Sheet Metal** ribbon bar.
   - Choose **Insert Form Feature** in the **Sheet Metal** menu.

Figure 3.462



- Type **bminsert** in the command bar, then press Enter. The **Select file to insert** dialog displays the **Form-Features** folder **[BricsCAD Installation folder]\Support\DesignLibrary\SheetMetal\FormFeatures: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingS** Select the desired form feature, then double click or press the **Open** button.
- Select the form feature from Form Features tab ofComponents (6.9.61) panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/mceclip1.png})
- Select the form feature on the **Form Features** tab of the Tool Palettes (1.13.7) panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/ToolPal

3. Move the cursor over the flange face you want to insert the feature in. The dynamic UCS aligns to the selected face. The orientation of the UCS depends on the edge the cursro enters the face. Dynamic dimensions display from the origin of the UCS to the insertion point of the form feature. **You are prompted:** Select insertion point or [Edit inserted entity/Rotate component/set Base point/Name/insertion Type/Flip/mUltiple] <0, 0, 0>:
4. (option) Choose an option (see the BmInsert (6.8.12) command for more details):
   - **Edit:** Allows to edit the parameters of the selected insert.
   - **Rotate:** Allows to define the rotation angle of the component.
   - **Base Point:** Allows to specify a different base point to insert the component.
   - **Name:** Allows to define the insert name of the component.
   - **Insertion Type:** Allows to specify the insertion type: Local or External
   - **Flip:** Inserts the form feature in the opposite face of the flange.
   - **Multiple:** Allows to insert multiple inserts of the same component or create an array.
5. To insert a form feature do one of the following:
   - Specify a point.
   - Type a distance in the dynamic dimension fields. Hit the TAB key to go back and forth between the entry fields.
6. Repeat step 5 to insert another copy or press Enter to stop.

In order to have Form Features in theComponents (6.9.61) panel, you must create or use theMechanical Profile (3.2.1)in theGet Started (1.10.6) dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/S

Figure 3.463

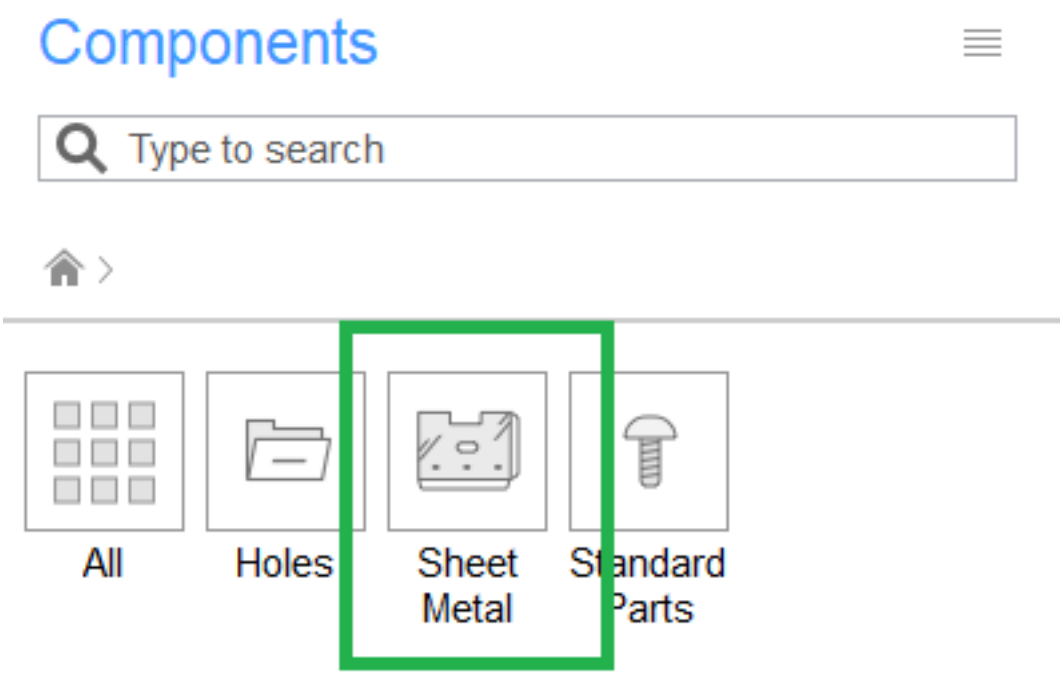

Components
Type to search
All
Holes
Sheet Metal
Standard Parts

Figure 3.464

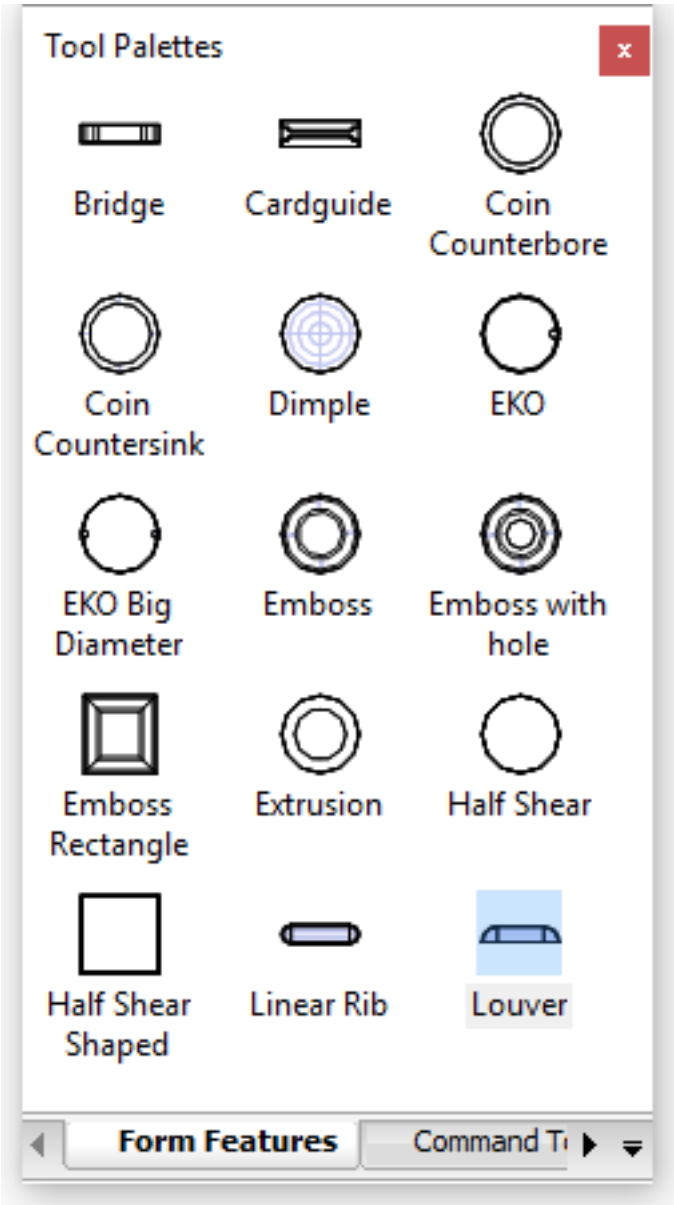
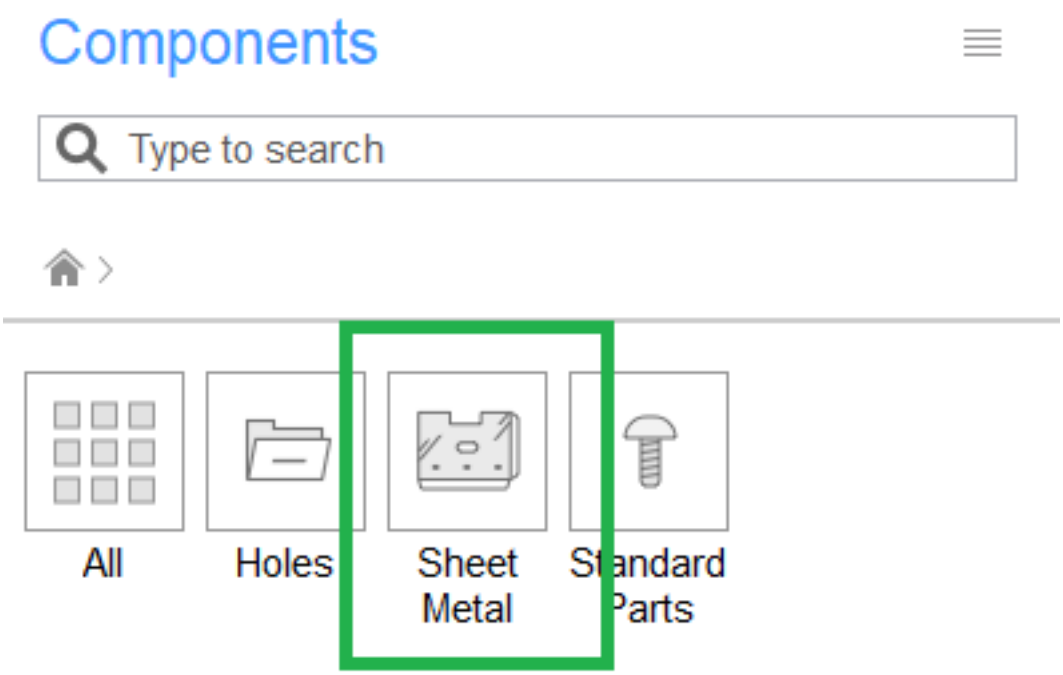

Tool Palettes
Bridge
Cardguide
Coin Counterbore
Coin Countersink
Dimple
EKO
EKO Big Diameter
Emboss
Emboss with hole
Emboss Rectangle
Extrusion
Half Shear
Half Shear Shaped
Linear Rib
Louver
Form Features
Command T

Figure 3.465


Components
Type to search
All
Holes
Sheet Metal
Standard Parts

Editing the Parameters of a Library Form Feature

Form features in the form feature library are parametric components. Several parameters control their dimensions. Parameters to control the Length, Width, Height and Radius of the form feature are correspondingly named L, W, H and R. The thickness of the form feature is set equal to the thickness of sheet metal part automatically. The parameters of a single form feature can be edited either in the Properties Panel (1.12.4) or in the Mechanical Browser (3.2.2). Parameters of multiple form features can be edited in the Properties Panel (1.12.4)only.

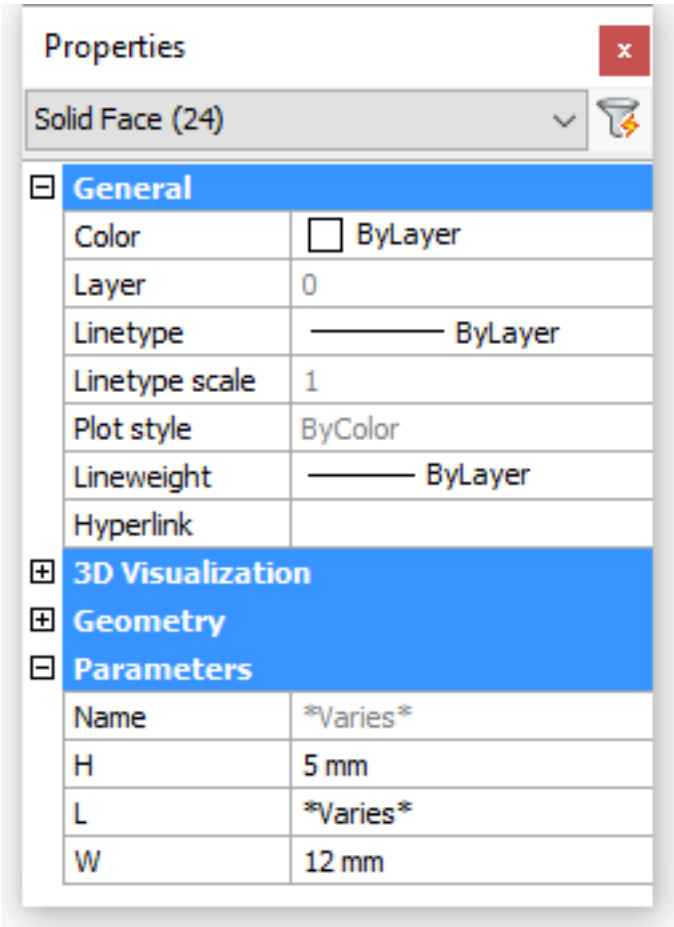
Methods to select form features

- Check the **Select Faces** option of the SELECTIONMODES (1.13.11) system variable, then click the feature(s) in the model.
- Select the feature(s) in the Mechanical Browser (3.2.2), then press the space bar or right click and choose **Select** in the context menu. Press and hold the CTRL key to select multiple form features.
- A sheet metal part can have lots of different form features, which might complicate operations such as deletion or replacement, especially if only a selection of features, that meet certain conditions, need to be processed. In order to help you to select form features more easily, the SmSelect (6.28.27) command allows to select form features that are either identical to a selected feature or similar.

Editing parameters in the Properties Panel

1. Select the form feature(s).
2. Select a parameter in the **Parameters** node in the Properties Panel (1.12.4). (\vref {fig:C:/Dev/Task-2448/images/staticbr

Figure 3.466

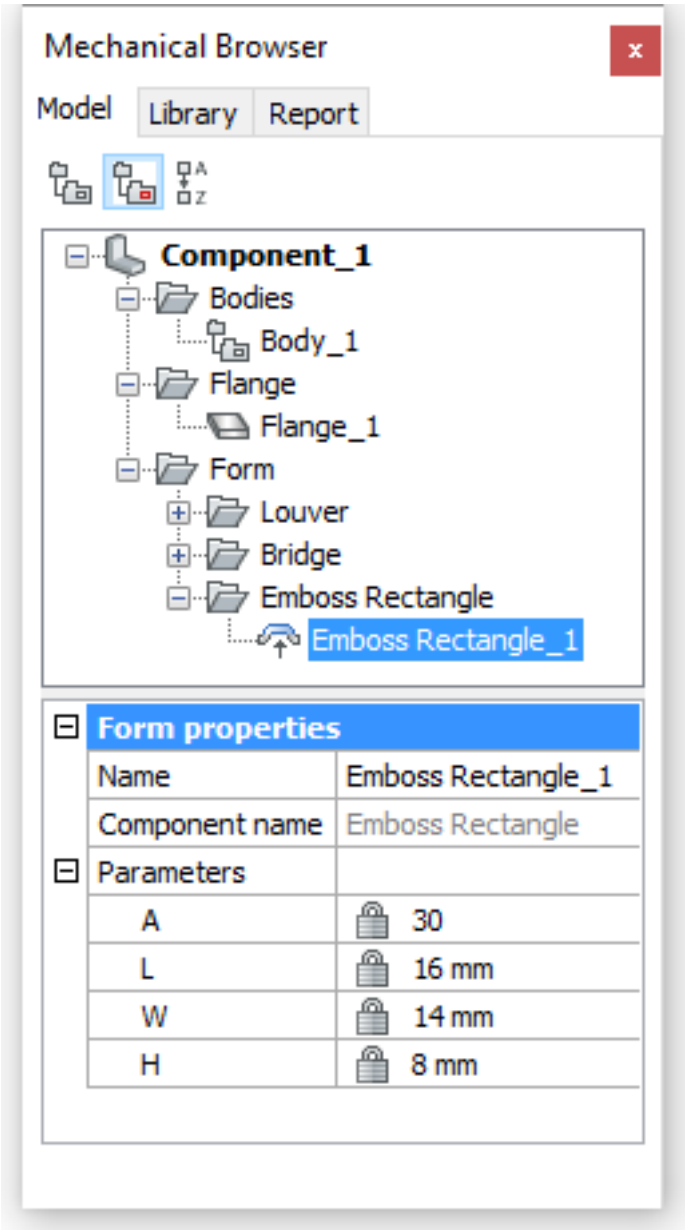


Editing parameters in the Mechanical Browser

1. Select the form feature in **Form** node of the component structure tree. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys

2. Select a parameter in the lower panel ofMechanical Browser (3.2.2) and type a new value.

Figure 3.467


Mechanical Browser
Model
Library
Report
Component_1
Bodies
Body_1
Flange
Flange_1
Form
Louver
Bridge
Emboss Rectangle
Emboss Rectangle_1
Form properties
Name
Emboss Rectangle_1
Component name
Emboss Rectangle
Parameters
A
30
L
16 mm
W
14 mm
H
8 mm

Creating a custom form feature

1. Open a new drawing using **Mechanical-metric.dwt** as the template.
2. Create the 3D geometry of the form feature. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrG
3. Execute the SmConvert (6.28.4) () command.
4. Execute the BmMech (6.8.15) () command. **You are prompted:** Name for mechanical component <Component_2>:
5. Type a name for the form feature. **You are prompted:** Convert blocks and external references to mechanical components? [Yes/No] <Yes>:
6. Select the **No** option.
7. In the **Extension type** field in Mechanical Browser (3.2.2) select **Sheet Metal Form Feature**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/CreateFormI
8. Select the Flange_1 and Form_1 features in the **Bodies** node, then right click and select **Disable** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03Dra
9. Create 2 new layers (6.20.2): UNFOLD_SYMBOL_UP and UNFOLD_SYMBOL_DOWN.
10. Turn layers 0 and UNFOLD_SYMBOL_DOWN **Off** and set layer UNFOLD_SYMBOL_UP **Current**.
11. Create the unfold symbol, which will be placed on the face the form feature is placed in. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/CreateFormI

Figure 3.468

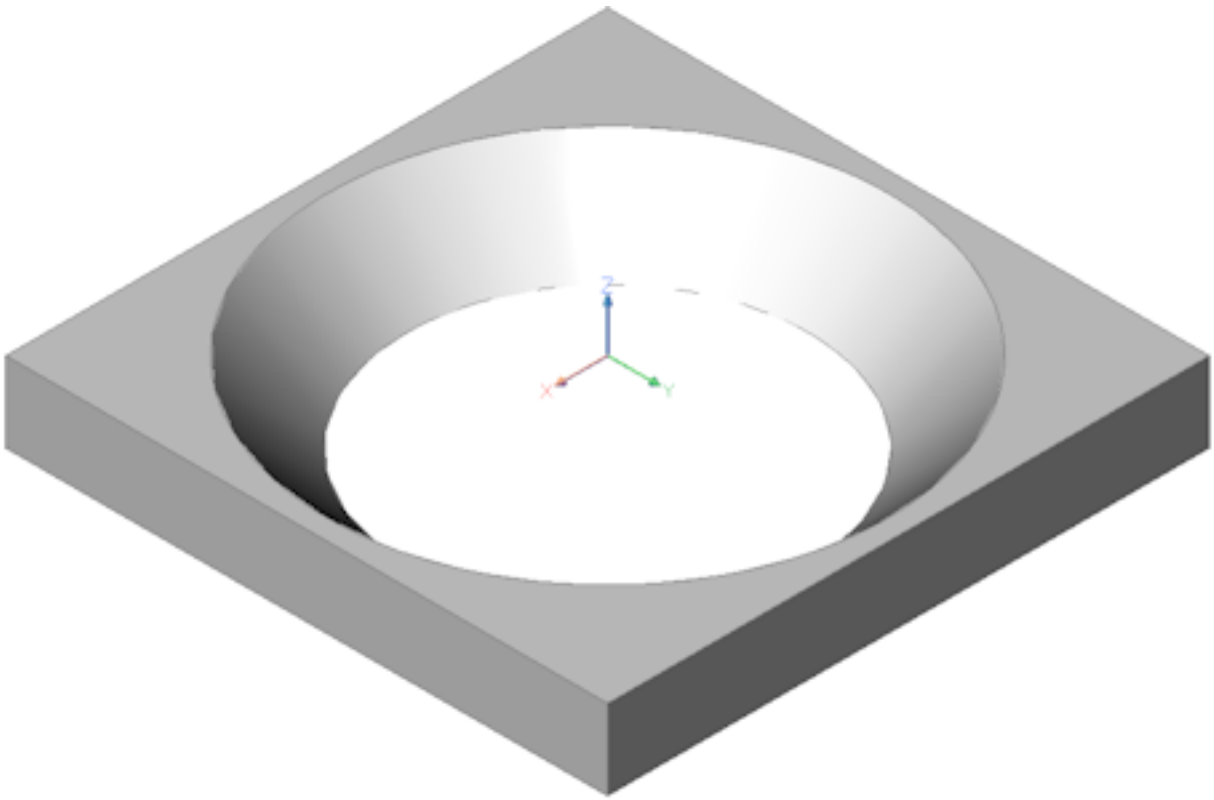


Figure 3.469

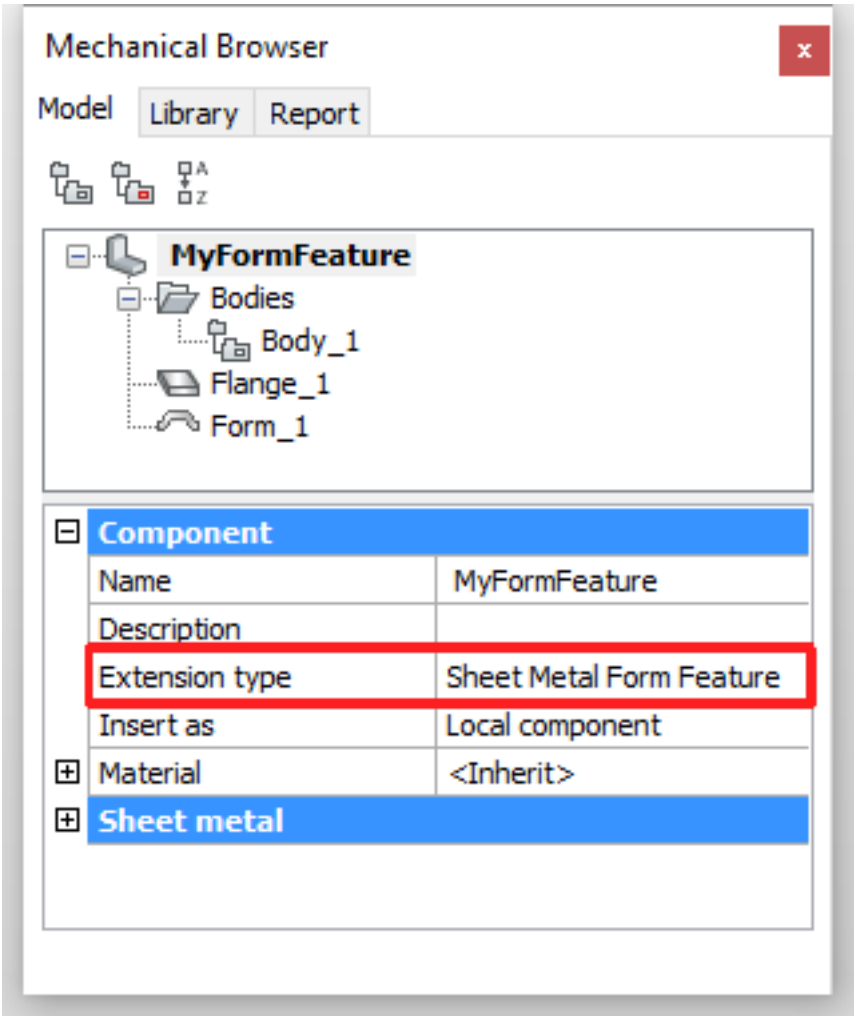
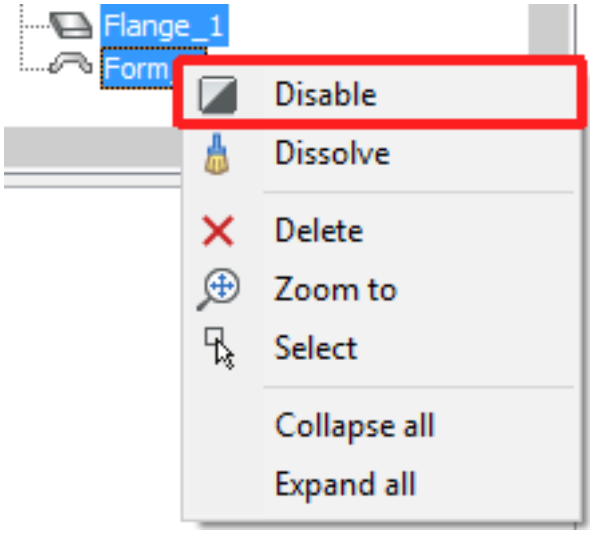


Figure 3.470



12. Turn **off** the UNFOLD_SYMBOL_UP layer and turn **on** the UNFOLD_SYMBOL_DOWN layer. Set the UNFOLD_SYMBOL_DOW layer **current.**
13. Create the unfold symbol, which will be placed on the opposite face the form feature is placed in. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/A
14. Turn **on** layer 0 and turn the UNFOLD_SYMBOL_UP and UNFOLD_SYMBOL_DOWN layers **off.** Set layer 0 **current.**
15. Save the drawing.

Figure 3.471

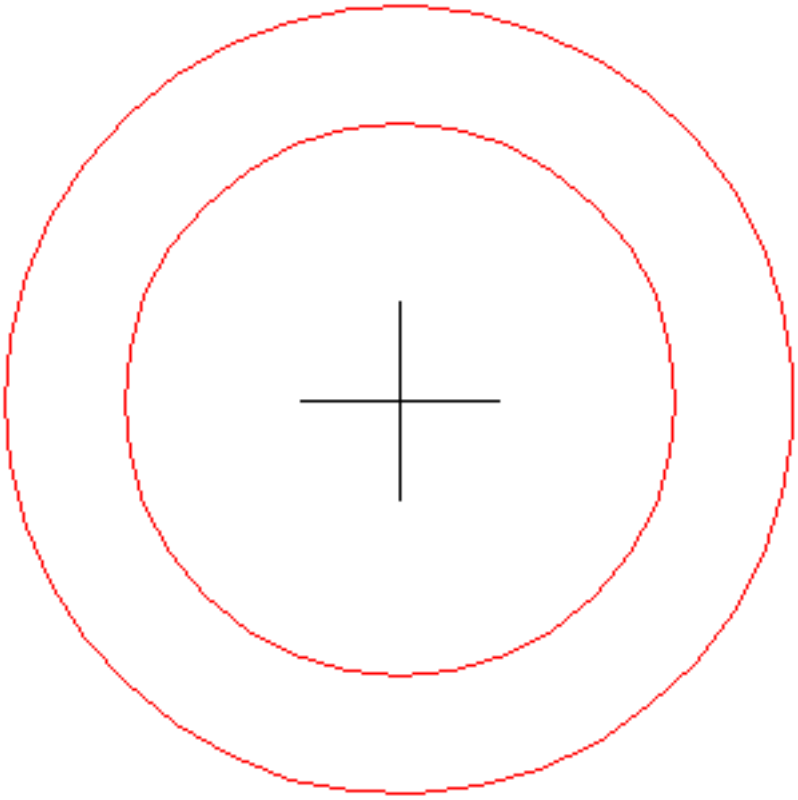
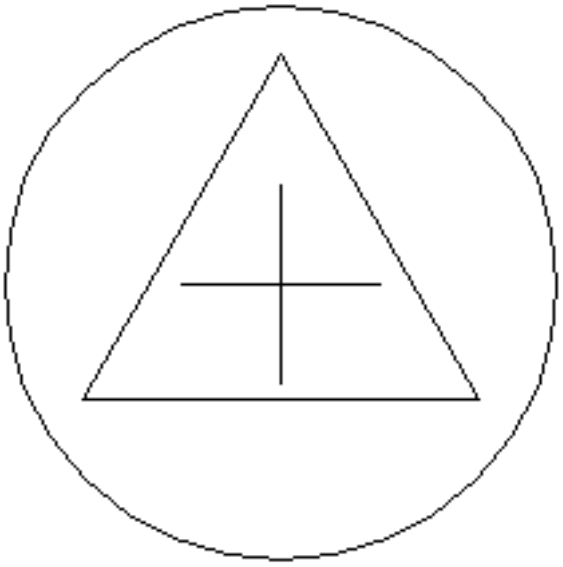


Figure 3.472



Creating a custom parametric form feature

We will now parametrize the custom form feature we created in the previous procedure. Two parameters are needed here:

- **D** for a diameter of the bigger edge of the conical hole.
- **A** for an angle of the face of the conical hole.

In Sheet Metal we have a default parameter **_SM_THICKNESS** which controls the thickness of the sheet metal part.

1. We will first create an auxiliary body for constraining on a new CONSTRUCTION layer. Make this layer current and switch off all other layers.
2. Create a Region (6.26.18) in the XY plane of the WCS on the CONSTRUCTION layer. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/CreateParan
3. Apply a DmFix3D (6.12.15) constraint ( ) to the region. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
4. Switch ON layer 0.
5. Apply a DmCoincident3d (6.12.5)constraint ( ) between the region face and the flange top face. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/A
6. Apply a DmDistance3d (6.12.12)constraint ( ) between the flange top and bottom faces. First select the top face, then hit the TAB key to select the bottom face. Accept the default value in the dynamic dimension.
7. Type _SM_THICKNESS in the **Expression** field of the Distance_3 constraint. (\vref {fig:C:/Dev/Task-2448/images/staticbr
8. Since our Form Feature is conical we can modify the flange from parallelepiped to cylinder. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/CreateParan
9. In the Mechanical Browser (3.2.2) right click the feature name and select **Add new parameter** in the context menu. Rename new parameter to **D**. Select mm for the **Units** field. Type 16 in the **Expression** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/A D.png})
10. Apply a DmRadius3d (6.12.21)constraint ( ) to the top edge of the conical face. Accept the default value in the dynamic dimension.
11. Type D/2 in the **Expression** field of the Radius_5 constraint. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/
12. In the Mechanical Browser (3.2.2) right click the feature name and select **Add new parameter** in the context menu. Rename new parameter to **A** . Leave the **Units** field empty. Type 45 in the **Expression** field.

Figure 3.473

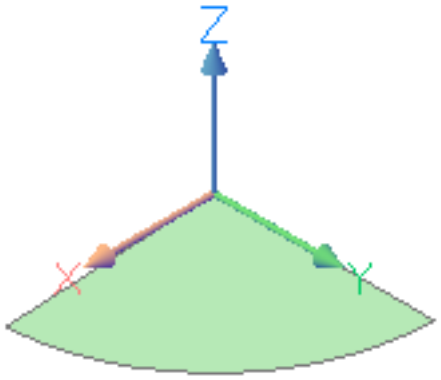


Figure 3.474

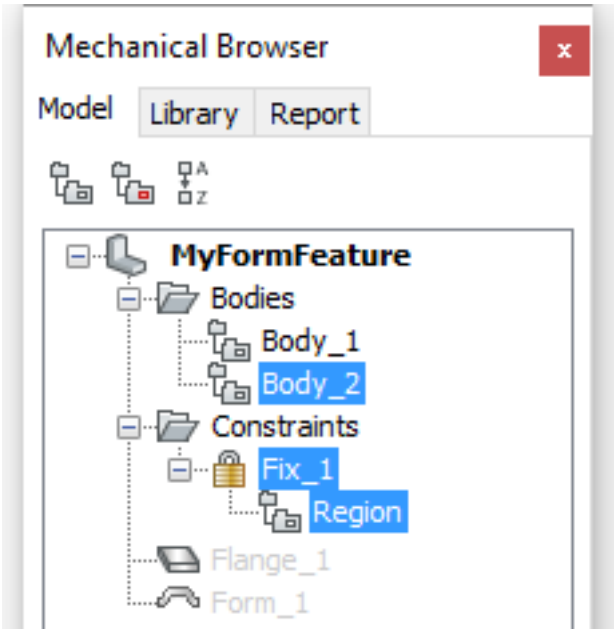


Figure 3.475

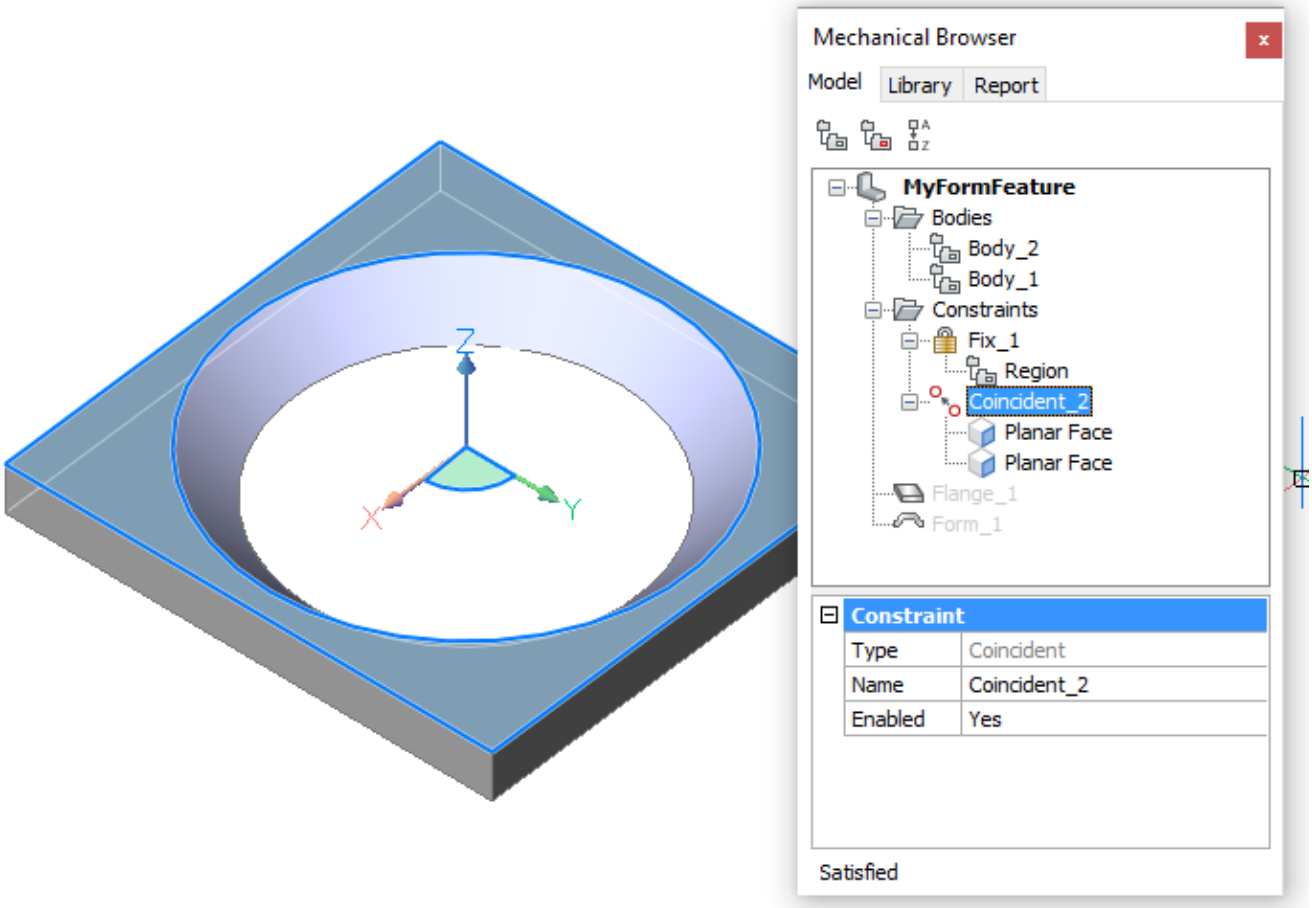


Figure 3.476

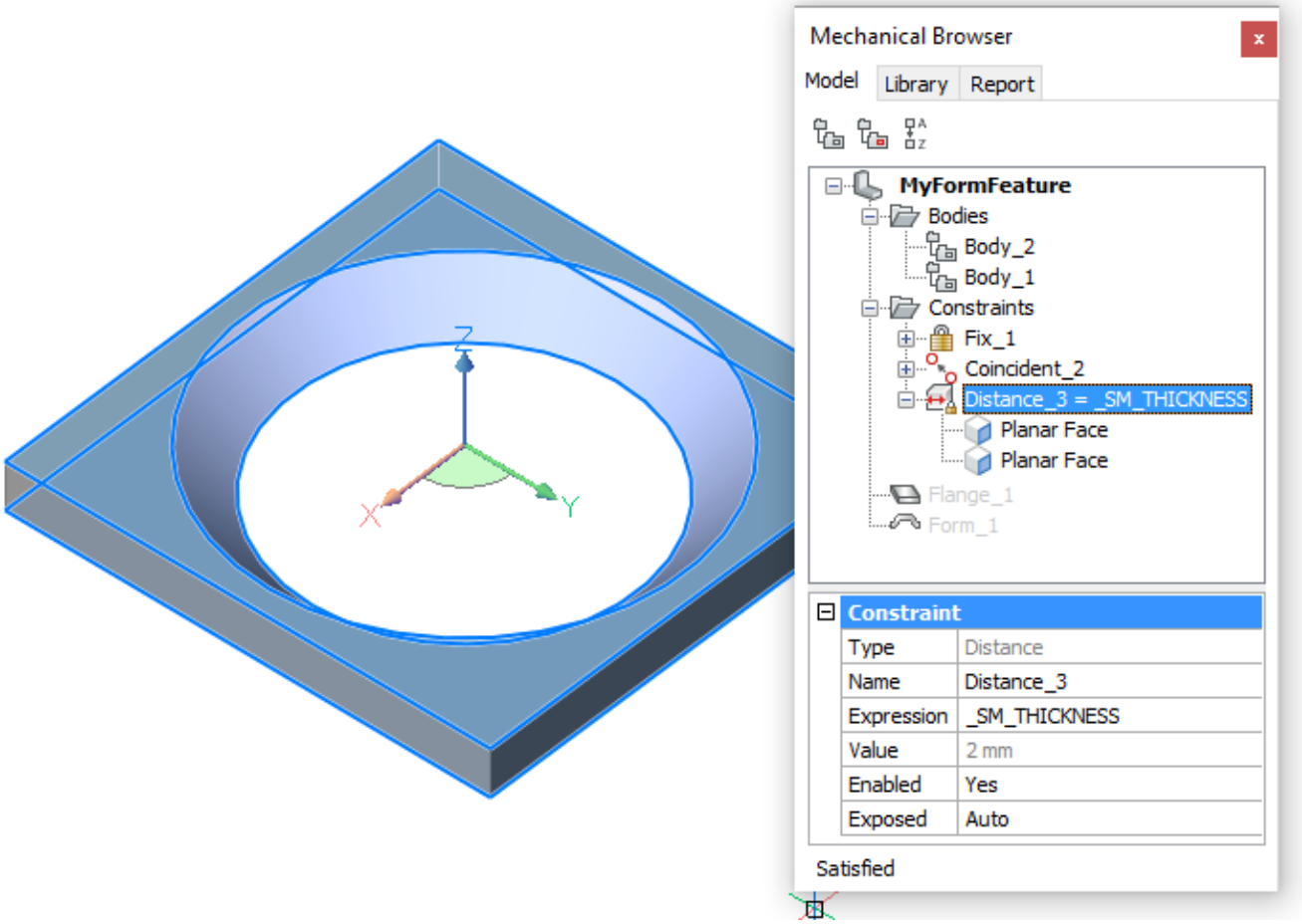

Mechanical Browser
Model
Library
Report
MyFormFeature
Bodies
Body_2
Body_1
Constraints
Fix_1
Coincident_2
Distance_3 = _SM_THICKNESS
Planar Face
Planar Face
Flange_1
Form_1
Constraint
Type
Distance
Name
Distance_3
Expression
_SM_THICKNESS
Value
2 mm
Enabled
Yes
Exposed
Auto
Satisfied
Z
Y
X

Figure 3.477

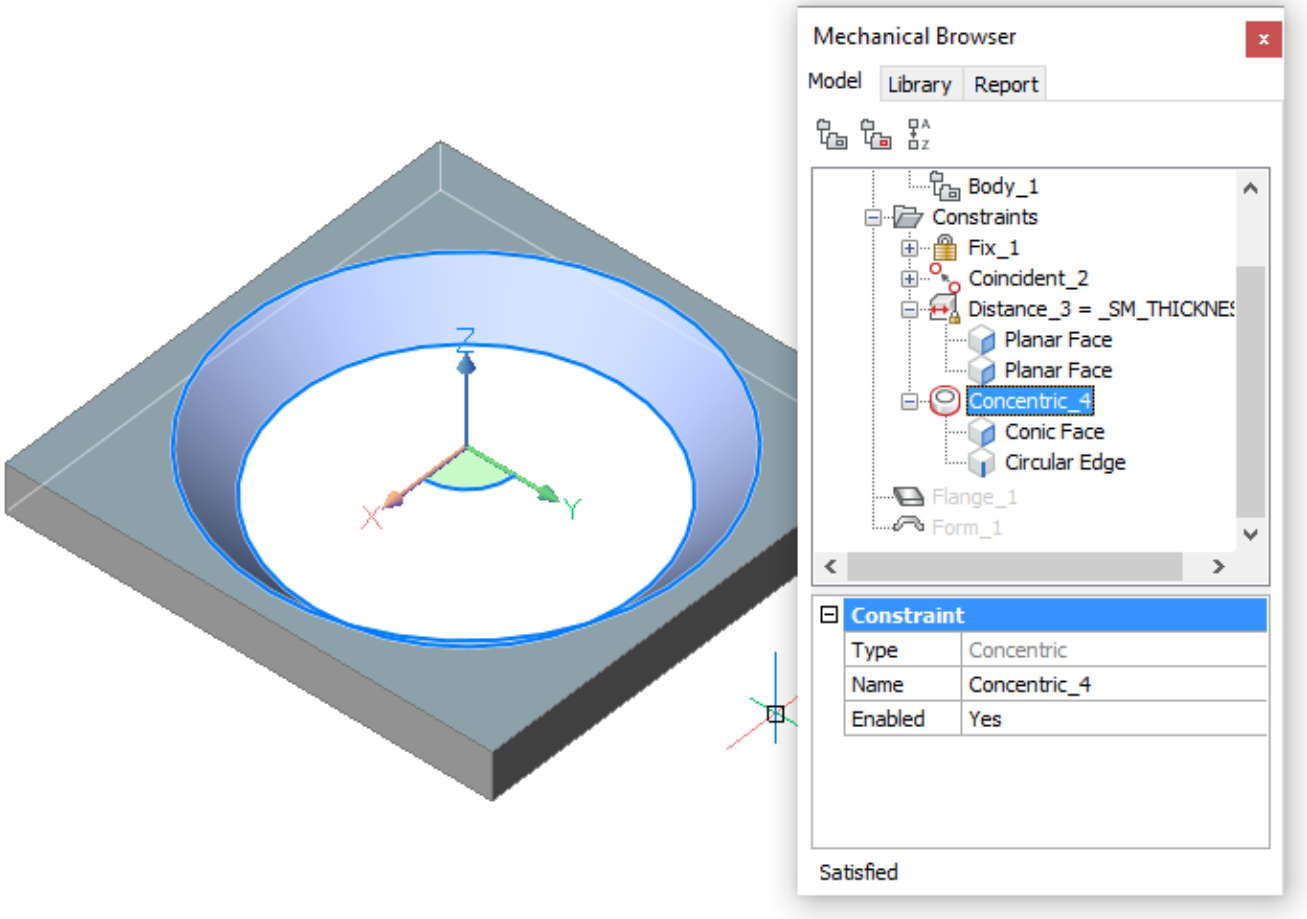

Mechanical Browser
Model
Library
Report
Body_1
Constraints
Fix_1
Coincident_2
Distance_3 = _SM_THICKNES
Planar Face
Planar Face
Concentric_4
Conic Face
Circular Edge
Flange_1
Form_1
Constraint
Type
Concentric
Name
Concentric_4
Enabled
Yes
Satisfied

Figure 3.478

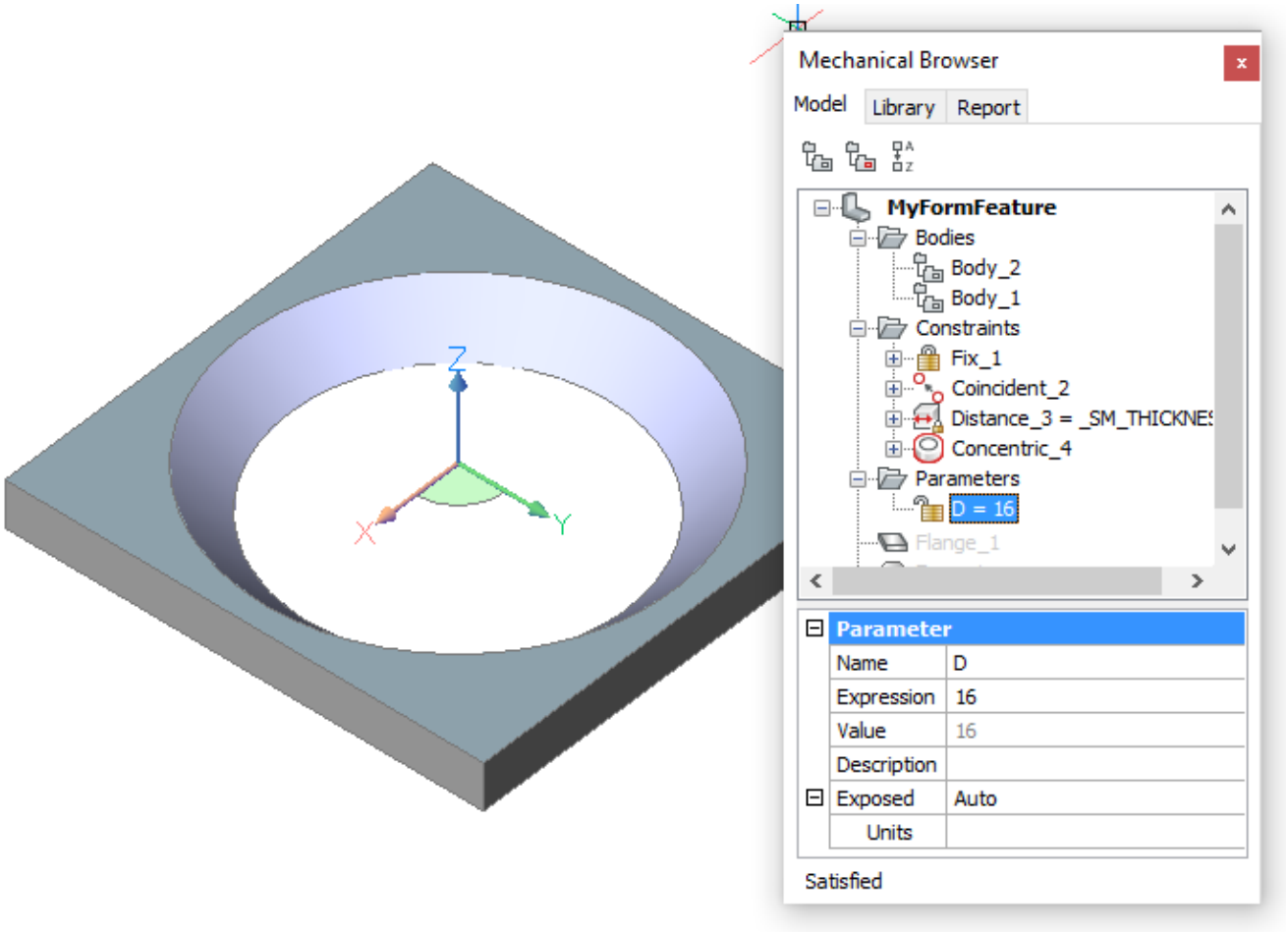

Mechanical Browser
Model
Library
Report
MyFormFeature
Bodies
Body_2
Body_1
Constraints
Fix_1
Coincident_2
Distance_3 = _SM_THICKNES
Concentric_4
Parameters
D = 16
Flange_1
Parameter
Name
D
Expression
16
Value
16
Description
Exposed
Auto
Units
Satisfied
Z
Y
X

Figure 3.479

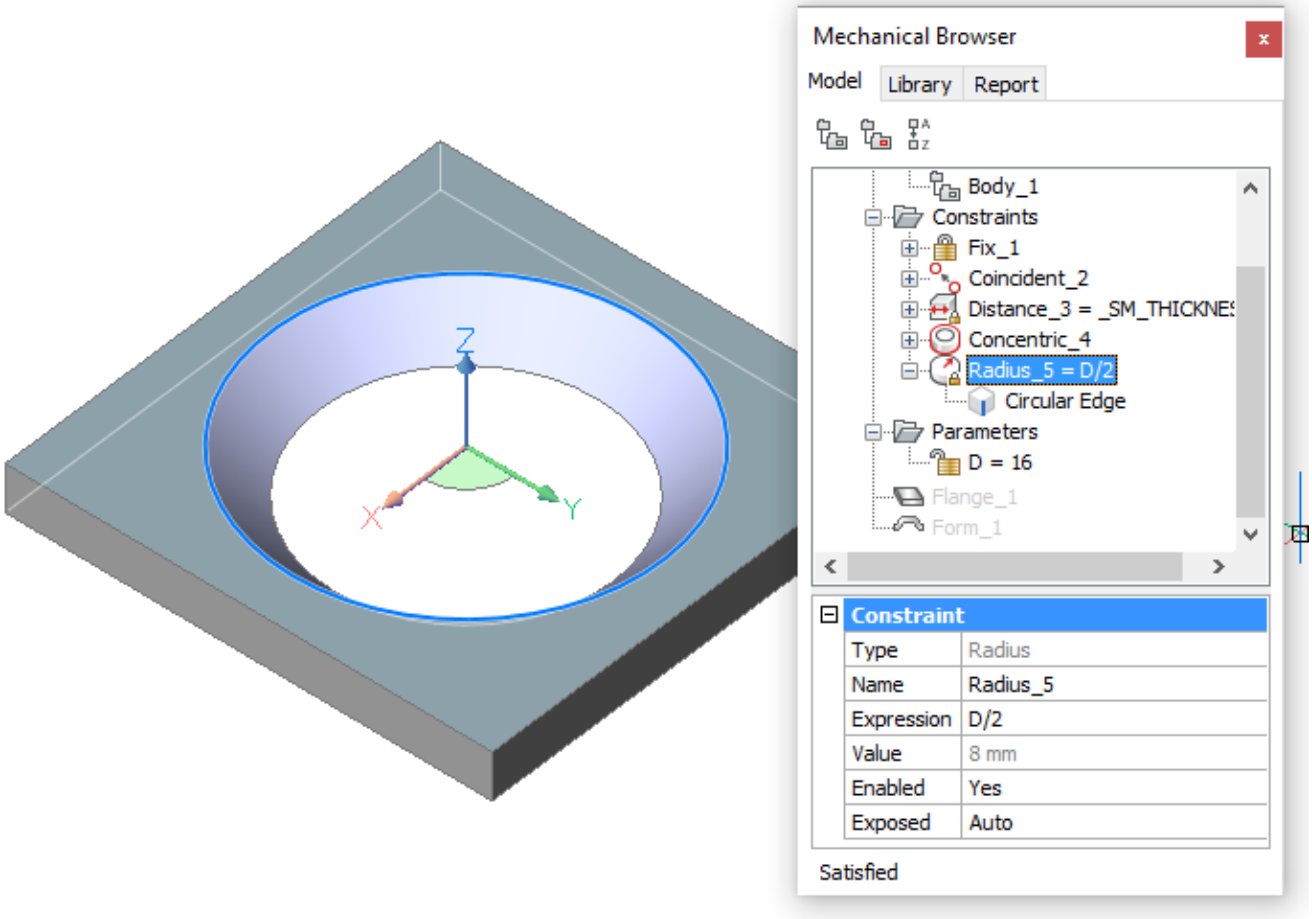
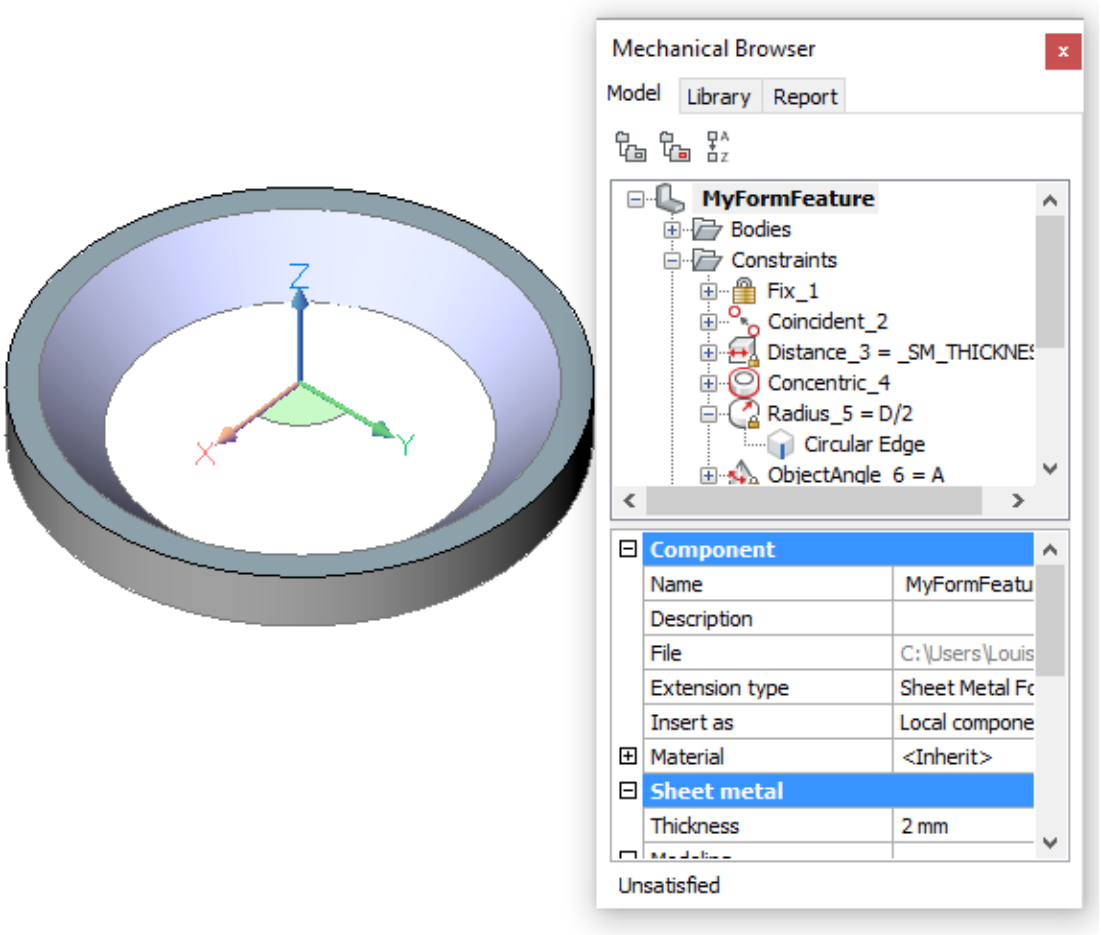

Mechanical Browser
Model
Library
Report
Body_1
Constraints
Fix_1
Coincident_2
Distance_3 = _SM_THICKNES
Concentric_4
Radius_5 = D/2
Circular Edge
Parameters
D = 16
Flange_1
Form_1
Constraint
Type
Radius
Name
Radius_5
Expression
D/2
Value
8 mm
Enabled
Yes
Exposed
Auto
Satisfied
Z
Y
X

Figure 3.480



13. Do one of the following to create a **Cone Angle constraint** to the conical face:

    - Launch the DmAngle3d (6.12.1) command ( ), then choose the **Cone Angle constraint** option, then select the conical face.
    - Hover over the conical face, then choose **Cone Angle constraint** ( ) in the **Constraints** command group in the Quad (2.2.3).

    Accept the default value.

14. Type A in the **Expression** field of the ObjectAngle_6 constraint.

15. Since our Form Feature is conical we will modify the flange from parallelepiped to cylinder:

    - Draw a circle on top of the flange. The radius of the circle is 9 mm.
    - Select **Enable Boundary Detection** in the **Settings** panel on the **Home** tab in the Ribbon.
    - Place the cursor over the area between the circle and the square edge of the flange, then choose **Extrude** ( ) in the Quad (2.2.3) and move the cursor downwards. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he

16. Apply a DmConcentric3d (6.12.6)constraint ( ) between the circular edge of the region and the cylindrical face. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03Draw

17. Apply a DmRadius3d (6.12.21)constraint ( ) to the cylindrical face. Accept the default value in the dynamic dimension.

18. Type D/2+0.1*_SM_THICKNESS in the **Expression** field of the Radius_8 constraint. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/CreateParan

19. Set the **Exposed** field of both parameter A and D **On**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
    We will now create 2D constraints for the unfold symbols.

Figure 3.481

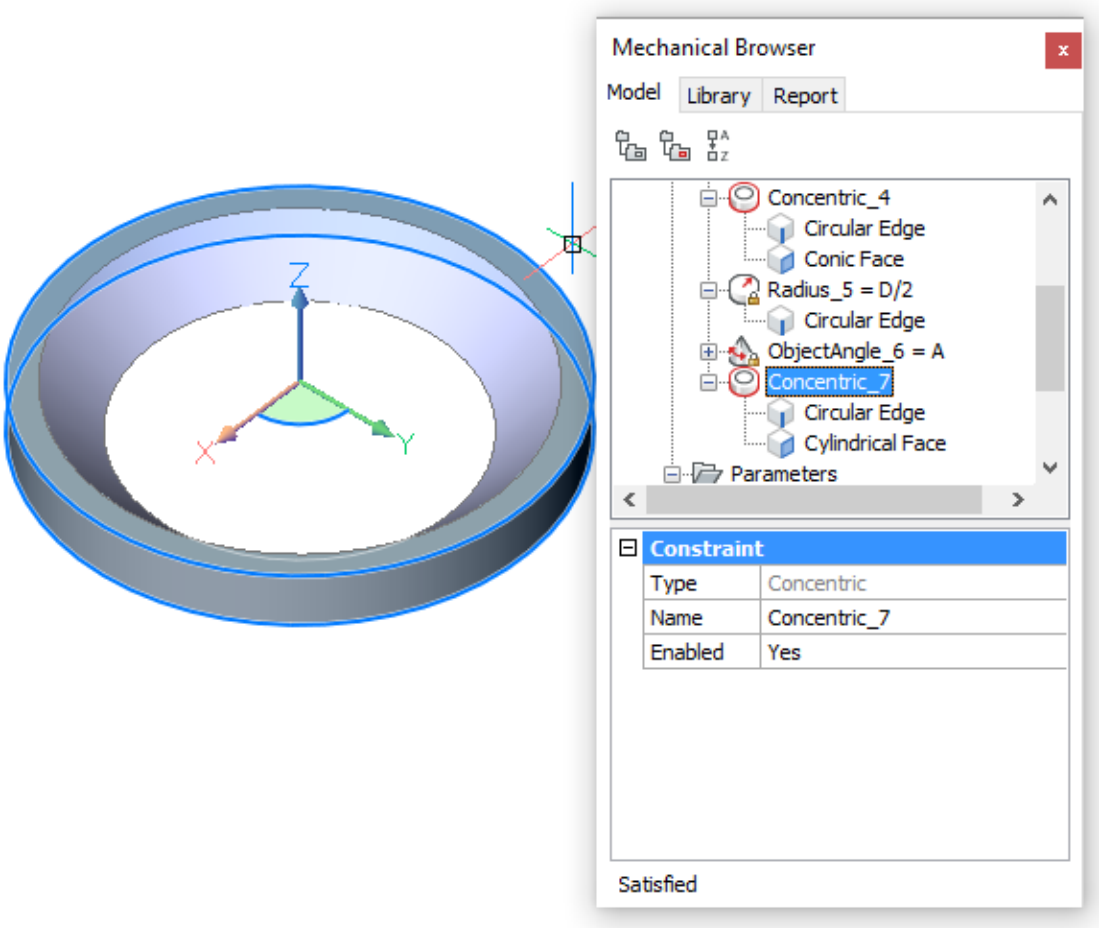

Mechanical Browser
Model
Library
Report
Concentric_4
Circular Edge
Conic Face
Radius_5 = D/2
Circular Edge
ObjectAngle_6 = A
Concentric_7
Circular Edge
Cylindrical Face
Parameters
Constraint
Type
Concentric
Name
Concentric_7
Enabled
Yes
Satisfied
Z
Y
X

Figure 3.482

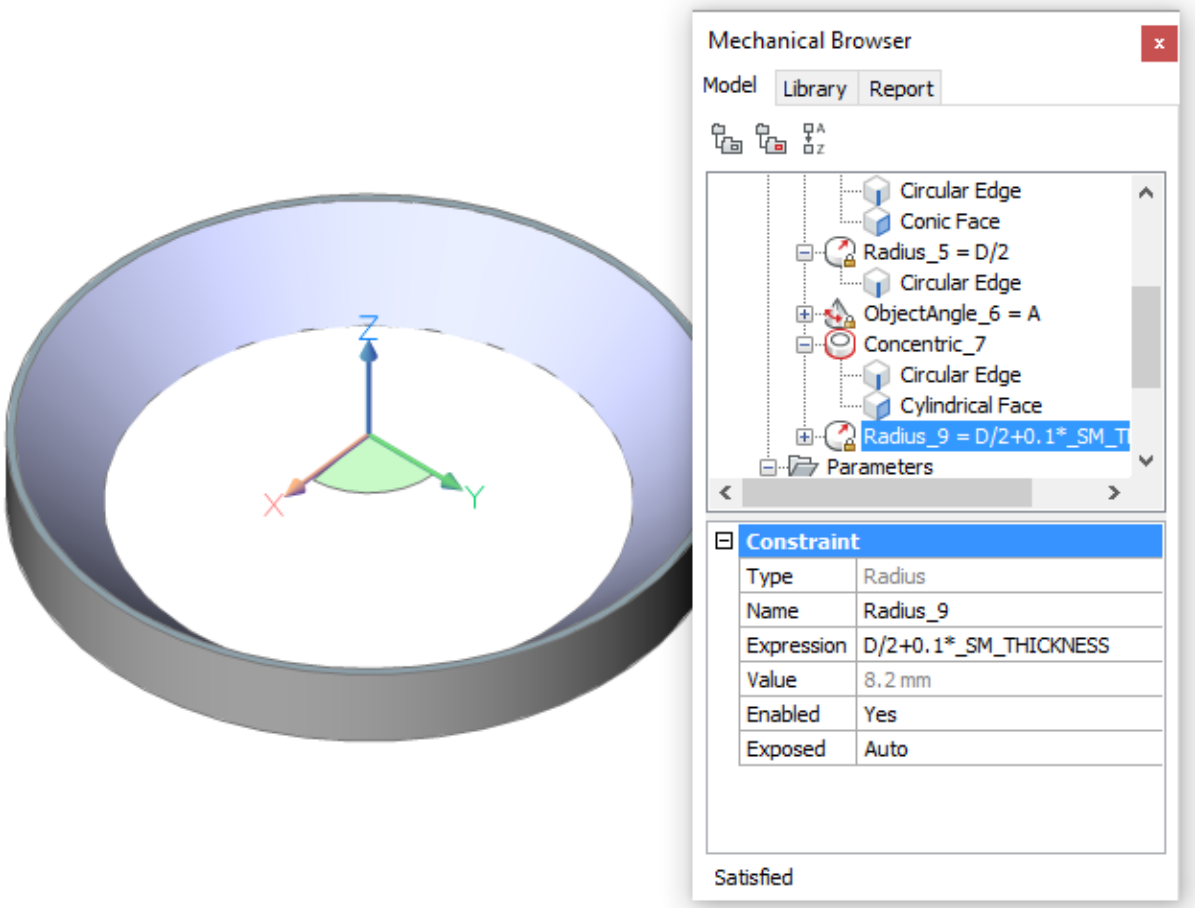


Figure 3.483

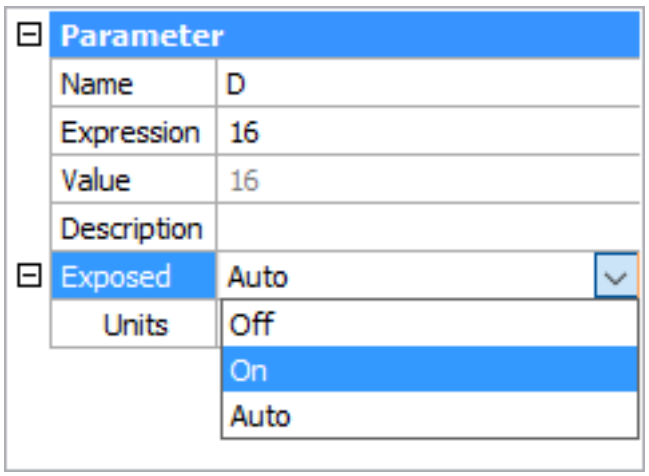
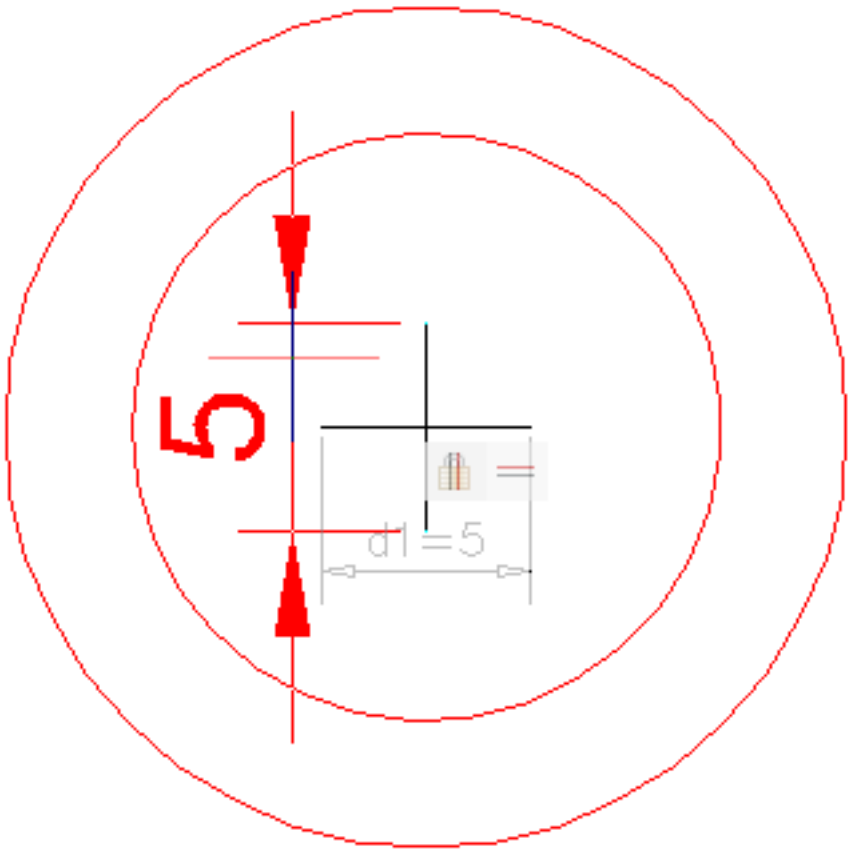


Figure 3.484



20. Make layer UNFOLD_SYMBOL_UP **current** and switch **off** all other layers.
21. Apply a GcFix (6.15.6) constraint ( ) to both crossing lines indicating the center of the circles.
22. Apply a GcHorizontal (6.15.7) constraint ( ) to the horizontal center line.
23. Apply a GcVertical (6.15.13)constraint ( ) to the vertical center line.
24. Apply a GcConcentric (6.15.3) constraint ( ) between the two circles.
25. Apply a DcLinear (6.10.39) constraint ( ) between the endpoints of the center lines. Accept the default value. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingS Distance.png})
26. Apply a DcDiameter (6.10.8) constraint ( ) to both circles. Accept the default value for dia1 (=inner circle) and dia2 (= outer circle). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Draw Diameter.png})
27. Adjust the **Expression** fields of the 2D constraints as follows:
    - d1 = 1
    - d2 = d1
    - dia 1 = D - 2*_SM_THICKNESS/tan(A)
    - dia 2 = D

    (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingS Values.png})
28. Switch **off** the display of all layers, but layer 0, then save the file. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h

The form feature can now be used to make conical parametric openings in a sheet metal flange. Use the **Insert Form Feature** tool ( ), which launches the BmInsert (6.8.12) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel Optionally: Add the form feature as a component to a tool palette (1.13.7).

Figure 3.485

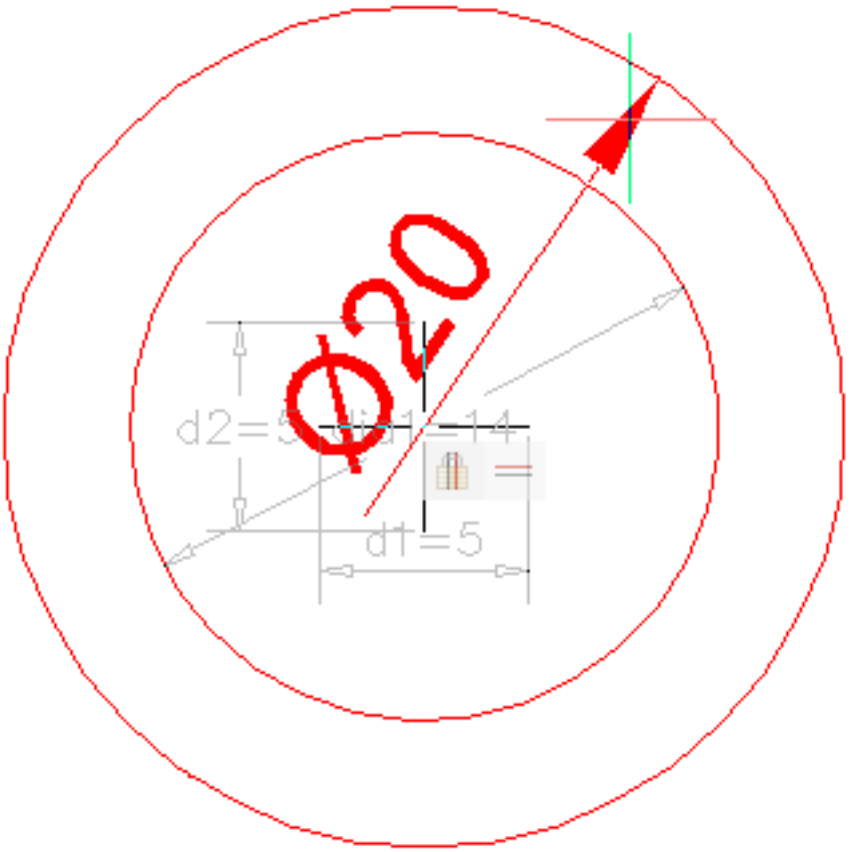


Figure 3.486

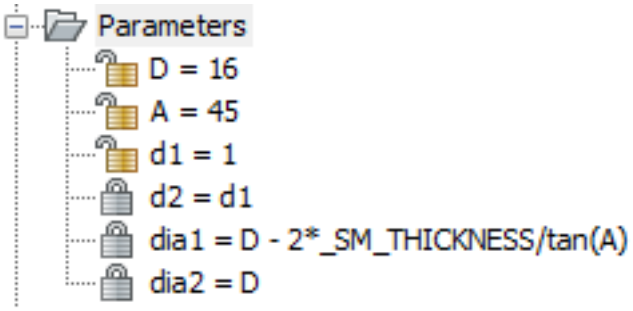


Figure 3.487

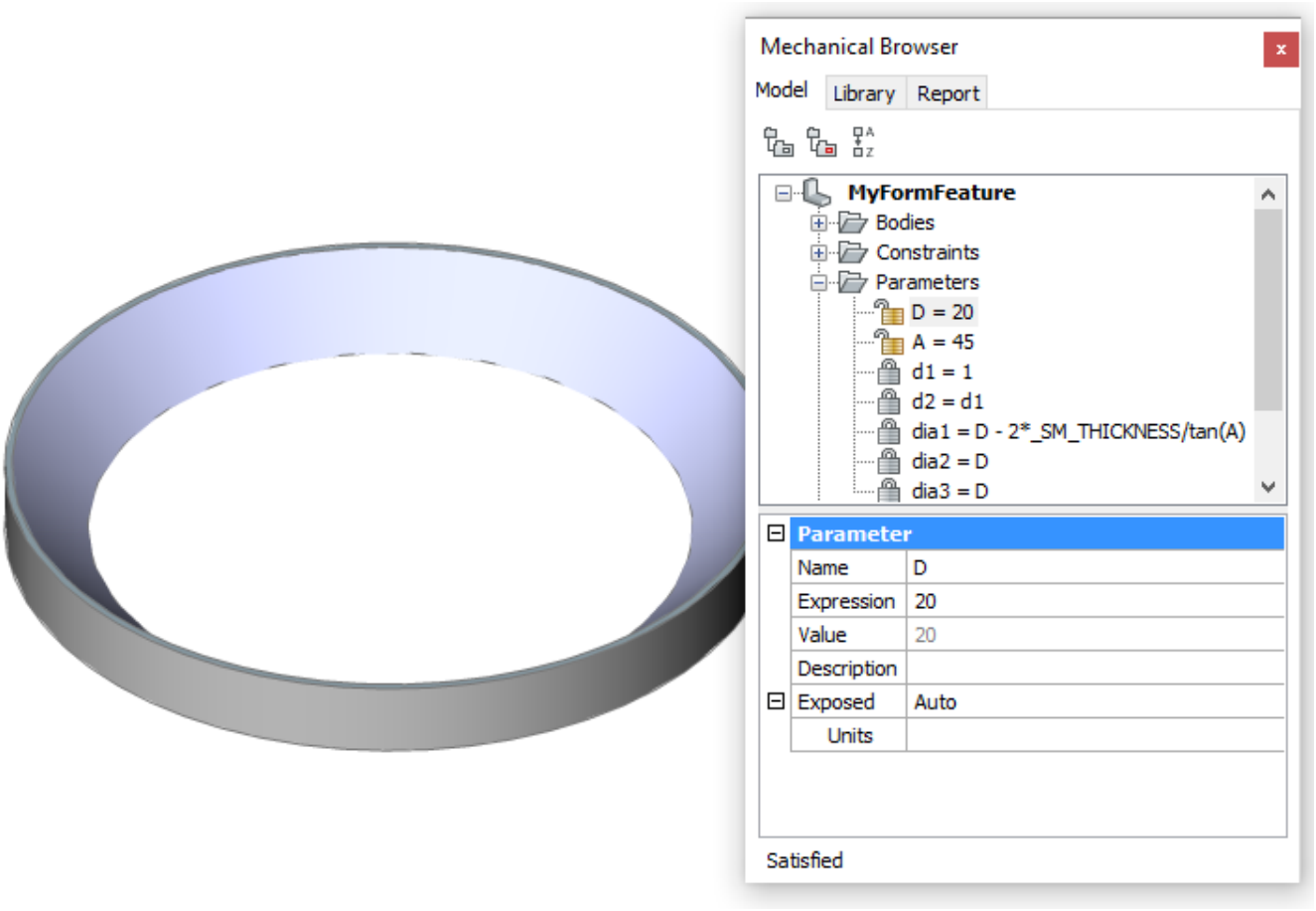
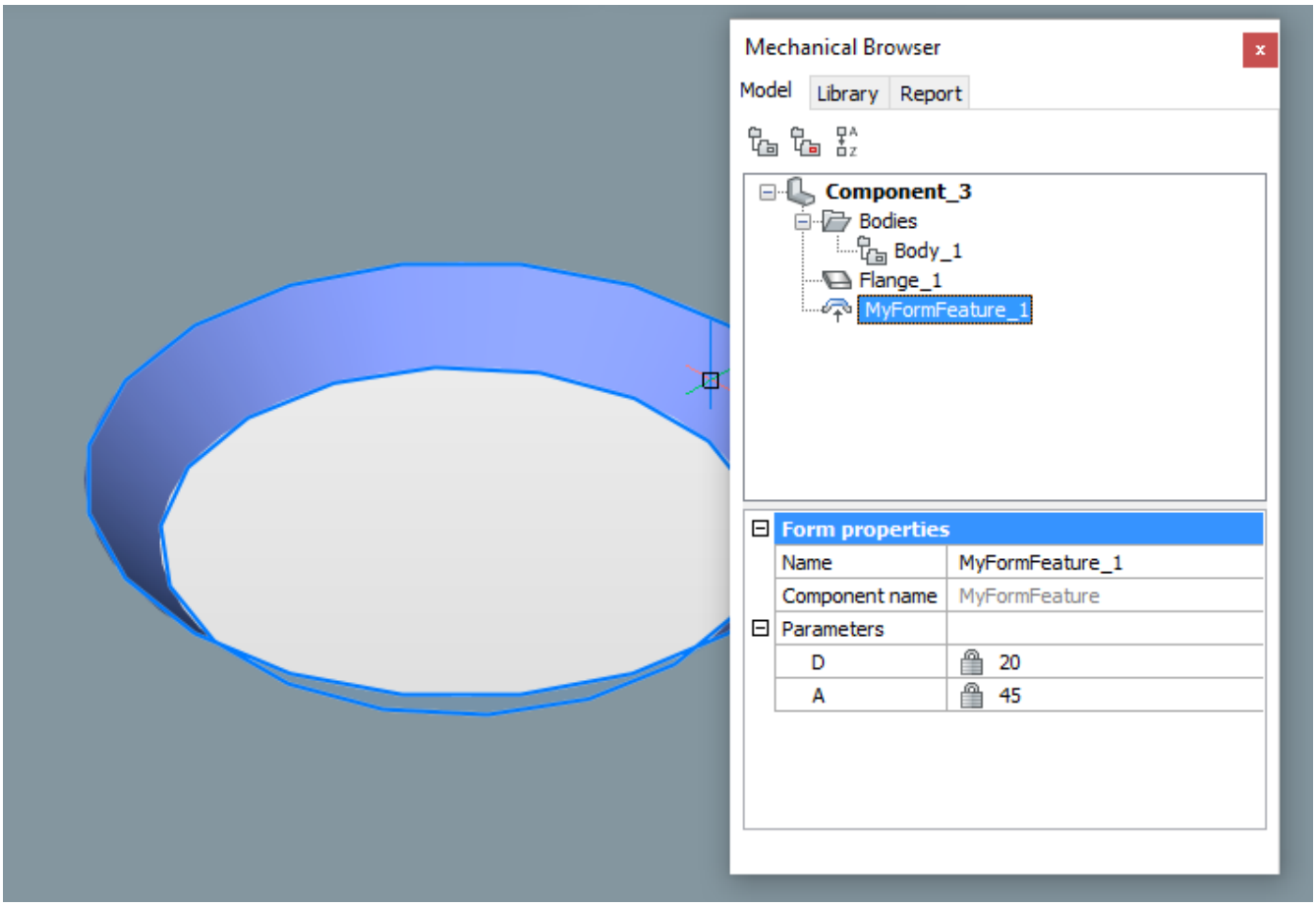


Figure 3.488



To create a rib feature

Rib features are created by theSmRibCreate (6.28.26) command from a 2D curve on flange face.

1. Do one of the following:

   - Click the **Create Rib Feature** tool button ( ) on the **Sheet Metal** toolbar.
   - Click the **Create Rib Feature** tool button ( ) on the **Modify** panel of the **Sheet Metal**tab on the ribbon.
   - Type **smribcreate** in the command bar, then press Enter.

   Prompts you: Select flange face:

2. Select the flange face. **Prompts you:** Select curve or [Profile radius/Round radius]:

3. Select a curve on the selected flange face. Lines, polylines, circles, arcs, ellipses, elliptical arcs, and splines are accepted. The 2D curves can be open or closed but cannot intersect itself or other form features. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/A
   After selection of the curve rib feature is created: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18

Figure 3.489

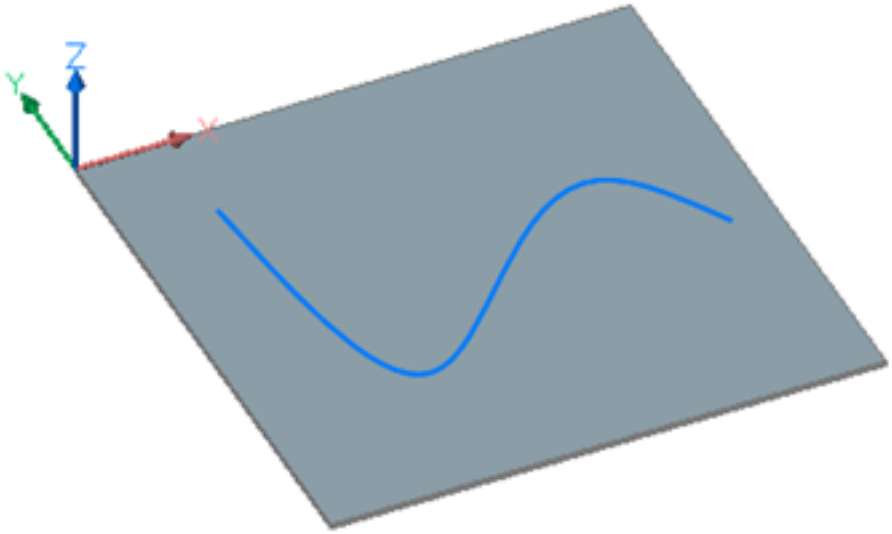
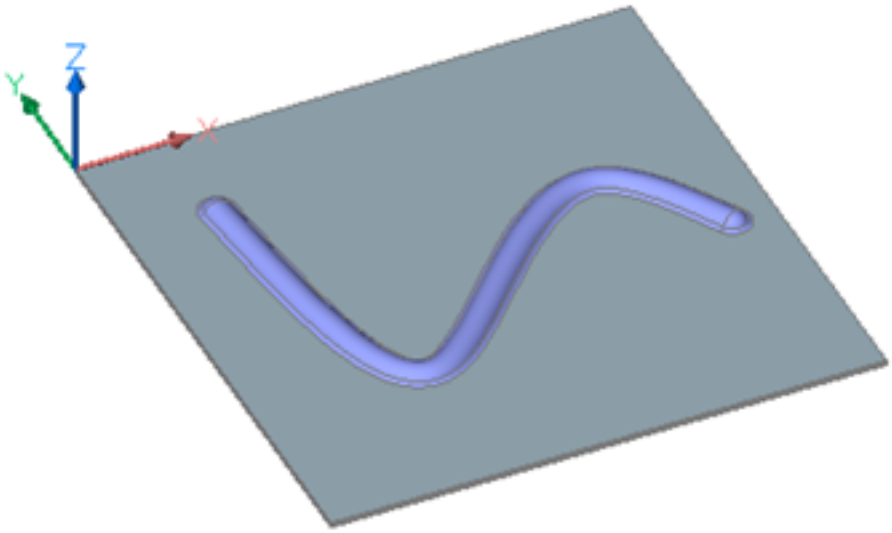


Figure 3.490



To modify a rib feature

To modify a rib feature:

- Modify the generatrix curve using its grip points.
- Edit the **Profile** and **Round** radius in the Mechanical Browser (3.2.2).

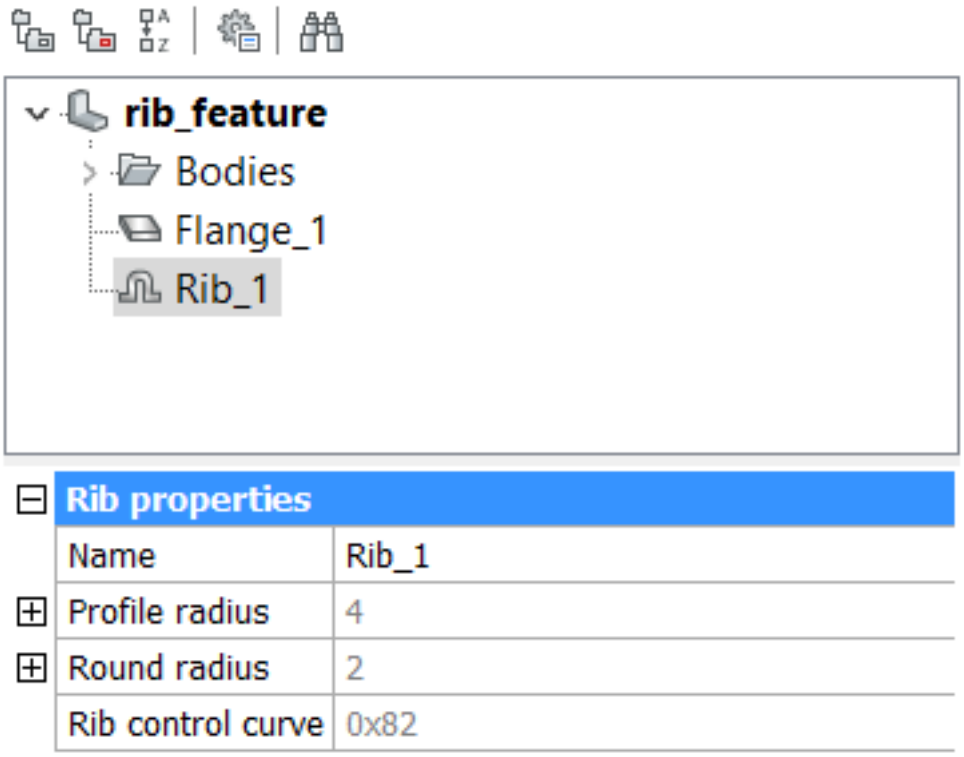
Modify the generatrix curve

1. Select the curve. Because the curve is hidden by the rib feature, press the TAB key to select obscured entities, or select the curve in the Structure (6.27.50) panel.
2. Grip edit the curve. The Rib feature updates according to the changes of the curve.
3. After selecting the flange face, **you are prompted:** Select curve or [Profile radius/Round radius]:
4. Select curve on selected flange side.

Edit the properties in the Mechanical Browser

1. Select the rib feature in the Mechanical Browser (3.2.2). The rib feature highlights in the model.
2. Select the property in the **Rib Properties** grid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Me
3. Type a value. The selected rib feature is updated.

Figure 3.491



Understanding the difference between Control and Trajectory curves

When creating a rectangular rib feature, we first draw a rectangle which is then used to create a rib feature bytheSmRibCreate (6.28.26) command. When this control curve is grip edited, the rib feature is updated accordingly. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/mceclip0.png}) On the production side, the rectangular control curve is replaced by a filleted trajectory curve, according to the physical limitations of the wheel tool: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/mc
The trajectory curve, however correct, is hard to manipulate, e.g. when you want to move a corner. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/mceclip2.png}) In the design stage we use the**Control Curve**, which isas simple as possible: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/
When creating the rib feature,SmRibCreate (6.28.26)adds fillets according to the value of the**Rib fillet radius** parameter inMechanical Browser (3.2.2). This parameter is not controlled per feature, since it relates to the properties of physical wheel tool: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/mceclip
When zooming in closely you notice the difference between the 3D geometry of the rib feature and the control curve: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/mceclip5.png}) When the Polyline is grip edited ... (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/
... the rib feature is adjusted accordingly: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/
This illustrates the associativity between the 2D curve and 3D rib feature. The recognition of rib features inSmConvert (6.28.4) is controlled by the **SmConvertRecognizeRibControlCurves** user preference. When the part is unfolded by theSmUnfold (6.28.29) command, the curves follow the 3D design and have the fillets.The middle curve is called **Trajectory Curve**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/mc

Figure 3.492



Figure 3.493

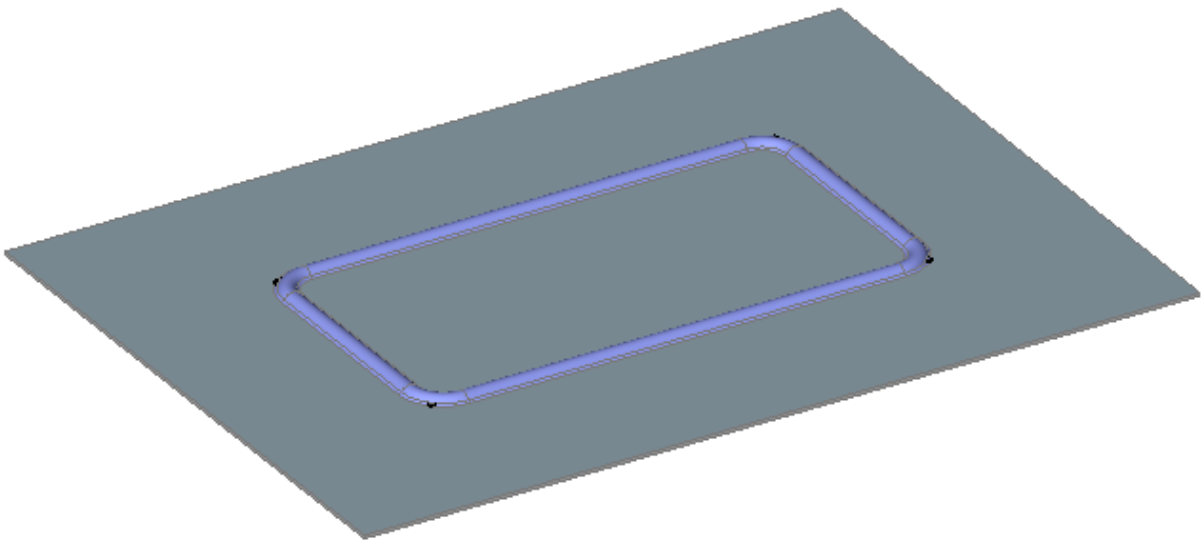


Figure 3.494

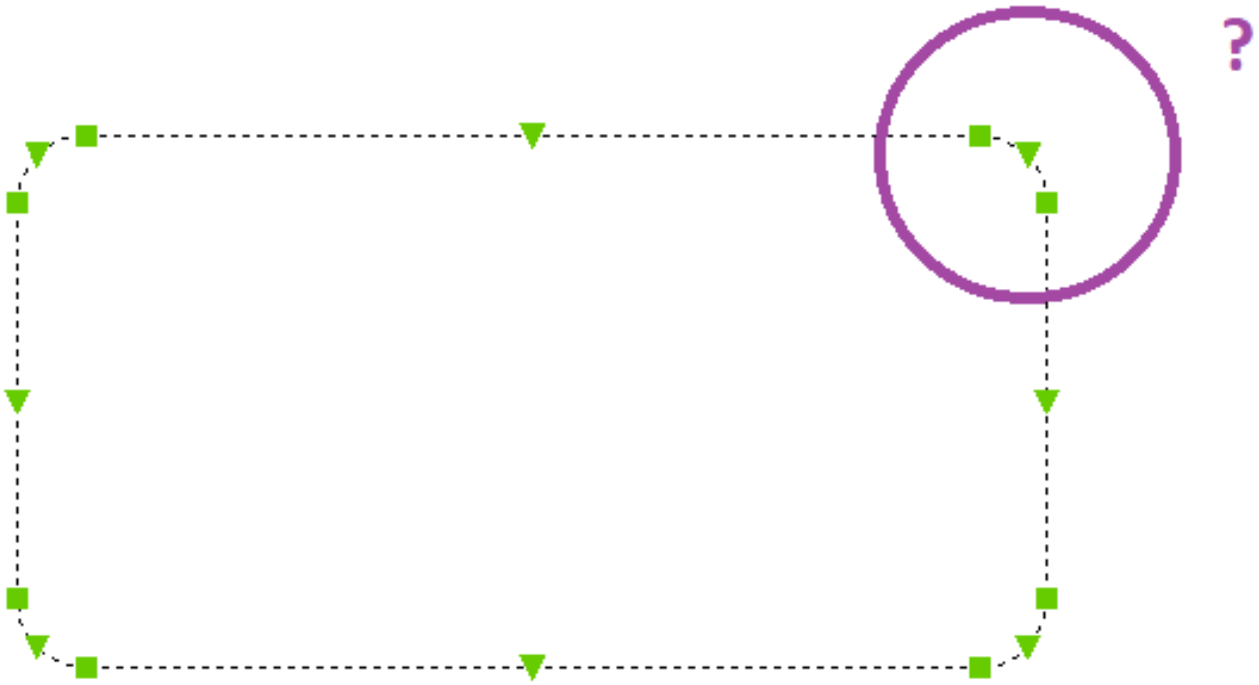


Figure 3.495

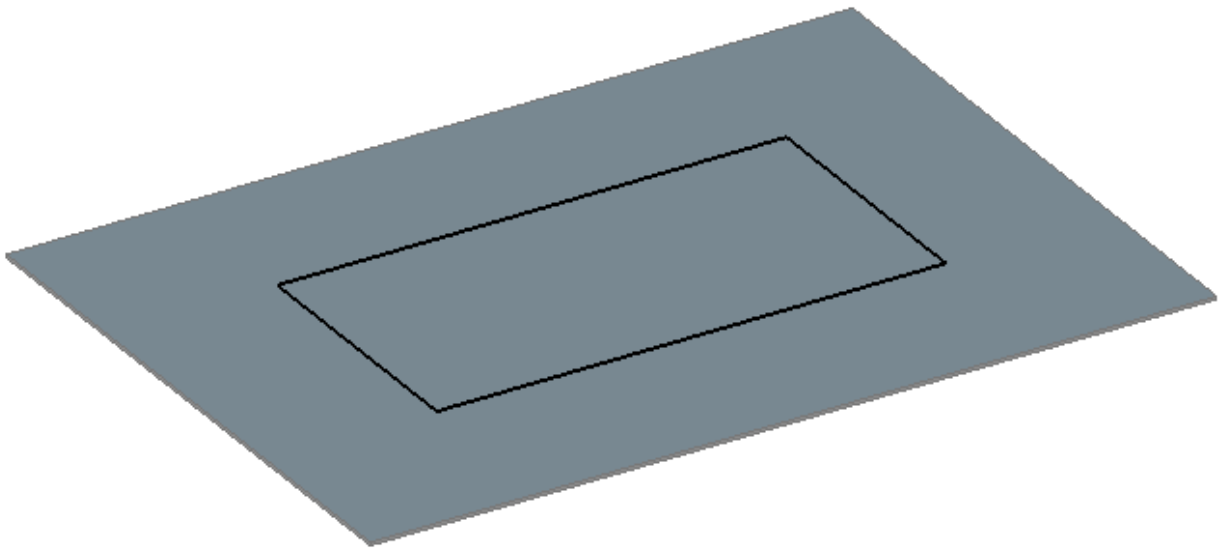


Figure 3.496

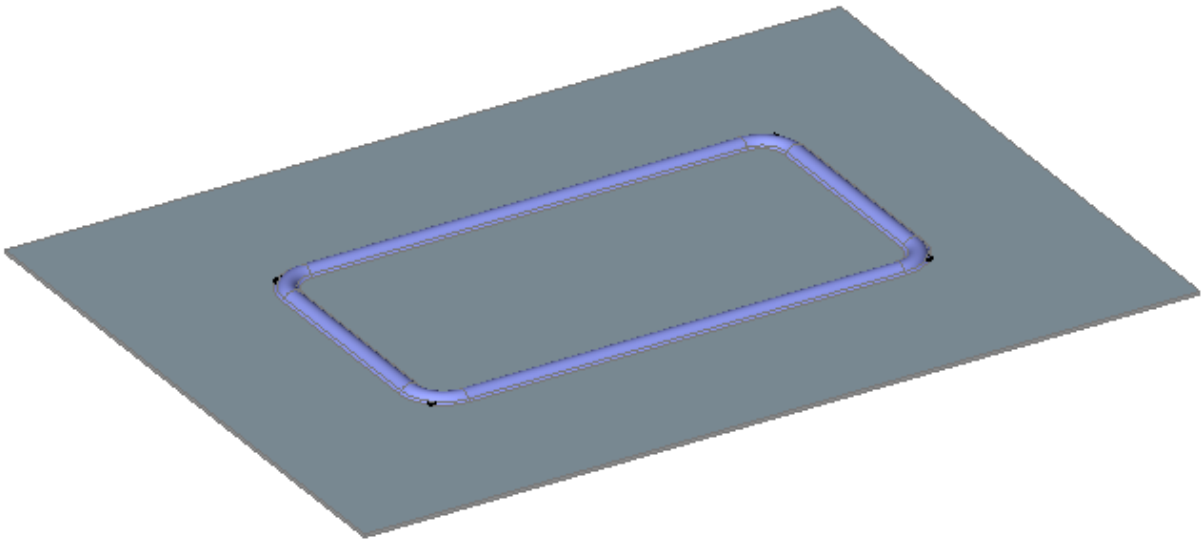


Figure 3.497

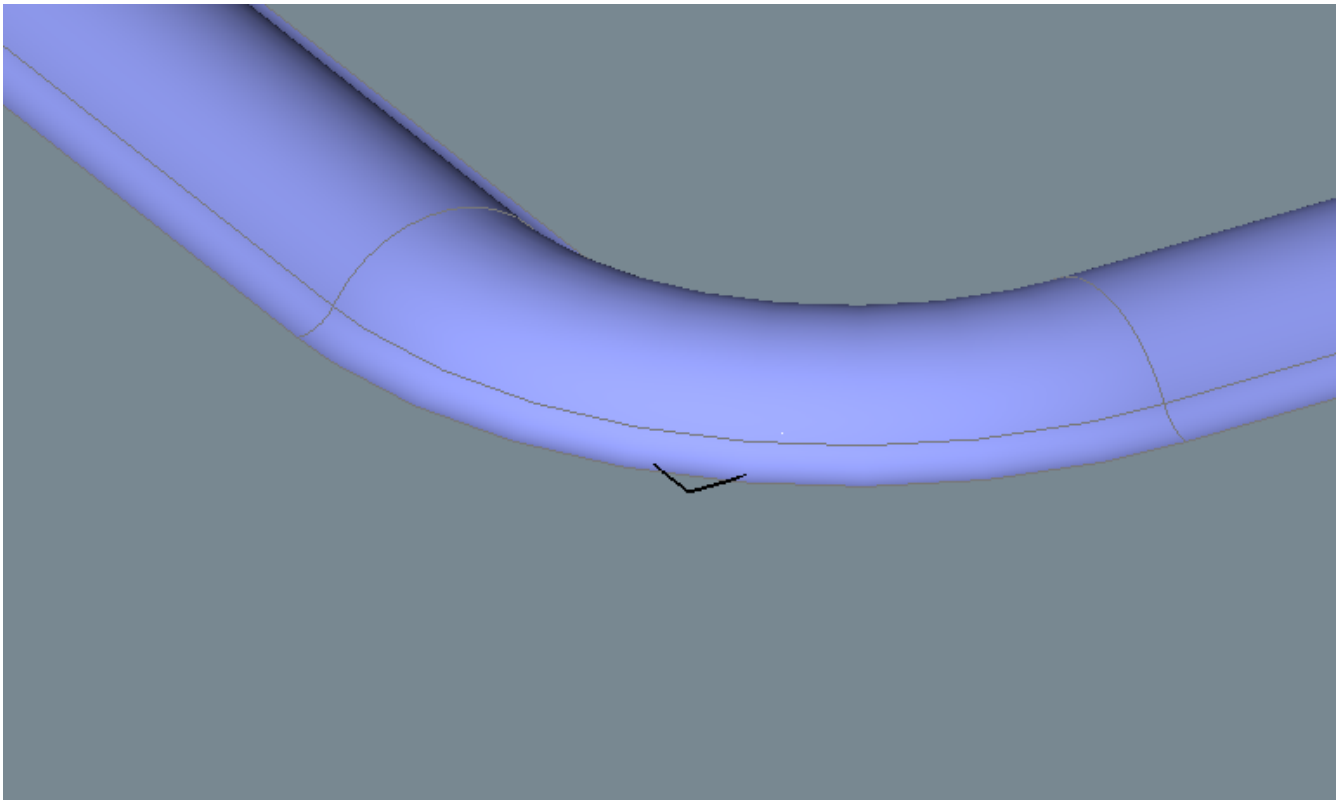


Figure 3.498

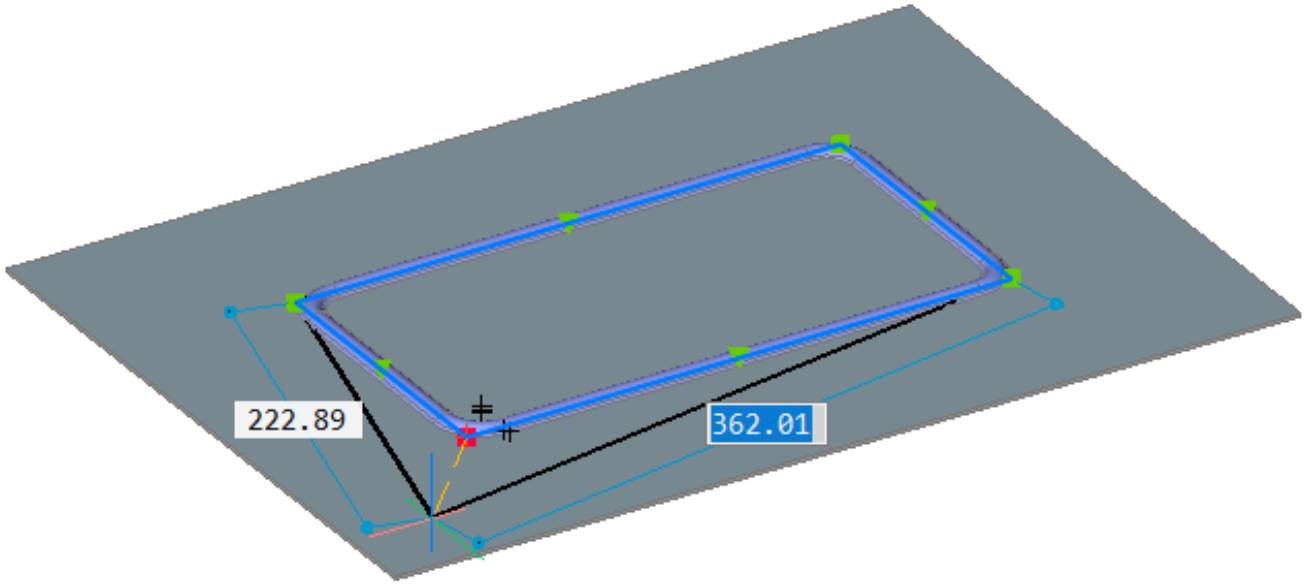


Figure 3.499

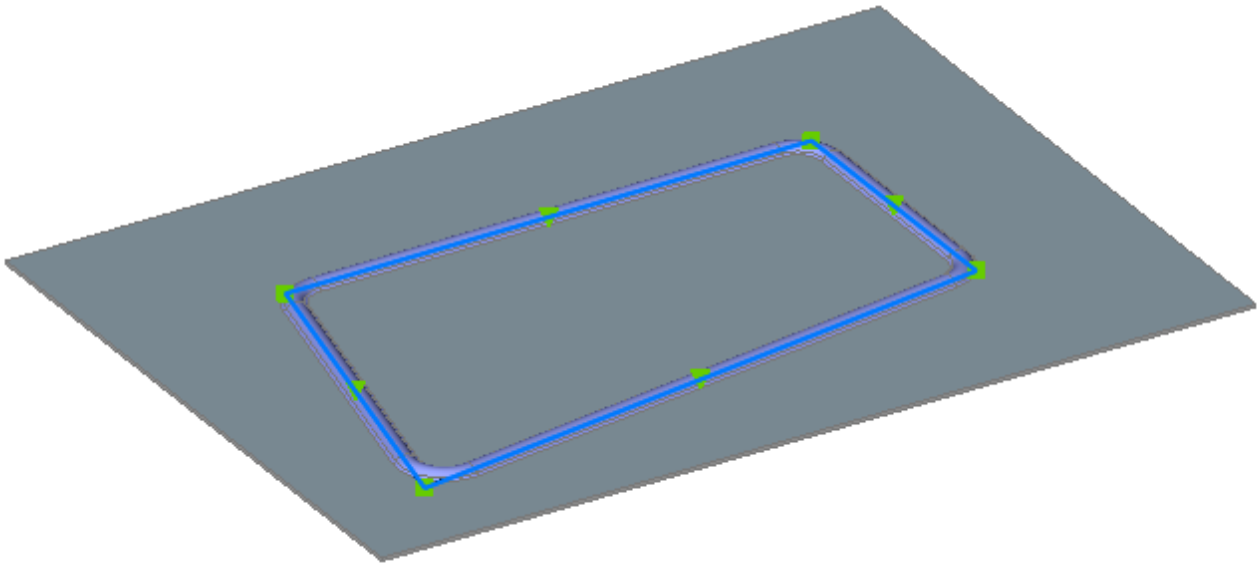


Figure 3.500

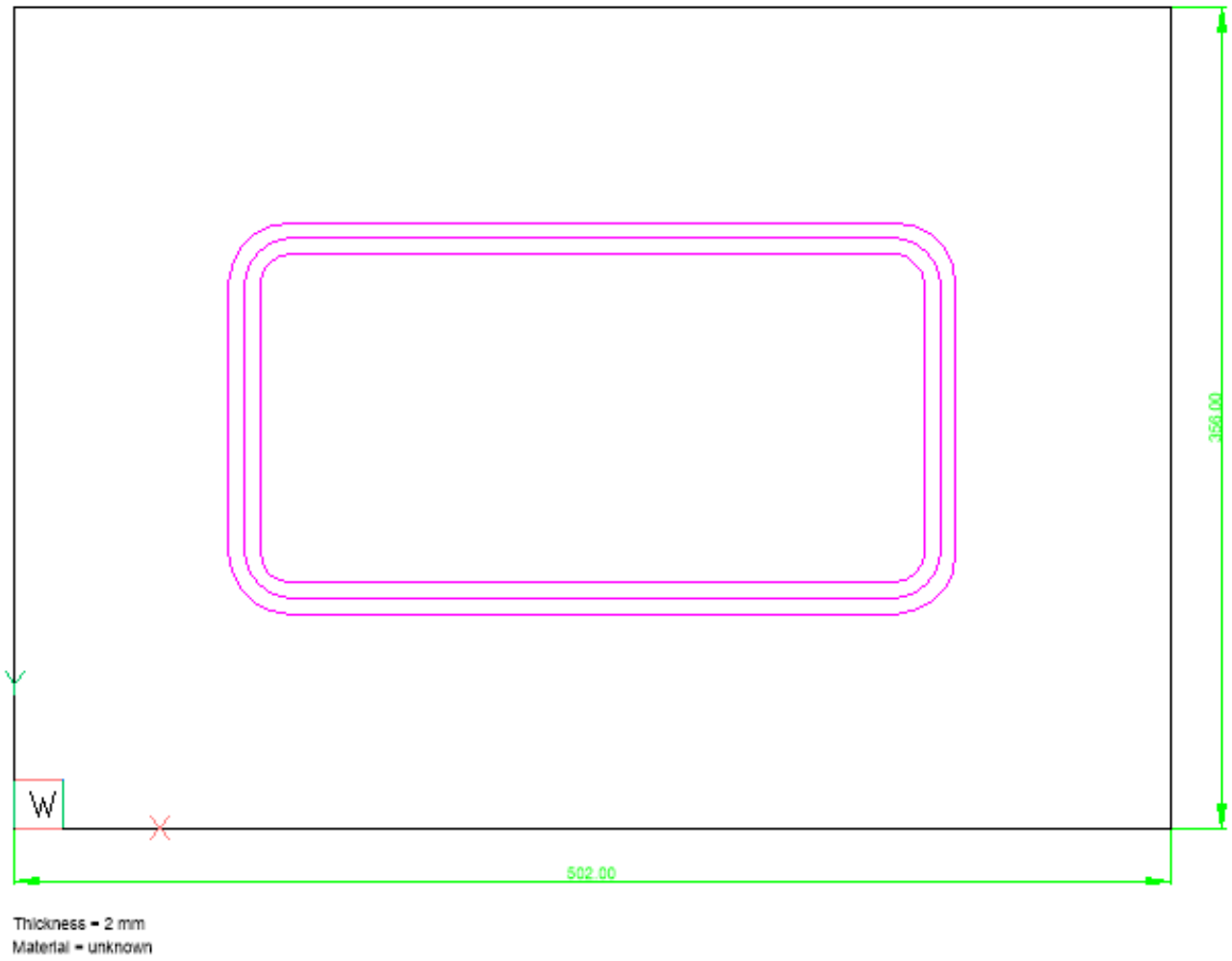
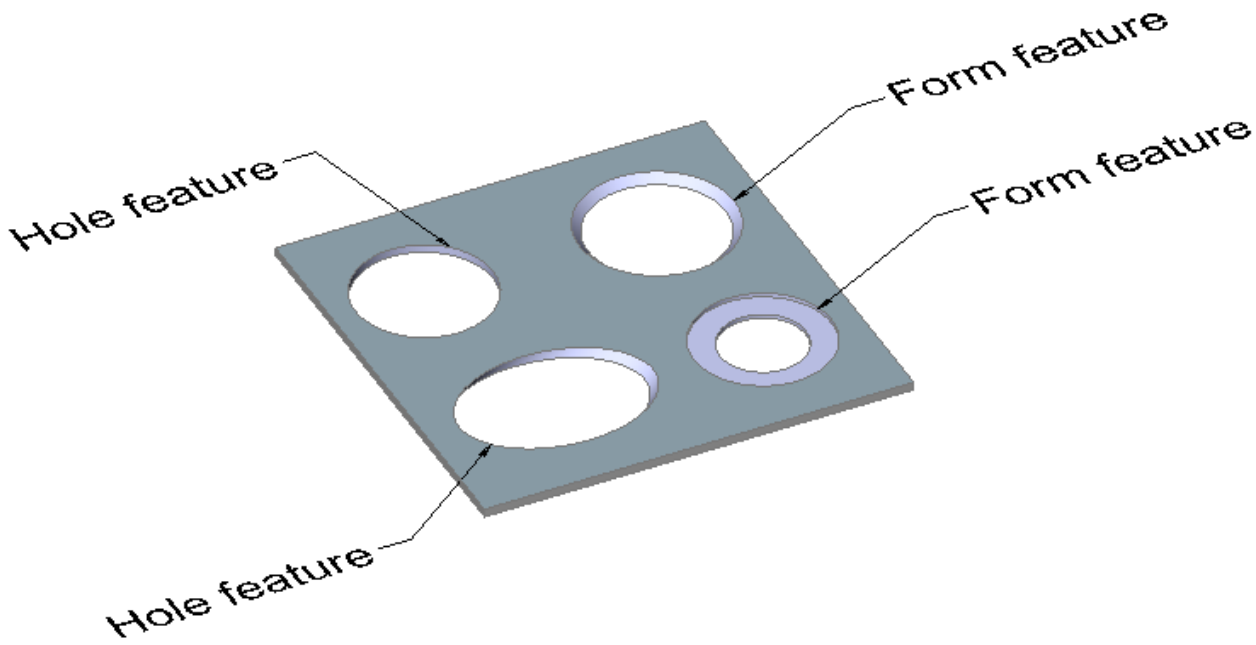


Figure 3.501



3.8.19 Working with Hole Features

Hole features are asimplified version of**Form Features**. In the image below you can see different features which have only thickness faces (theSmConvert (6.28.4) commandrespects these rules):

- Orthogonal holes are hole features
- Simple non-orthogonal holes are hole features
- Chamfered orthogonal holes are form features
- If the hole thickness face consists of two different regions, it is form feature

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/HoleFeaturesHoleFormFeat
Hole features are recognized by theSmConvert (6.28.4)command if theSmConvertRecognizeHoles (1.13.11) user preference is ON. The following basic operations are allowed for hole features:

- SmDelete (6.28.5),SmDissolve (6.28.6), disable andSmSelect (6.28.27) them using commands or in theMechanical Browser for Sheet Metal (3.8.5).
- Form Feature Modes in theSmUnfold (6.28.29) command, including **Symbol** mode.
- Associate your own 2D geometry in a flattened layout (see forWorking with Form and Rib Features (3.8.18) workflow of creation form feature component for creation of unfold symbol).
- Hole features are preserved by theSmRepair (6.28.24) command. Plain hole faces become orthogonal. A hole feature of the same geometry is not affected by theSmRepair (6.28.24) command.

Recognition of Arrays of Hole Features

When hole features exist in a body,SmParametrize (6.28.21) detects rectangular arrays of holes on flanges, according to the following rules:

- holes belong to same flange
- holes are on a rectangular grid
- there are no gaps (missing elements) in the array
- holes are equally oriented
- the minimal array size is either 1x3 or 2x2

Figure 3.502

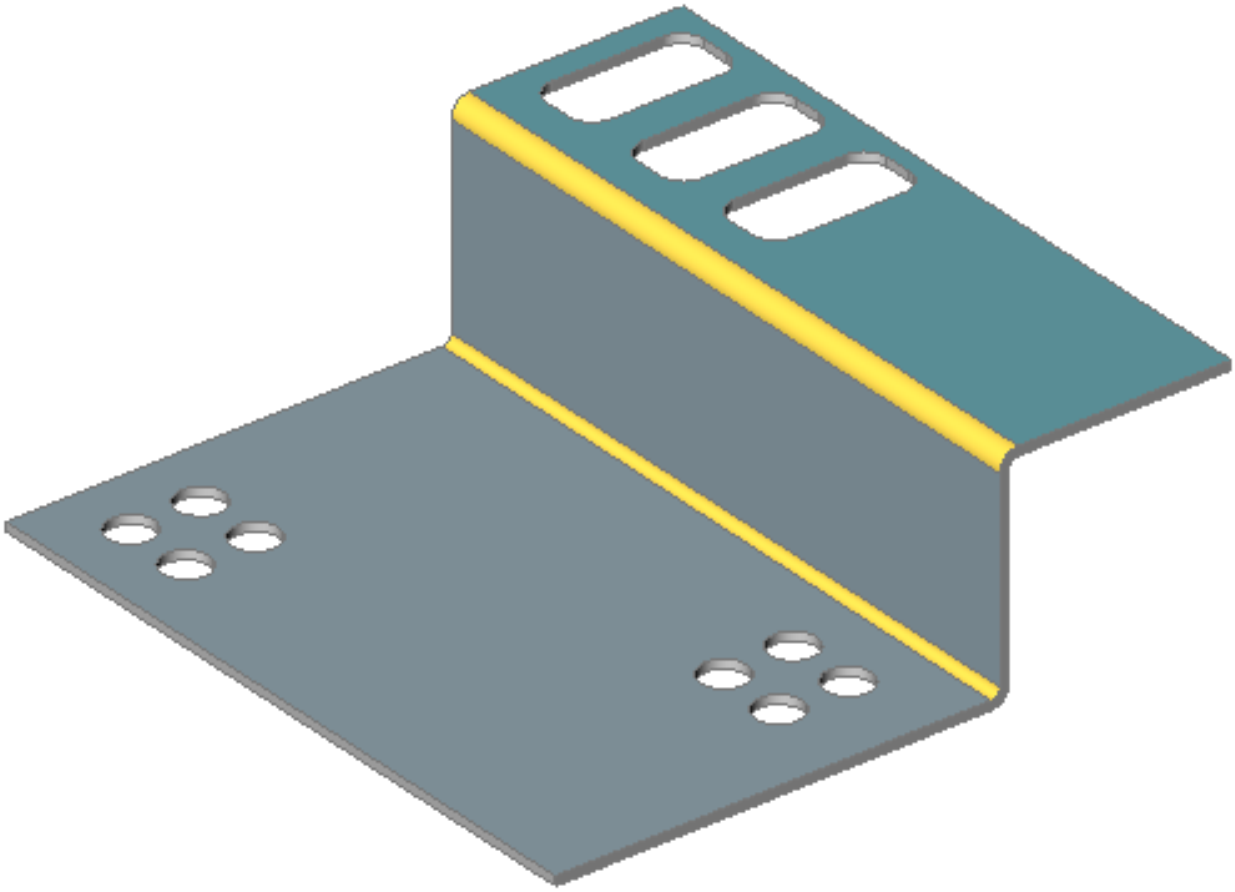
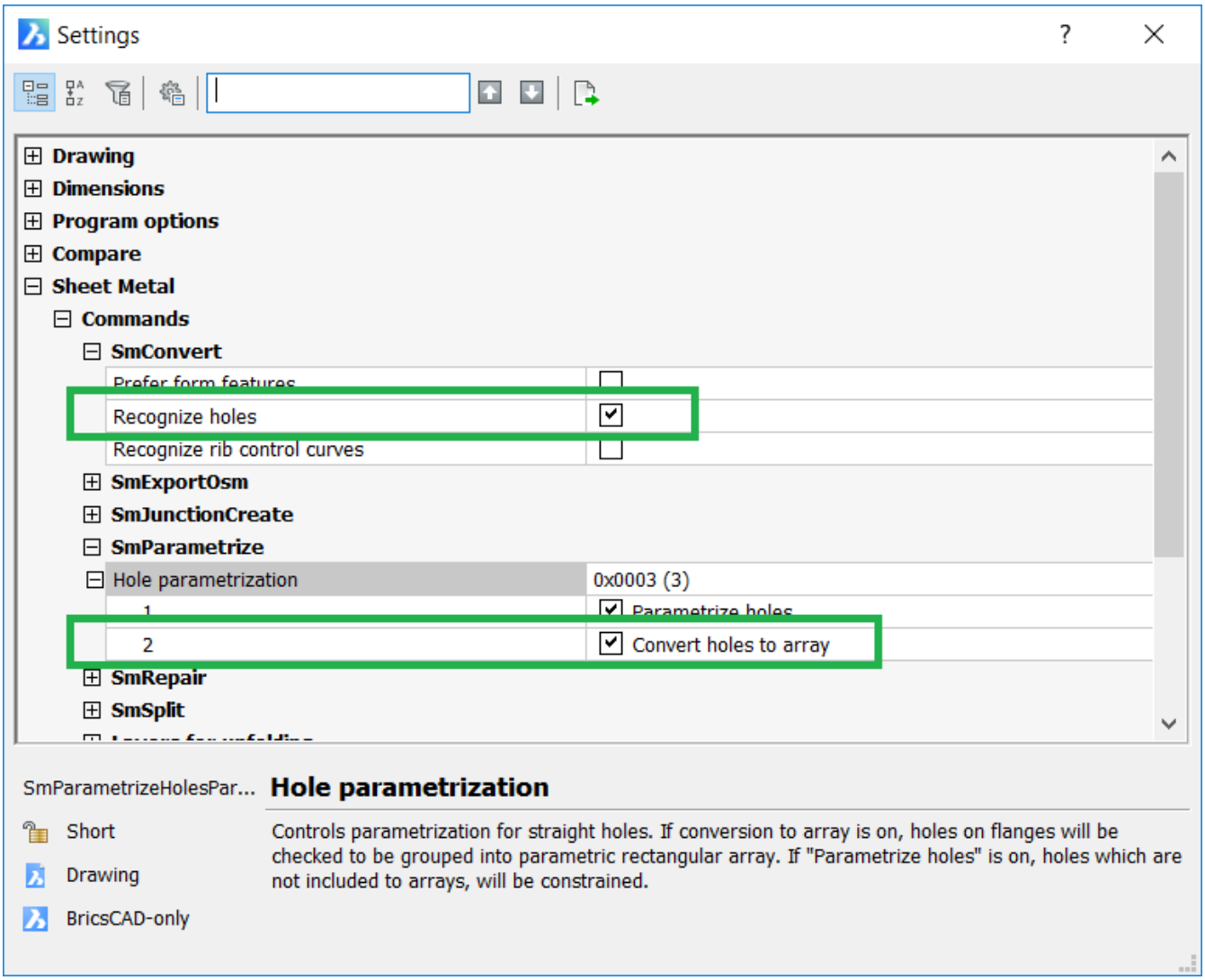


Figure 3.503



Lets illustrate the workflow on a part with Flange and Bend features. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he

1. In the Settings (1.13.11) dialog go to **Sheet Metal/Commands**.
2. Enable the **Reconize holes**option for the SmConvert (6.28.4) command.
3. Enable the **Convert holes to array**and Parametrize holes options for theSmParametrize (6.28.21) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/HoleFeaturesSettings.
4. Run theSmConvert (6.28.4)command.
5. Do one of the following:
   - Click the **Convert to Sheet Metal** tool button ( ) on the **Sheet Metal**toolbar.
   - Click the **Convert to Sheet Metal** tool button ( ) on the **Sheet Metal** ribbon bar.
   - Choose **Convert to Sheet Metal** in the **Sheet Metal** menu.

   You are prompted: Select 3D solids or [Entire model] <Entire model>:
6. Select the 3D solid to convert. **You are prompted:**Select 3D solids or [Entire model] <Entire model>:
7. Press Enter to convert the 3D solid to sheet metal part. Eleven hole features are recognized on the part. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/HoleFeaturesZ2.png})
8. Run theSmParametrize (6.28.21) command.

Figure 3.504

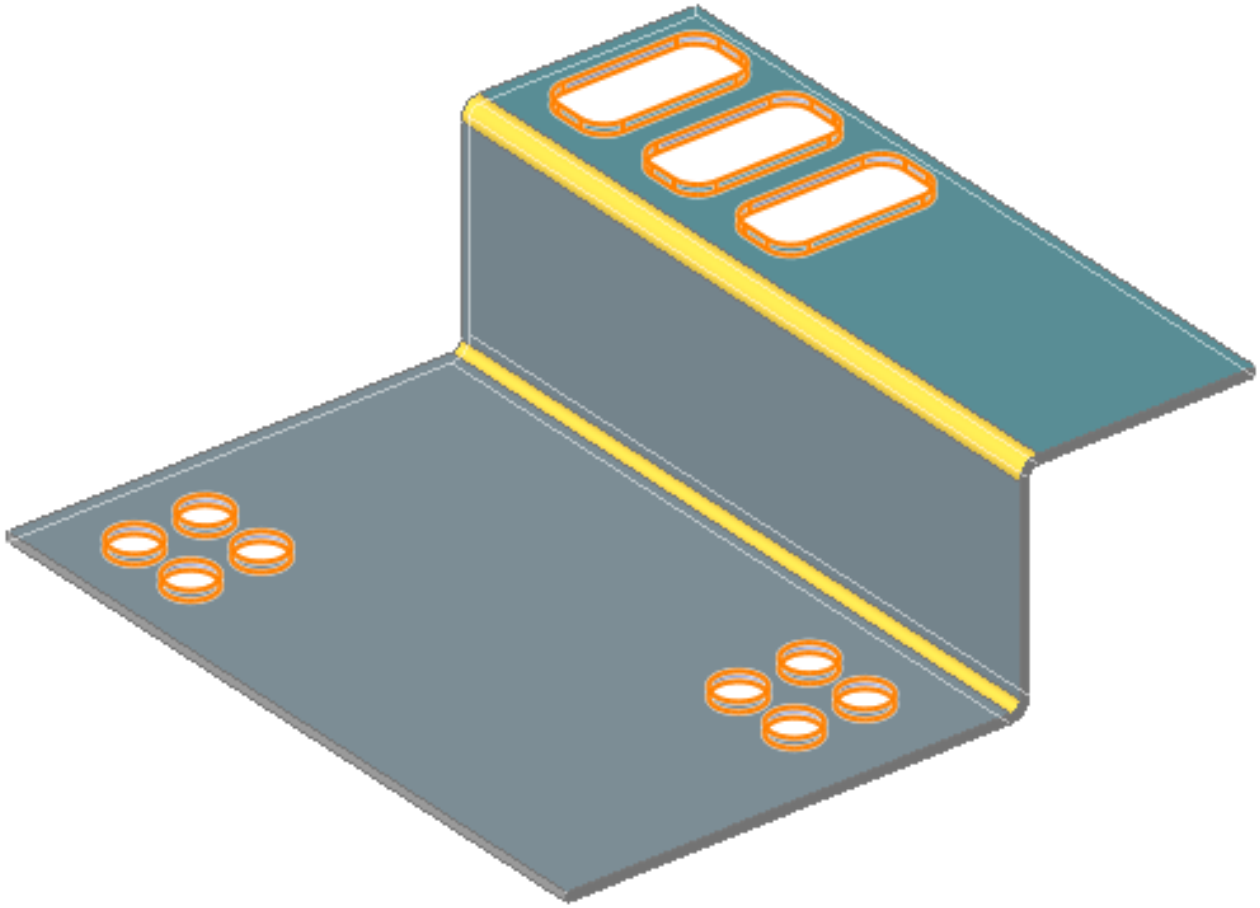
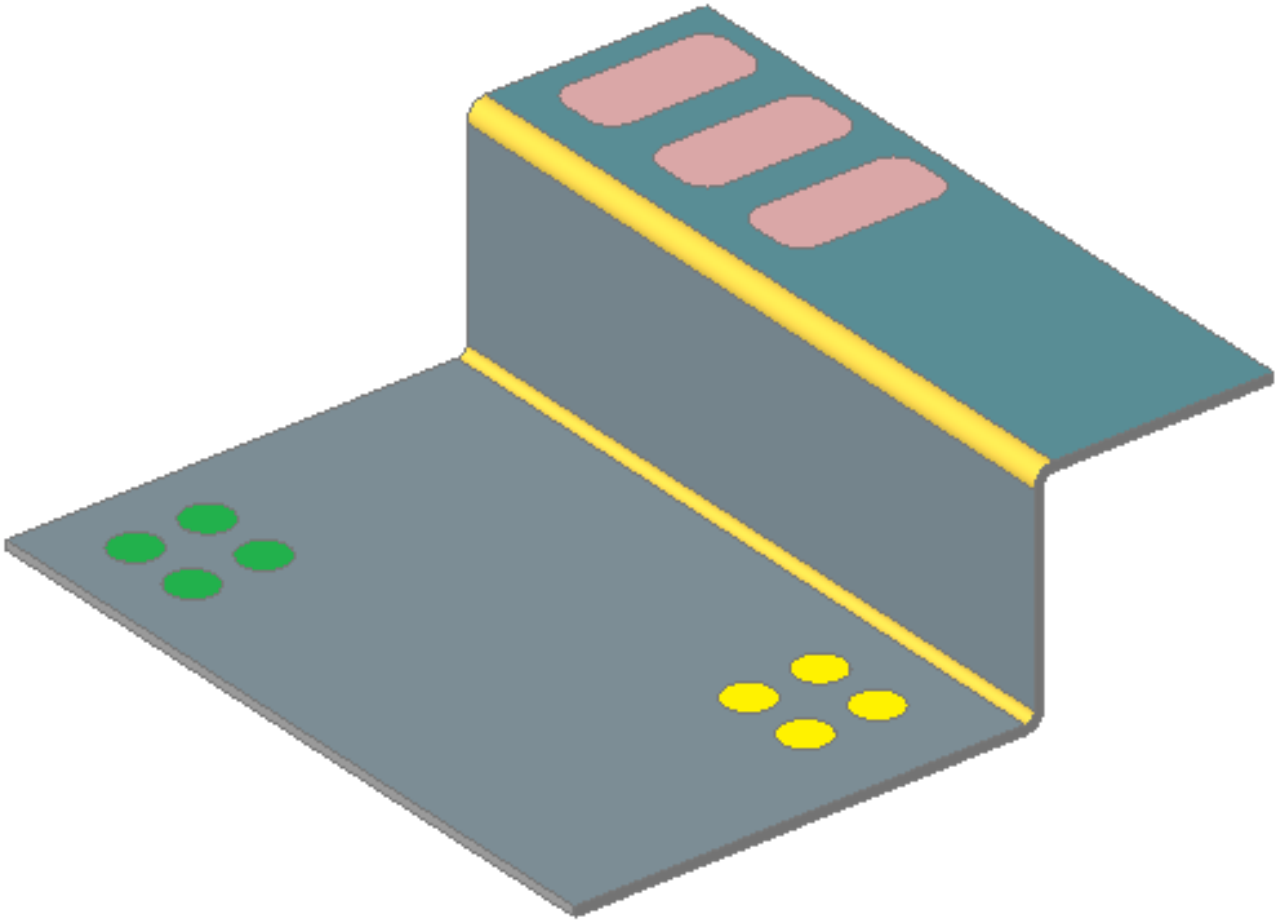


Figure 3.505



9. Do one of the following:

   - Click the **Parametrize** tool button ( ) on the **Sheet Metal** l toolbar.
   - Click the **Parametrize**tool button ( ) on the **Sheet Metal** ribbon bar.
   - Choose **Parametrize**in the **Sheet Metal** menu.

   You are prompted: Select 3D solids to parametrize or [Entire model] <Entire model>:

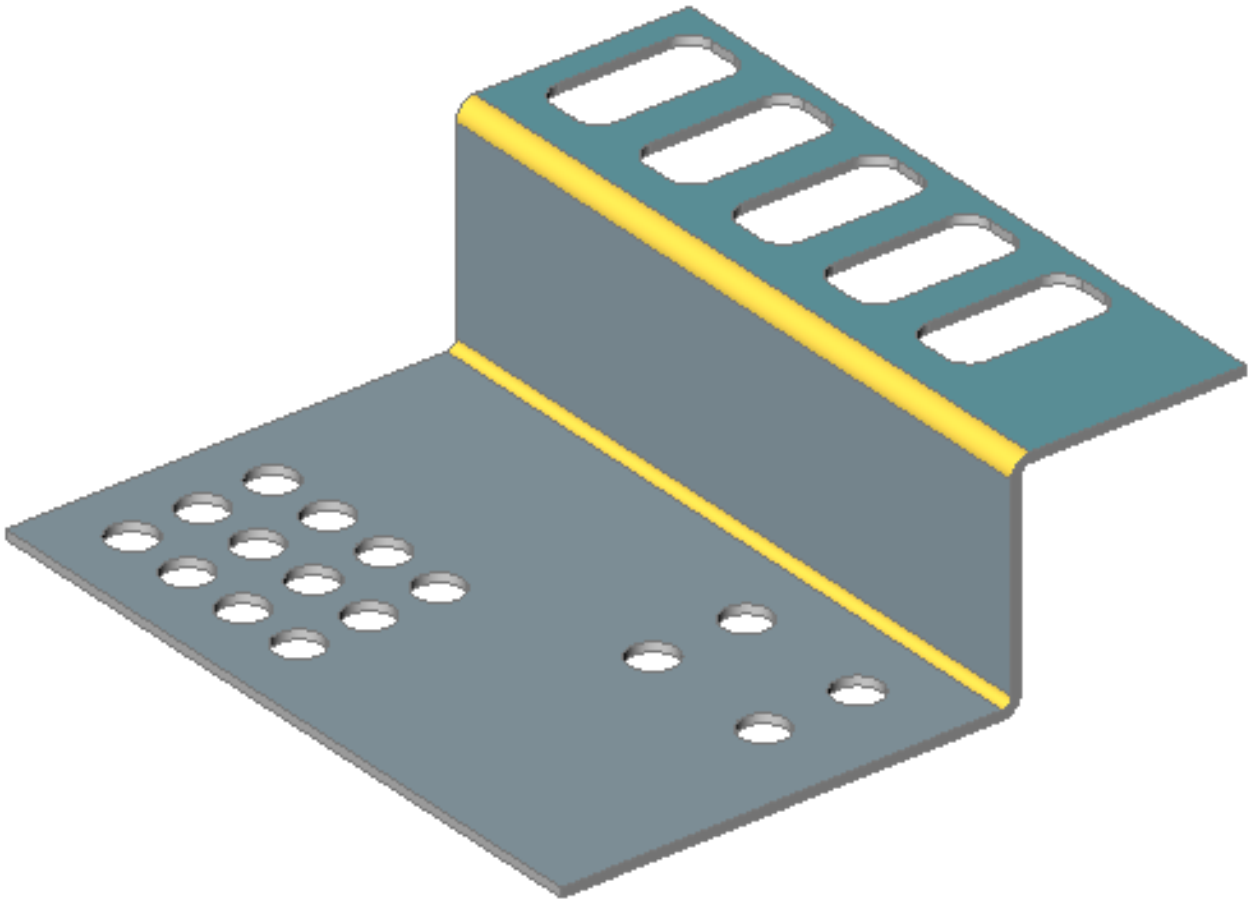
10. Press Enter process the sheet metal part. The command reports should look like the one below: Created 4 distance constraints Created 3 fix constraints Created 0 coincident constraints Created 0 tangent constraints Created 0 rigid set constraints Created rectangular array 2x2 basing on Hole_3 feature Created rectangular array 2x2 basing on Hole_7 feature Created rectangular array 3x1 basing on Hole_9 feature Total: 7 constraints and 3 array(s) created (Feature names and constraint distribution may differ depending on the BricsCAD version): To examine the arrays you can temporarily unfreeze theBC_SUBTRACT (1.22.22)layer. In the image below the arrays are colored manually. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical
    Notice that the green and yellow arrays are not unified. Otherwise to array would have missing holes.

11. Edit theArray Properties (3.6.5) in theMechanical Browser (3.2.2). In the image below the following has been modified:

    - Green array:number of rows and columns from 2x2 to 3x4
    - Yellow array: X and Y spacing
    - Red array:number of rows and columns from from 3x1 to 5x1

    (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/HoleFeaturesZ4.png})

Figure 3.506



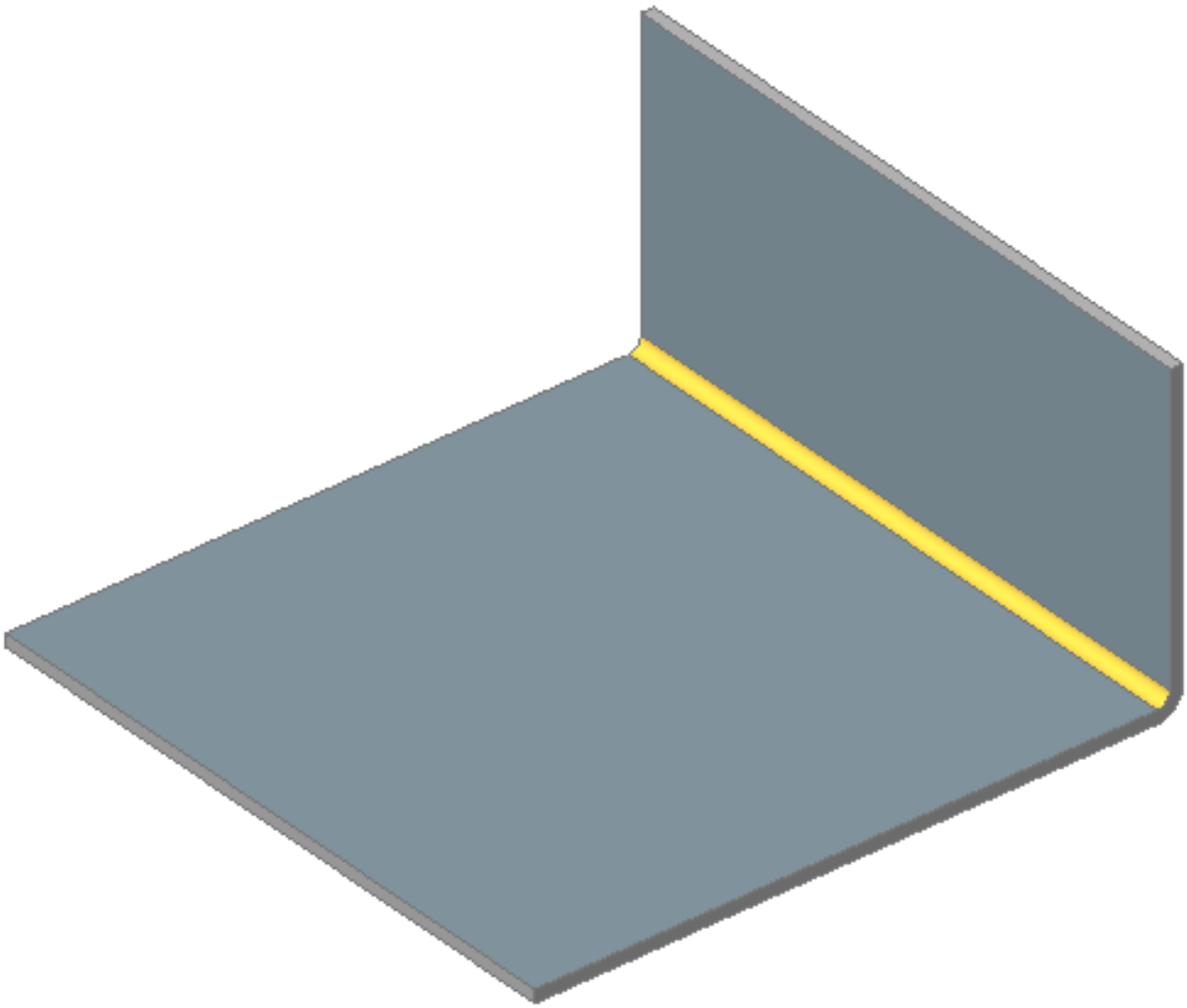
3.8.20 Modifying Flanges

BricsCAD provides several tools to modify flanges in sheet metal parts. These tools can be applied consistently to any part - independent of its design history. In this article we useSolid Modeling (3.4.3) tools and specific sheet metal tools. Almost all solid modeling tools apply to sheet metal parts, while preserving the design intelligence provided bySheet Metal Features (3.8.3).

To change flange size and shape

On a given simple model we will apply severalSolid Modeling (3.4.3) operations in order to modify the shape of its flanges. Please note that we work on thickness faces of the model only: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

Figure 3.507



To stretch flanges

1. Hover over the thickness face, then the select PushPull (6.12.20) in the **Model** command group in the Quad. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/ModifyinFlangesChan

2. Drag the face then right click. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/Sheet

3. The changes will be applied to the part, the flange is stretched over the specified distance.

Figure 3.508

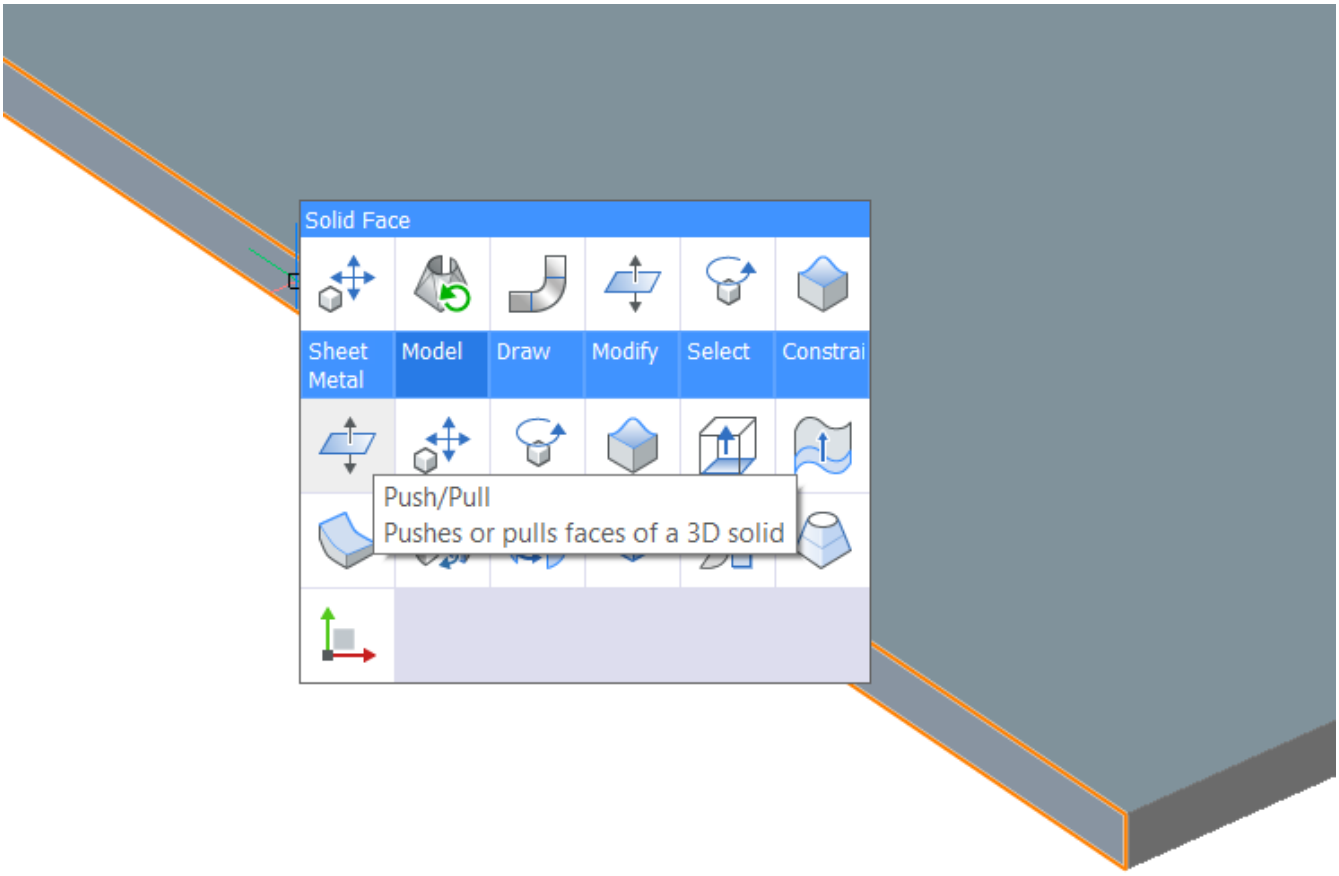
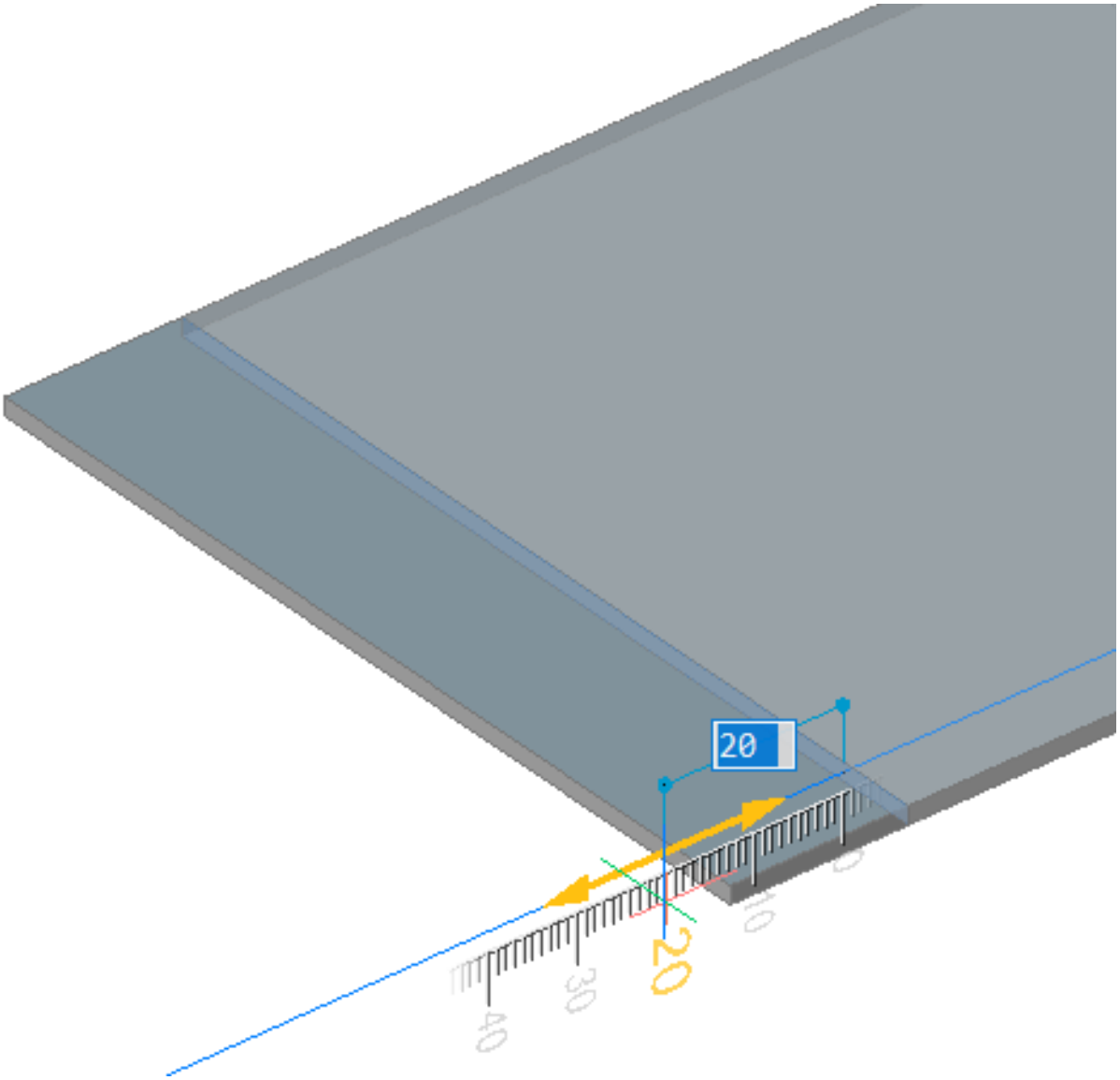

Solid Face
Sheet Metal
Model
Draw
Modify
Select
Constrai
Push/Pull
Pushes or pulls faces of a 3D solid

Figure 3.509



To rotate a thickness face

1. Hover over the face and choose Rotate (6.12.24)in the **Model**command group in the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/ModifyinFlangesChangeRot
2. Specify the rotation axis, base point and angle (for more details see theDmRotate (6.12.24)article in te Command Reference): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/Mod
3. We will now fillet the sharp edges. Hover over a thickness edge and choose **Same Length Edges**()in the **Select**command group in the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mec
4. Press Enter. Edges with the same length are selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS
5. Choose **Fillet**in the **Model**command group in the Quad (2.2.3).
6. Specifythe desired fillet radius: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/She
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/ModifyinFlangesChan

Figure 3.510

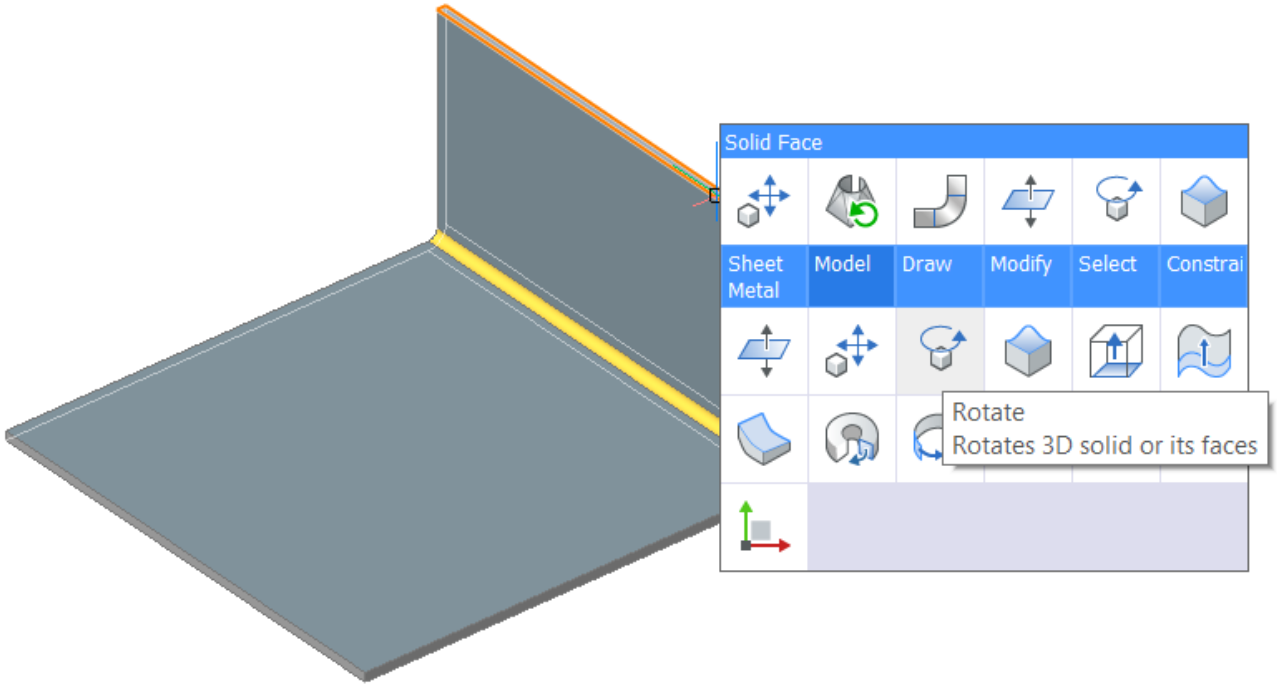

Solid Face
Sheet Metal
Model
Draw
Modify
Select
Constrai
Rotate
Rotates 3D solid or its faces

Figure 3.511

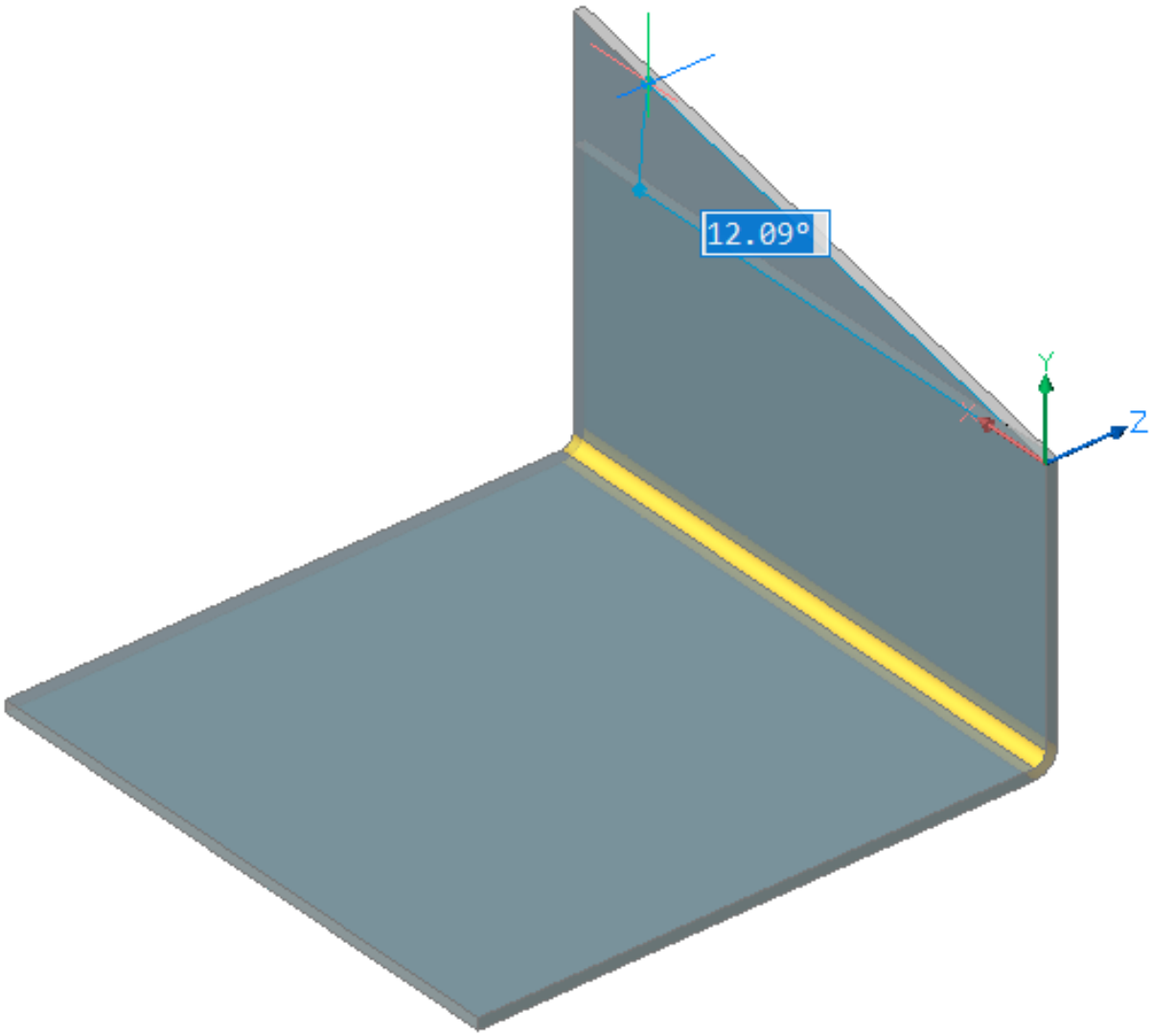


Figure 3.512

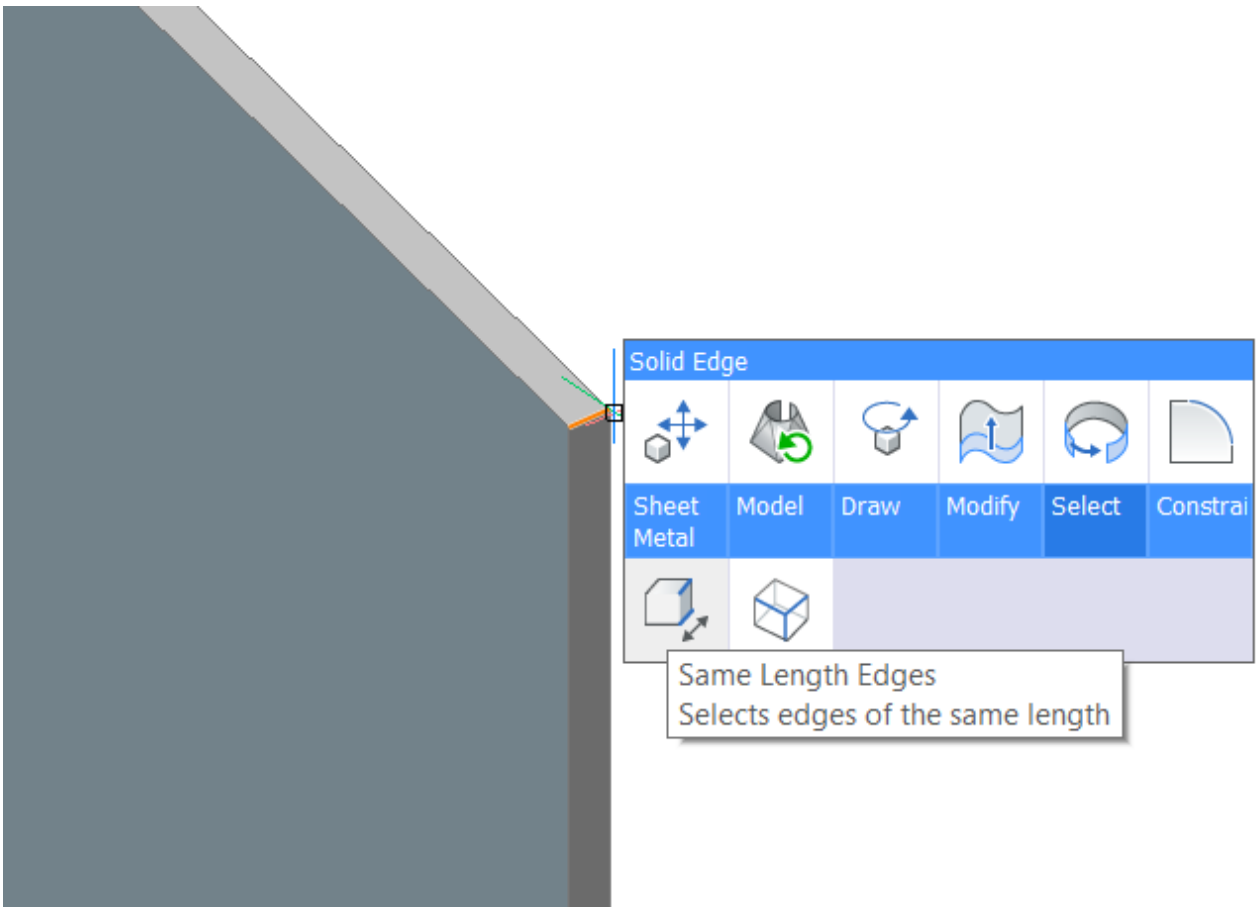

Solid Edge
Sheet Metal
Model
Draw
Modify
Select
Constrai
Same Length Edges
Selects edges of the same length

Figure 3.513

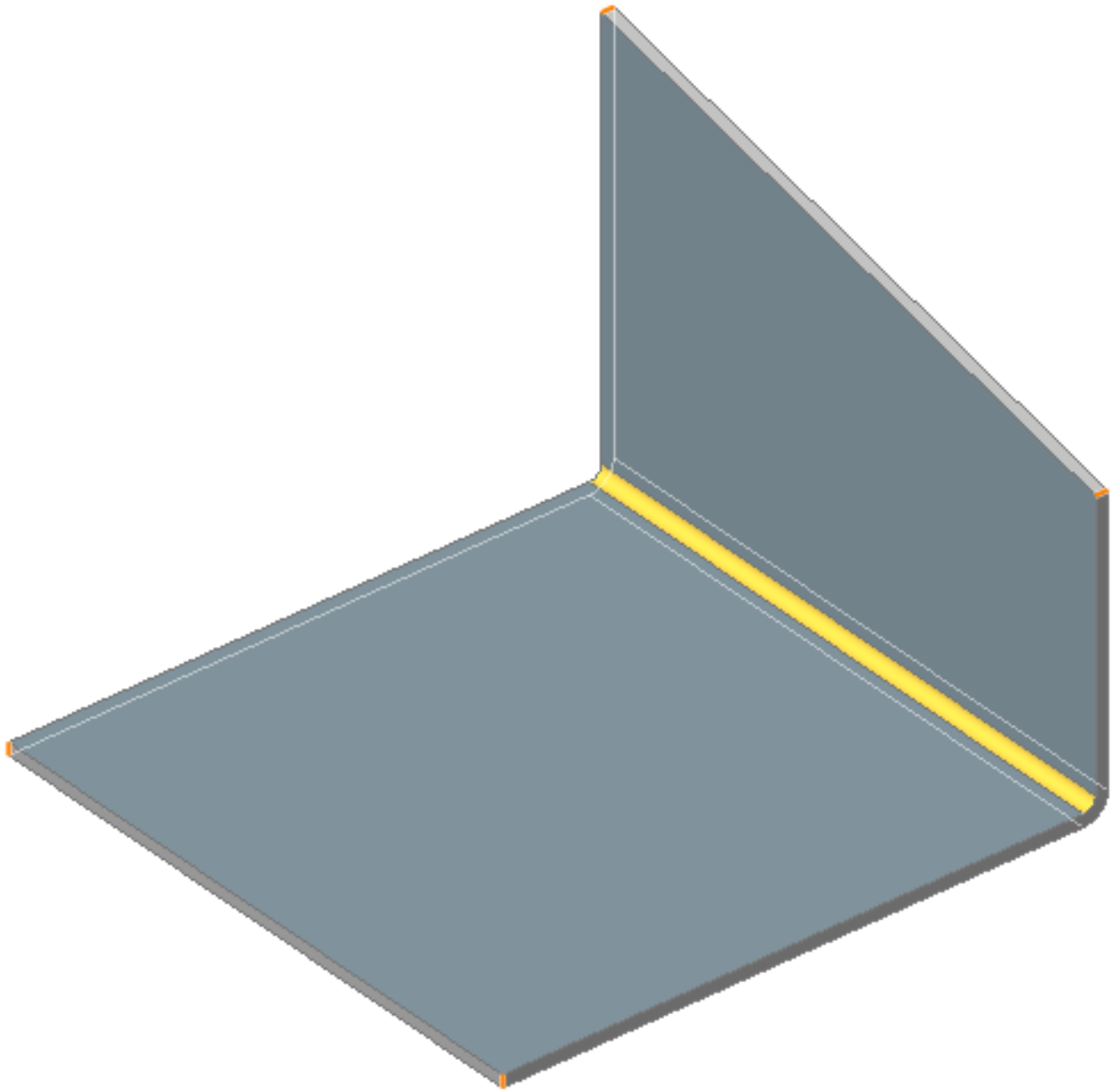


Figure 3.514

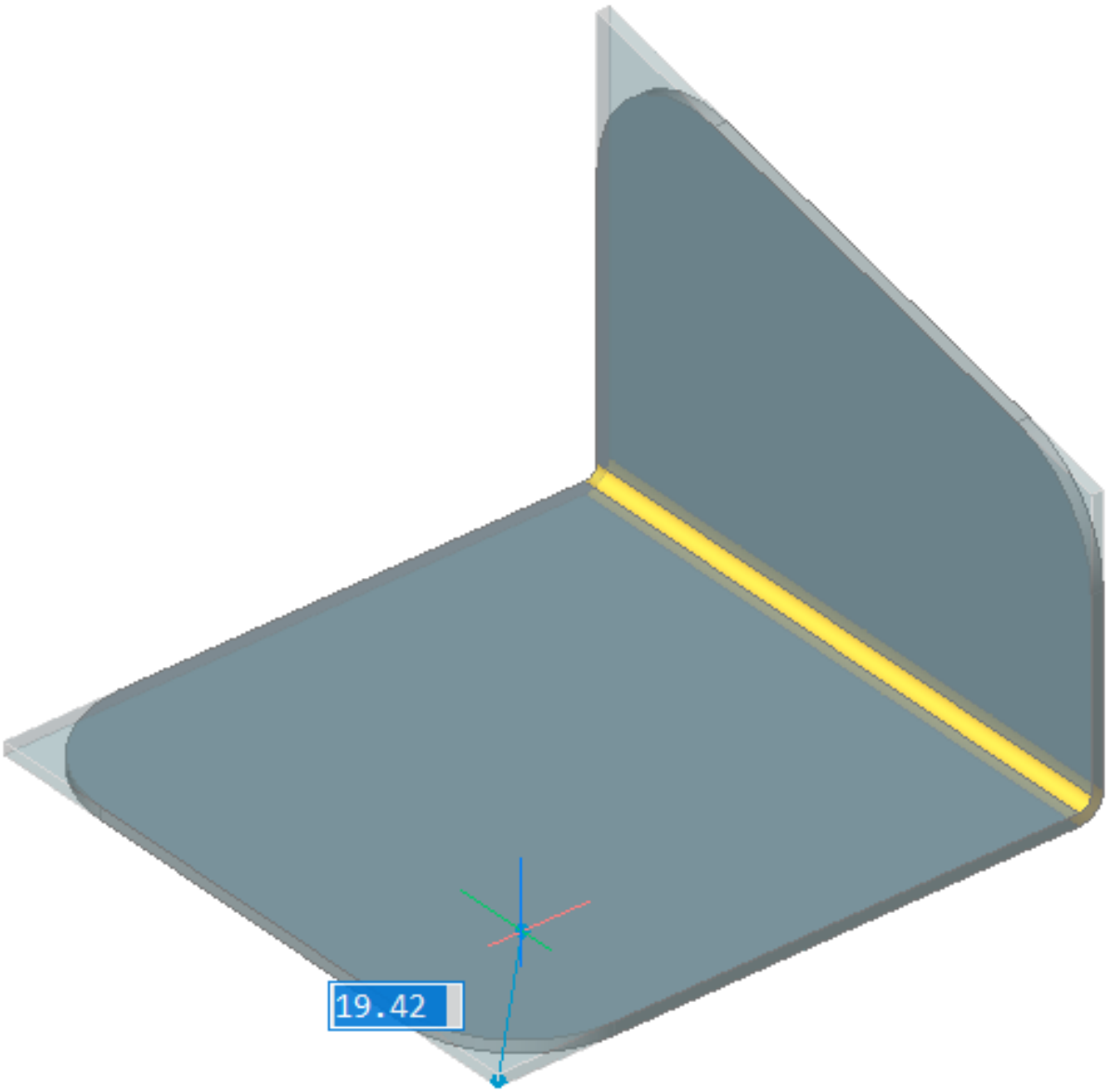
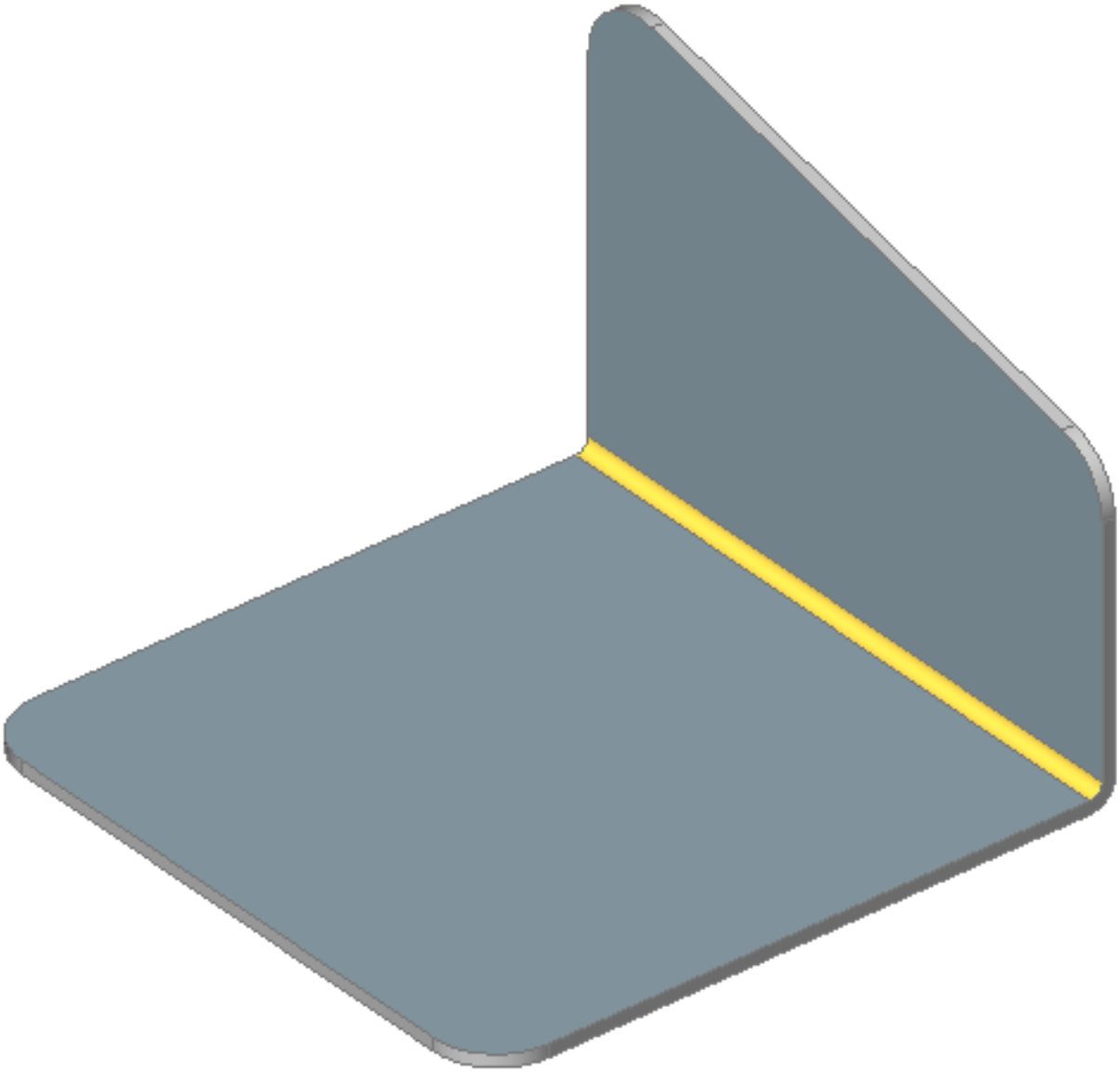


Figure 3.515



To make a hole in a flange

1. Create 4 circles as indicated in the image below. Use the Snap to Center entity snap to define the center points. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/ModifyinFlangesHole1
2. Select the circles.
3. Choose **Solid extrude**in the **Model**command group in the Quad (2.2.3). (\vref {fig:C:/Dev/Task-2448/images/staticbricsy
4. Move the cursor to create the holes. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/ModifyinFlangesHole4

See alsoWorking with Form and Rib Features (3.8.18)andWorking with Hole Features (3.8.19) articles to create holes.

Figure 3.516

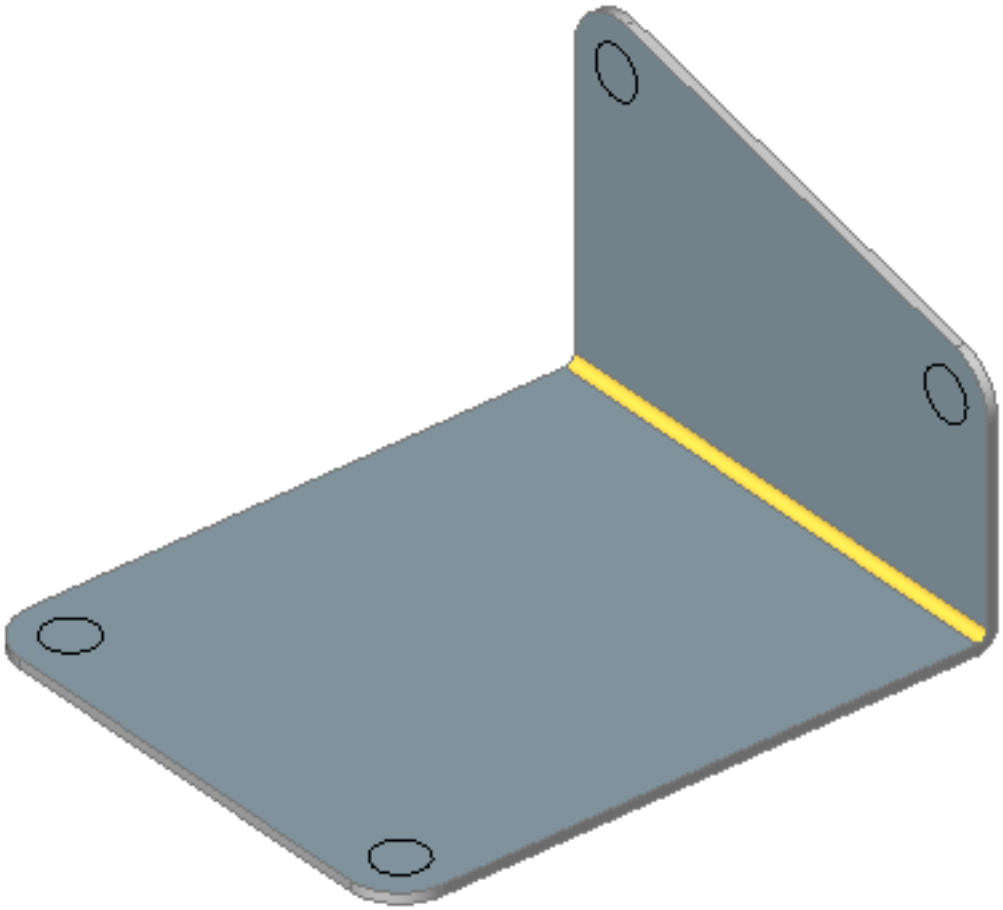


Figure 3.517

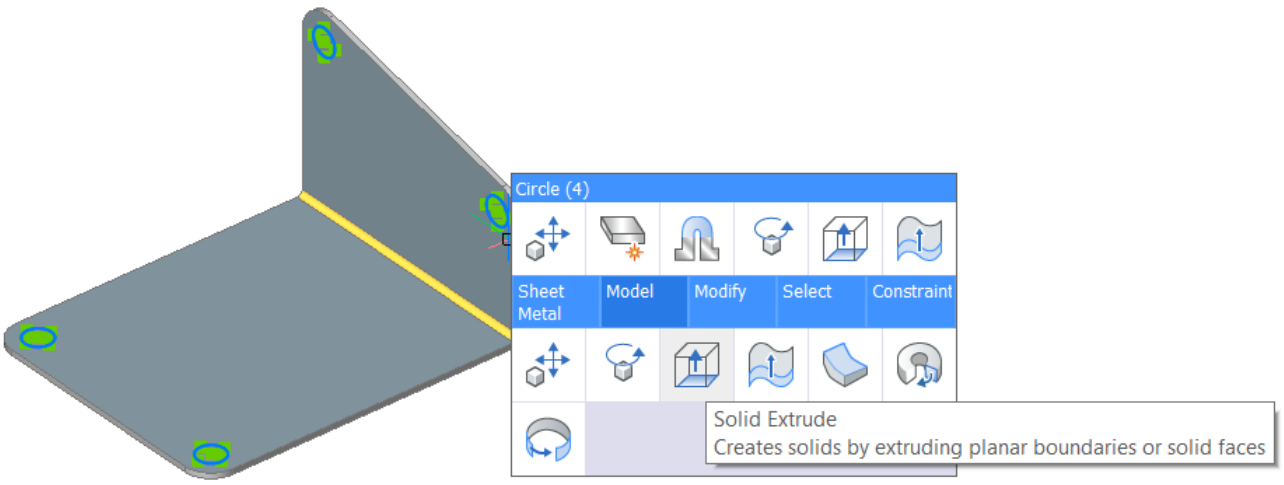


Figure 3.518

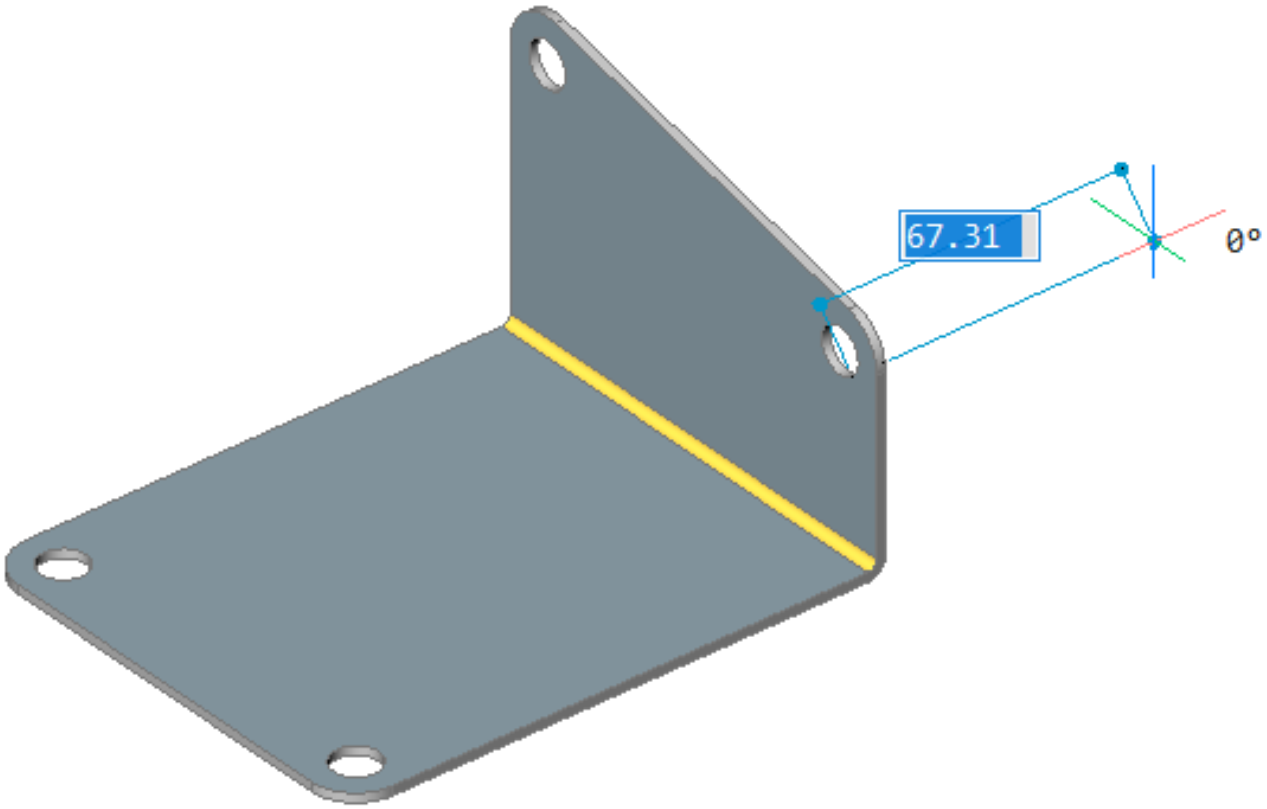
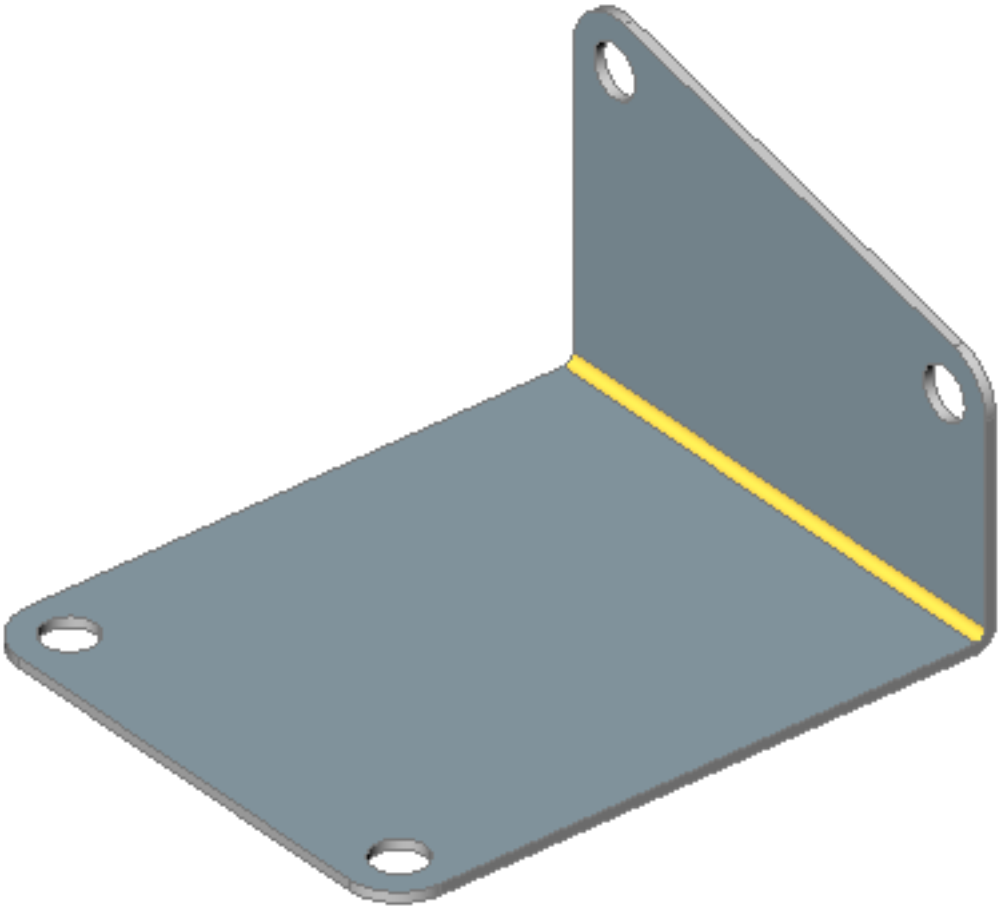


Figure 3.519



To move a flange

1. Do one of the following:

   - Click the **Push/Pull** tool button ( ) on the **Direct Modeling**toolbar.
   - Choose **Direct Modeling > Push/Pull** in the **Model**menu.
   - Choose **Push/Pull**( ) in the **Edit**panel of the **Sheet Metal** tab in the ribbon.
   - CallDmPushPull (6.12.20) command for given face.
   - Hover over the face you want to move and choose **Push/Pull**in the **Model**command group in the Quad (2.2.3).

   (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/ModifyinFlangesMove

2. Select new position of the flange. Note that due to presence of Sheet Metal Features (3.8.3), part thickness is preserved. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/ModifyinFl (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Mechanical/SheetMetal/ModifyinFlangesMove

Figure 3.520

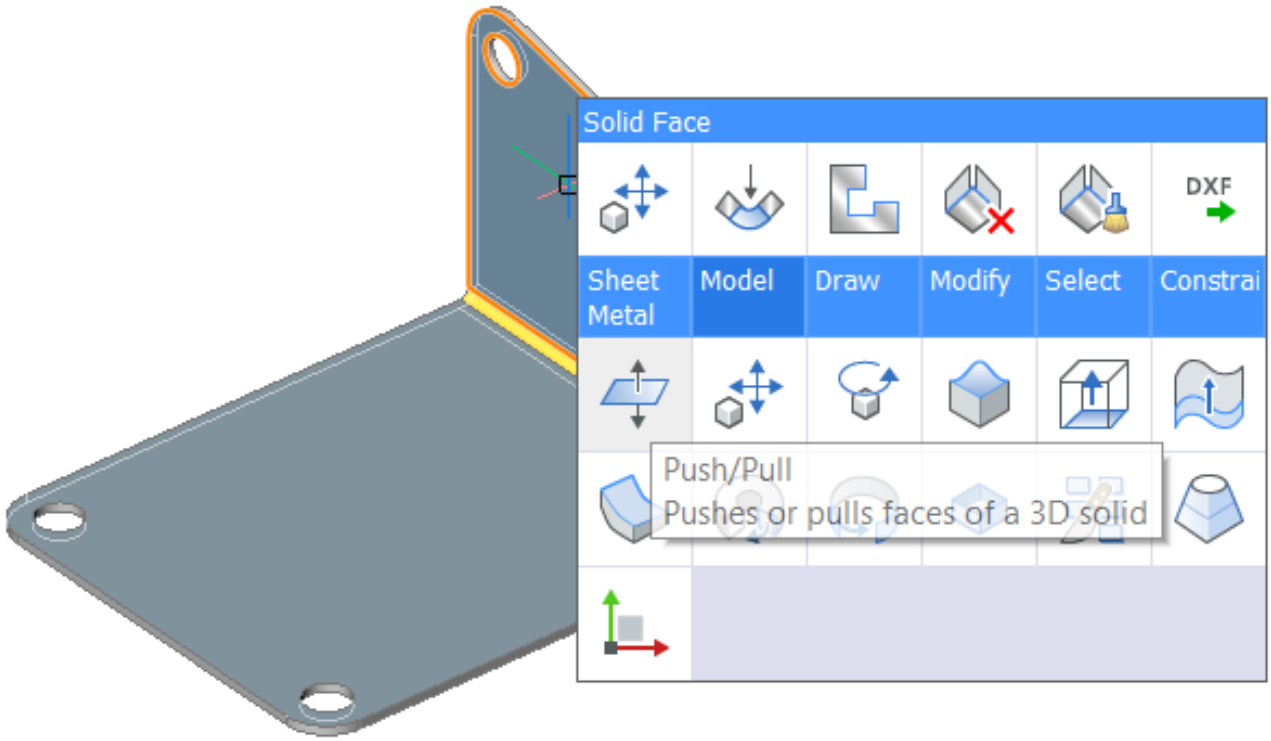


Figure 3.521

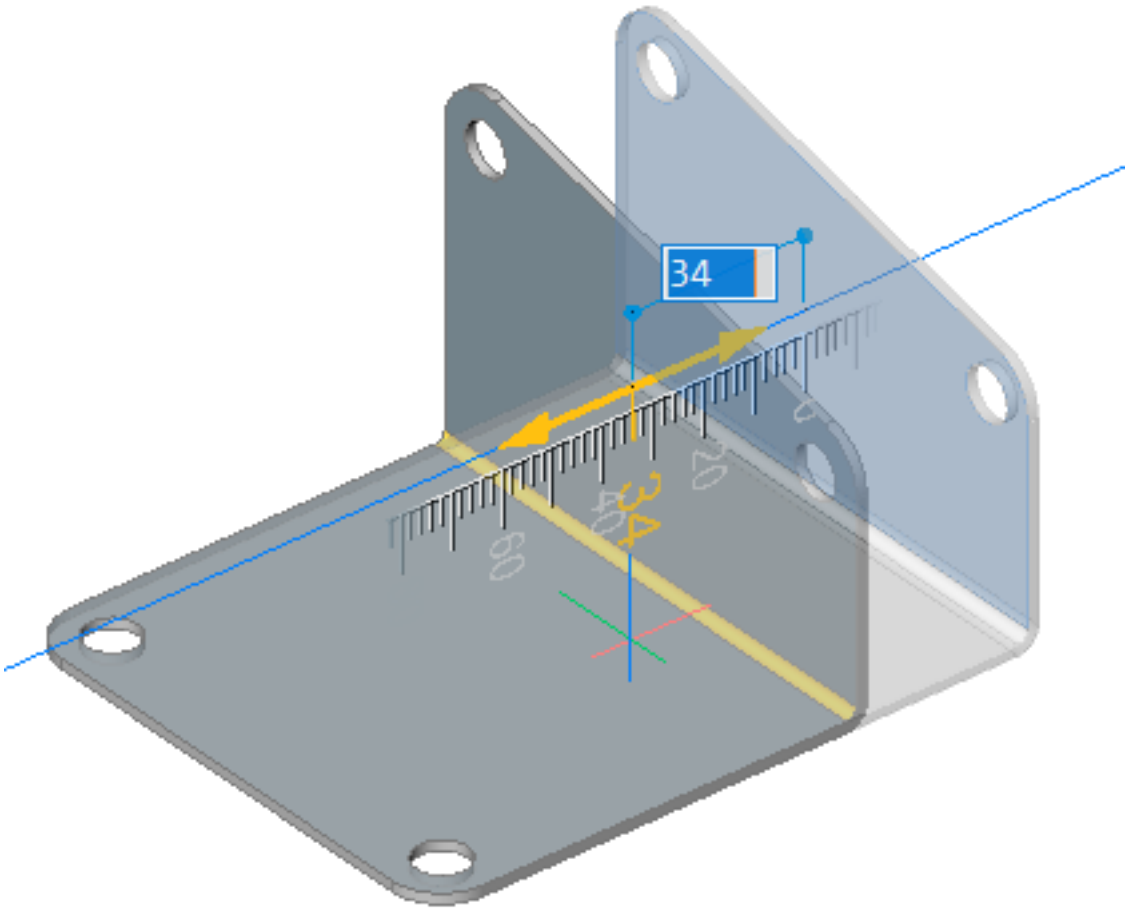
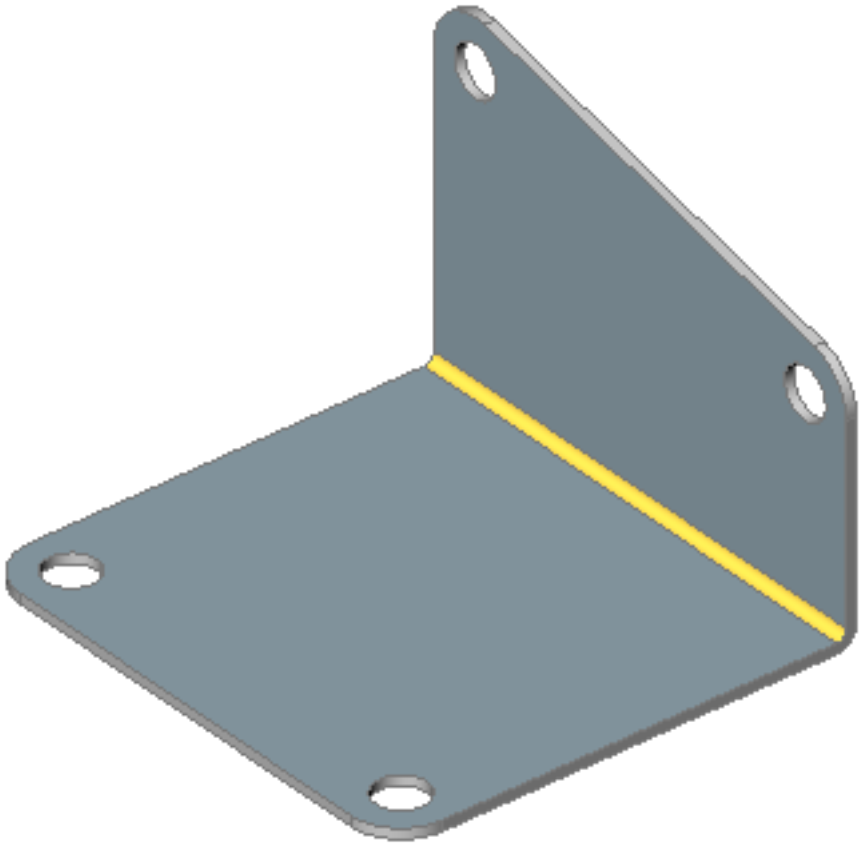


Figure 3.522



To rotate a flange

Although it is possible to rotate a flange using theDmRotate (6.12.24) command ( ), it is recommended to useSmFlangeRotate (6.28.14) ( ) instead, because this command automatically selects the rotation axis to respect the design intent of the sheet metal part.

1. Launch the SmFlangeRotate (6.28.14) ( ) command. **You are prompted: Select a flange face to rotate:** Flange faces under the cursor highlight.
2. Click to select a flange face. The flange rotates dynamically. A dynamic dimension displays the current angle with respect to the base flange.
3. Do one of the following:
   - Select a point.
   - Type a value in the dynamic dimension field.
   - Press the TAB key, then specify the absolute rotation angle in the dynamic dimension field.

NOTE	Wh theDmRotate (6.12.24) command is used to rotate a flange, which is connected to other flanges by junction features, these features are automatically dissolved before the rotation.

To split a flange

In some cases flanges are better split to minimize the material consumption, e.g. when a flange has a large hole, as shown in the image below: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10Drawing

1. Launch the SmSplit (6.28.28)( )command. **You are prompted:** Select flange or lofted bend face:
2. Select a flange face. **You are prompted:** Select lines or edges to split the flange or [use SMart split/draw a New line]<draw a New line>:
3. If no line exists yet on the flange face, right click. **You are prompted:** Start point of the line:

Figure 3.523

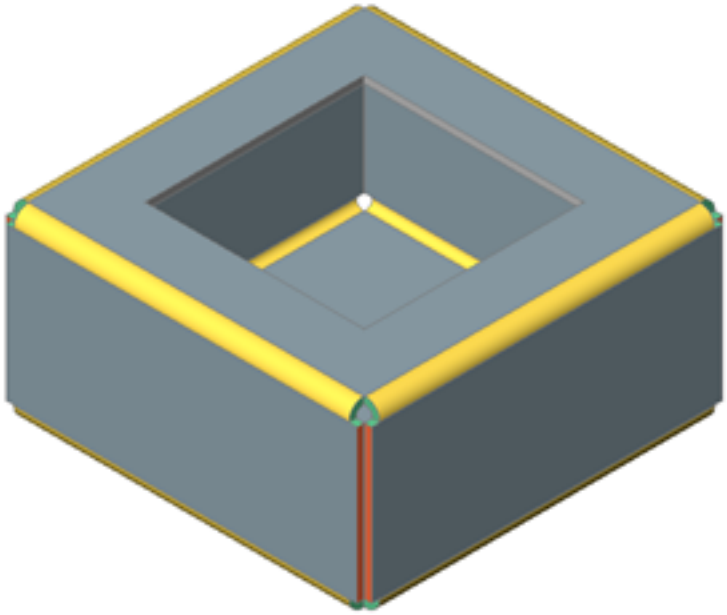
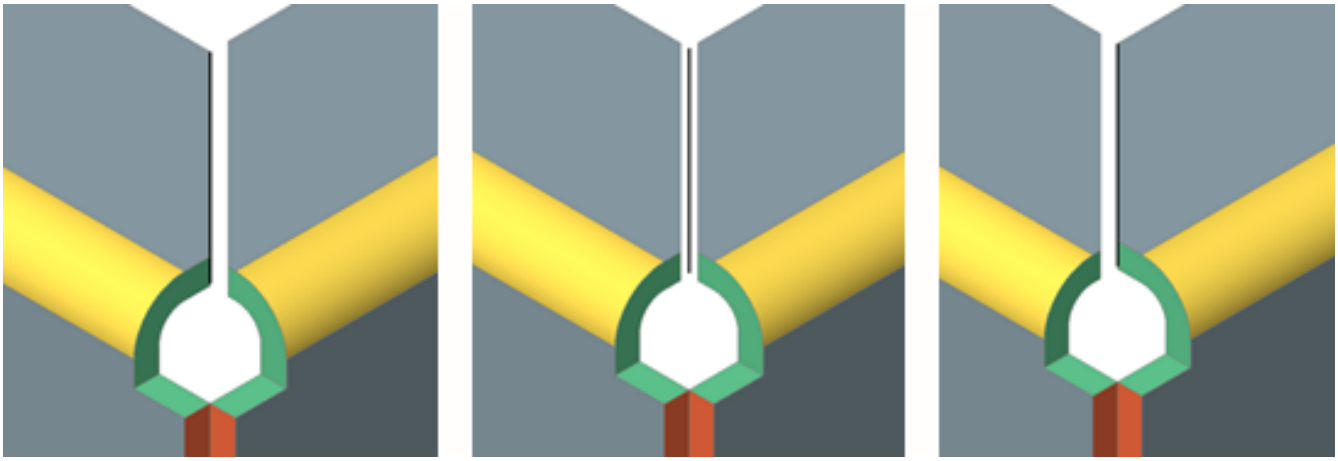


Figure 3.524



Smart tools to split a flange

The scenario below illustrates the other capabilities of the SmSplit (6.28.28)command:

- Smart split by selecting a single vertex in the corner.
- Split of the bend and convert the part adjacent to the corner into a junction. (\vref {fig:C:/Dev/Task-2448/images/staticbri

1. Do one of the following:

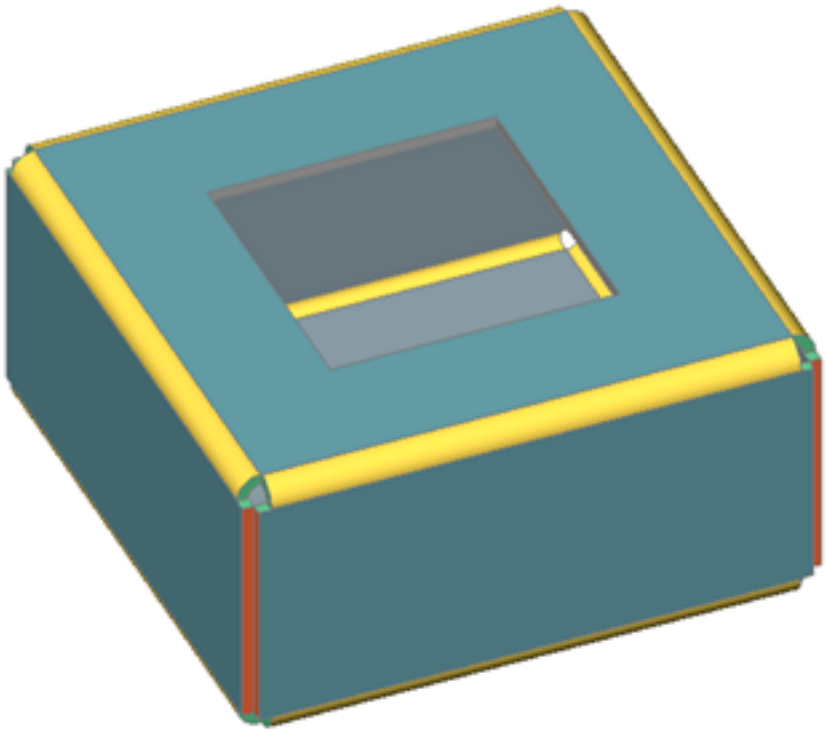
   - Click the **Use Smart Split Flange** tool () on the **Sheet Metal** ribbon tab;
   - Choose **Use Smart Split Flange** in the **Sheet Metal** menu;
   - Type **smsplit** in the command bar.

   You are prompted: Select flange or lofted bend face:

2. Select a flange face. **You are prompted:** Select lines or edges to split the flange or [use SMart split/draw a New line] <draw a New line>:

3. Select the **Use Smart split** option, then select a corner vertex: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel

Figure 3.525



You are prompted: Make split [(6.28.28)Center/Left/Right/Accept] <Accept>: The options are: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03DrawingSolids/ART/SMSmartSpl 1.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03Dr 2.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03Dr 3.png}) **Note:** The conversion of part of the bend into a junction is unrelated to the **Smart Split** mode.

4. To toggle the options, do one of the following:

   - Repeatedly hit the CTRL key. If TIPS (1.12.2) is ON the currently selected option is indicated in the Tips Widget.
   - Select **Center**, **Left** or **Right** in the Prompt Menu.
   - Type an option in the command bar, then press Enter.

   The flange is cut by a line, the bend is partially converted into the junction: (\vref {fig:C:/Dev/Task-2448/images/staticbric

In case of the**Center** option if Smart Features (3.8.16)are turned on, corner reliefs adjacent to the split will be converted into bend reliefs.

Figure 3.526

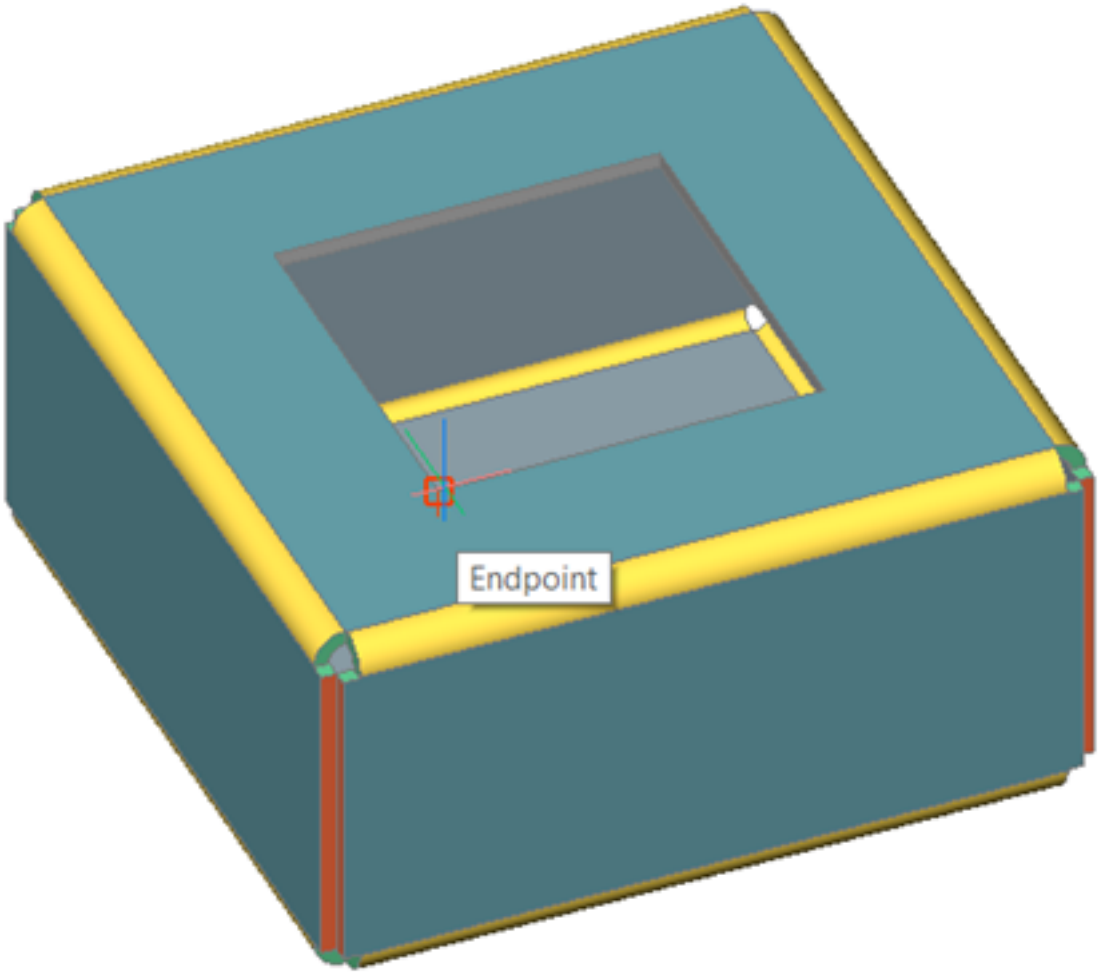


Figure 3.527

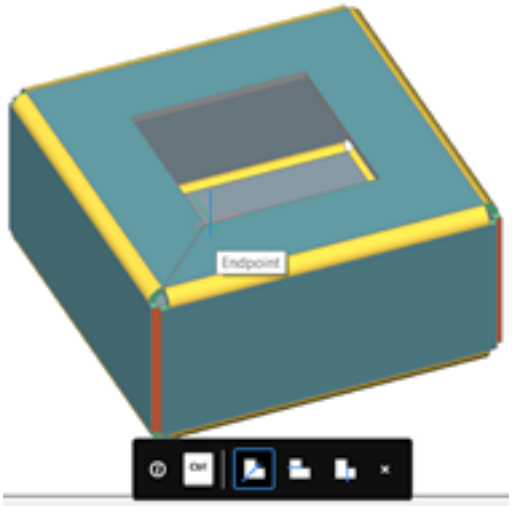


Figure 3.528

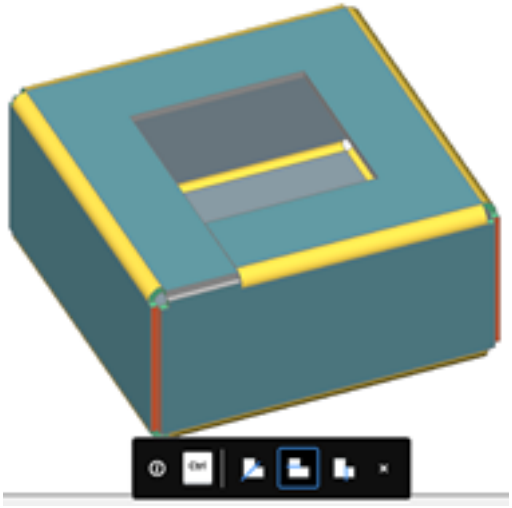


Figure 3.529

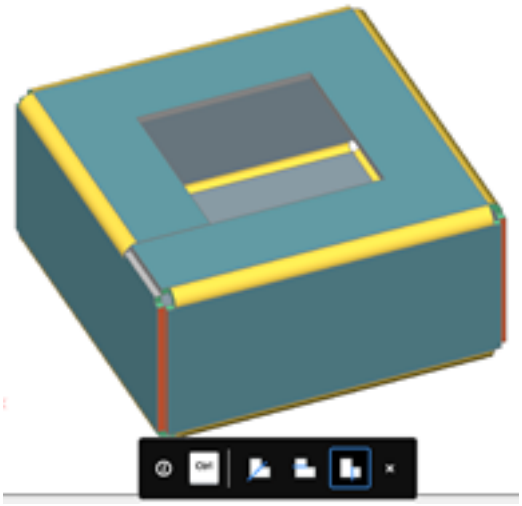
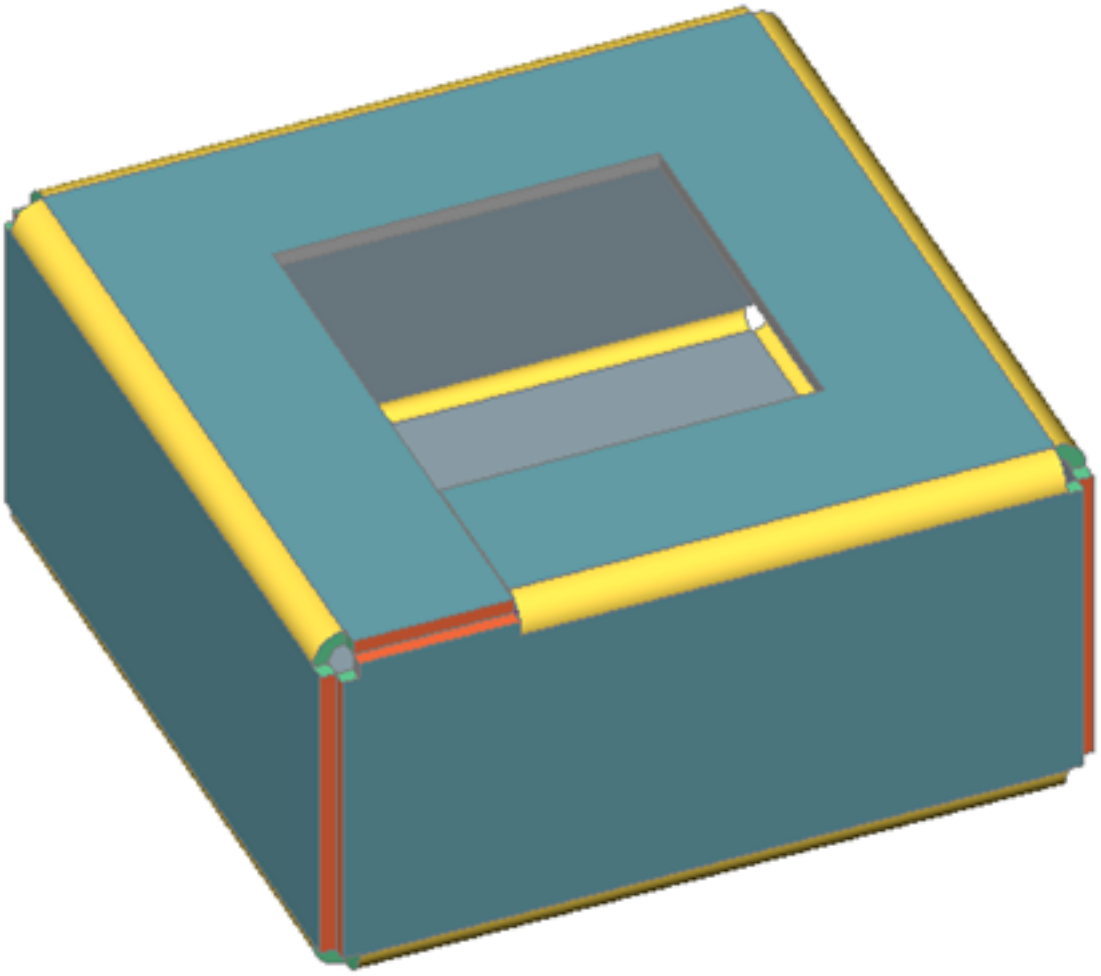


Figure 3.530



To bend a flange

The SmFlangeBend (6.28.11) command bends one or more flanges using a line. You can either draw the line first, then start the command or us the **New Line** option of the command.

To bend multiple flanges simultaneously

1. Launch theSmFlangeBend (6.28.11) command. **You are prompted:** Select a flange face:
2. Select the flange you want to bend. **You are prompted:** Select line or edge to bend the solid or draw a [New line] <New line>:
3. Do one of the following:
   - Select a line that overlaps the flange(s) to be bent.
   - Choose the **New Line** option. **You are prompted**to draw a lineon top of the flange(s) to be bent.
4. **You are prompted:** Pick a point on the side that will be moved: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he Pick a point at the side of the line you want to move. The selected part is bent dynamically. **You are prompted:**Enter bend angle or set [Radius/Switch side]:
5. (option)Type a value to define the bend angle, using the default bend radius.
6. (option) Select the **Radius** option.
   - **You are prompted:** Enter bend radius [Back] <Back>: Type a value to override the default bend radius and press Enter.
   - **You are prompted:** Enter bend angle or set [Radius/Switch side]: Type a value to define the bend angle. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/UsrGui/source/10DrawingEntities/03Draw

Figure 3.531

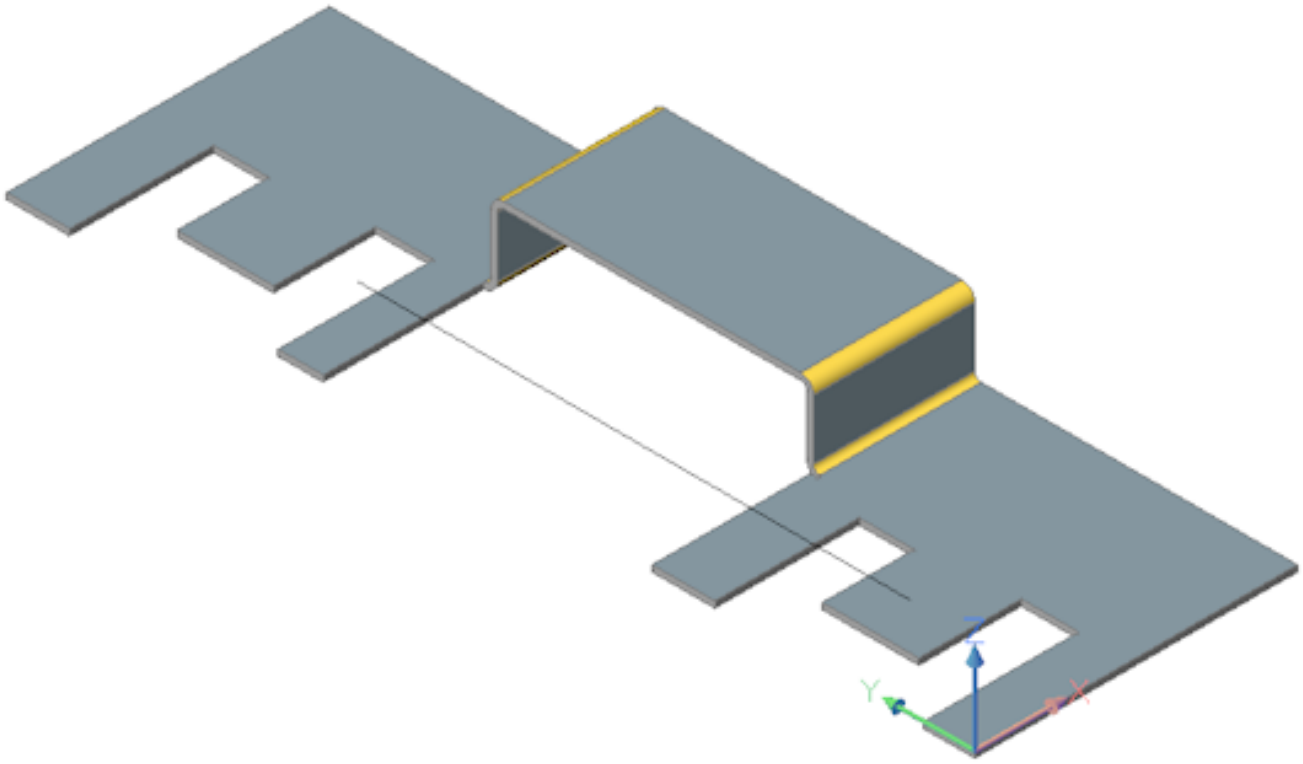
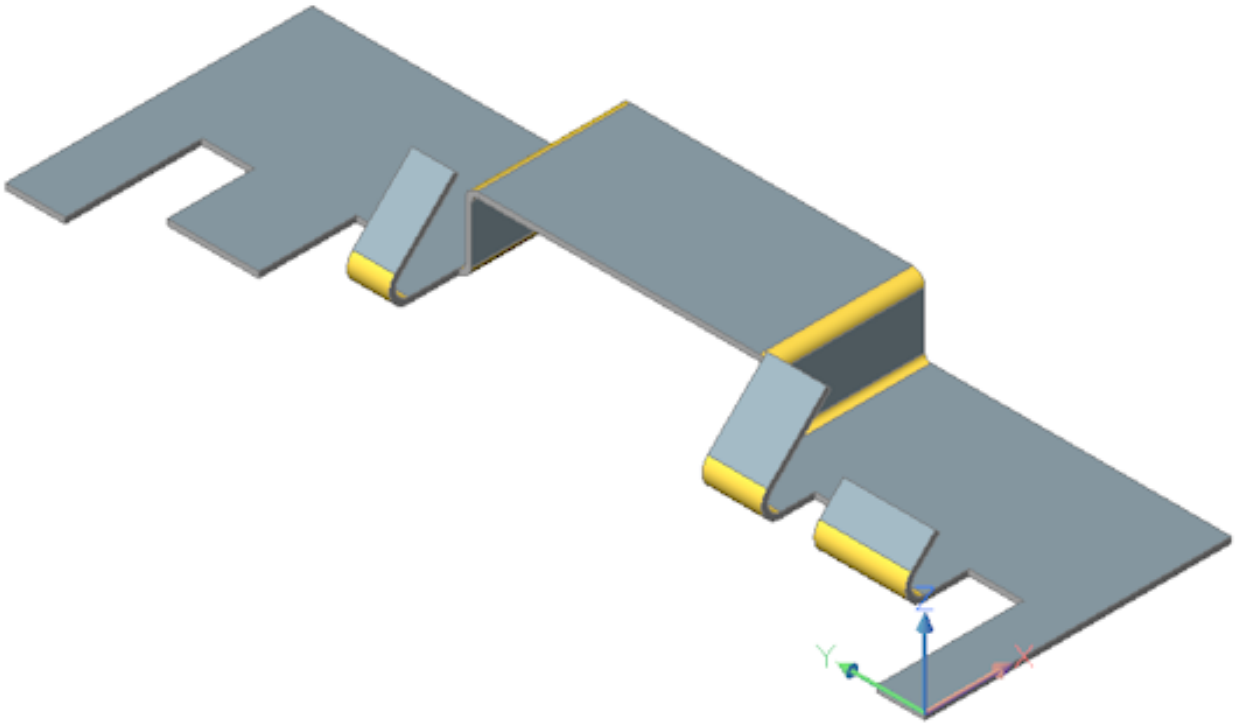


Figure 3.532



To connect flanges

The SmFlangeConnect (6.28.12) command closes gaps between two arbitrarily oriented flanges.

1. Launch theSmFlangeConnect (6.28.12) command. **You are prompted:** Select planar thickness faces of two flanges: Flange faces under the cursor highlight.
2. Select the thickness face of the first flange. **You are prompted:** Entities/subentities in set: 1 Select planar thickness faces of two flanges:
3. Select the thickness face of the second flange. The selected thickness faces are connected.

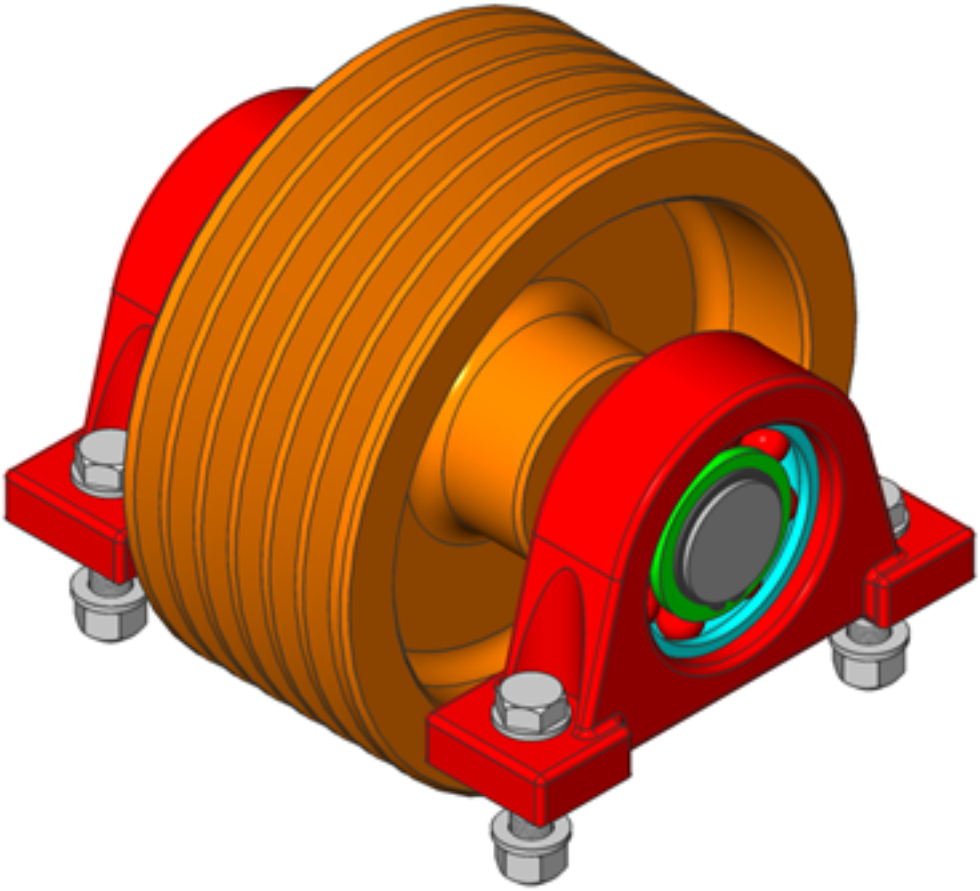
Note: After connection two flanges, you can convert the junction feature to a bend.

3.9 Generated Drawings - Workflows

3.9.1 Drawing generation

1. You need a**BricsCAD Platinum**license for this tutorial.
2. Open the **Small_Deflector_Sheave_assembly.dwg** file from Generative_drafting_files.zip. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040700934/mceclip0.png})

Figure 3.533



3.9.2 1. Creating Base Views

1. Click the **Base Views (6.31.3)** ( ) tool button on the **Annotate** | **Drawing Views** panel on the **Ribbon (1.13.6)**.
2. Hit **Enter** to accept <**Entire model**>.
3. Hit **Enter** to accept <**Layout1**>.
4. Select the **Orientation** | **Left** option on the **ViewBase** Prompt menu.
5. Select the **Scale** | **Standard scales**| **1:4** option on the **ViewBase** Prompt menu.
6. Set the position for the left view. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/3600407
7. Set the position for the top view. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/3600415
8. Hit **Enter** to accept **<Done>**.
9. Click the **Center Marks (6.9.6)** ( ) tool button on the **Annotate** | **Dimensions** panel on the **Ribbon (1.13.6)**to add the center mark to the circle in the **left view**.
10. Add a few radius dimensions (6.11.17) ( ) and a linear dimension (6.11.14) ( ), between bolts, to the left view using the dimensions tools from the **Annotate** | **Dimensions** panel on the **Ribbon (1.13.6)**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041531313/mceclip3.png})

Figure 3.534

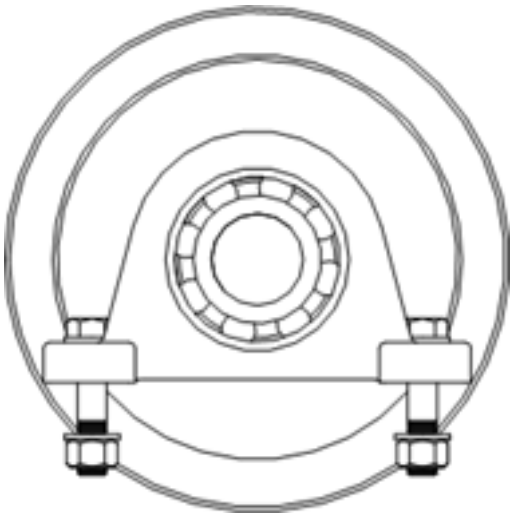


Figure 3.535

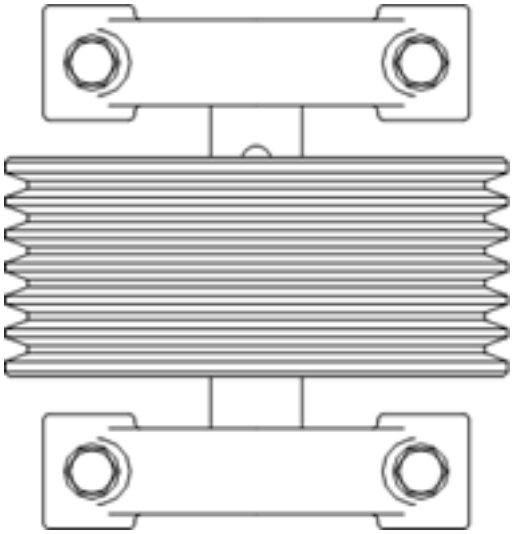
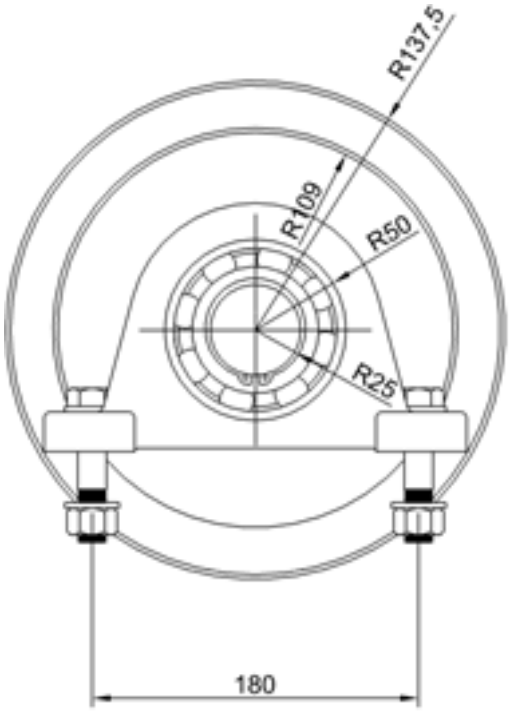


Figure 3.536



3.9.3 2. Creating Section Views

1. Click the **Section View (6.31.10)** ( ) tool button on the **Annotate** | **Drawing Views** panel on the **Ribbon (1.13.6)**to create vertical **Section View A-A**.
2. Select the left drawing view.
3. Specify the start point of the vertical section line.
4. Specify the next point of the vertical section line.
5. Move the cursor right and place the section view.
6. Use the**Linear dimension (6.11.14)** ( )and **Continue (6.11.9)** ( ) tool buttons from the **Annotate** | **Drawing Views** panel on the **Ribbon (1.13.6)**to add a few linear dimension to **Section View A-A**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041531393/mceclip4.png})
7. Click the**Section View (6.31.10)** ( )tool button on the **Annotate** | **Drawing Views** panel on the **Ribbon (1.13.6)**to create horizontal **Section View B-B**.
8. Select the top drawing view.
9. Specify the start point of the horizontal section line.
10. Specify the next point of the horizontal section line.
11. Move the cursor to the top and tap the CTRL key to turn off alignment. This will allow you to place the section view to the right-hand side.
12. Set the position for the section view. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/3600
13. Repeat step 1.9 to add**Center Marks (6.9.6)** ( )to the circle of **Section View B-B**.

Figure 3.537

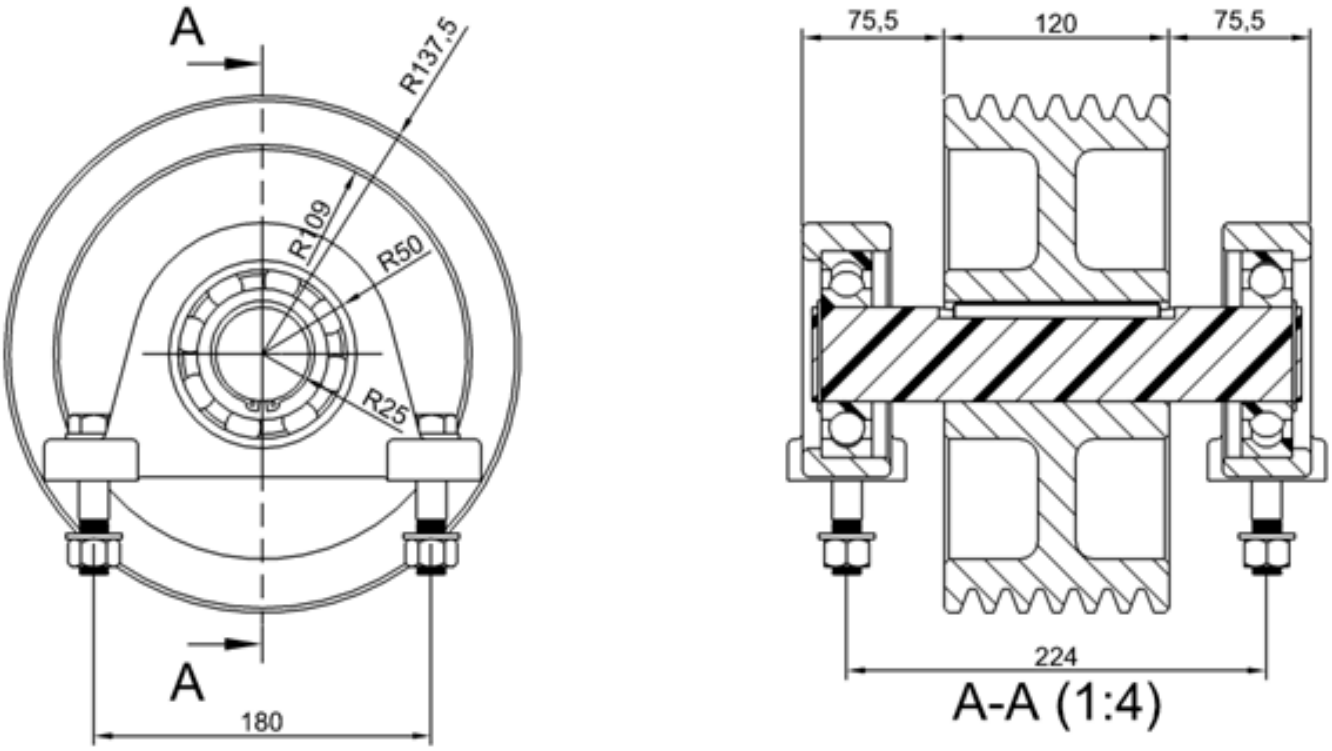
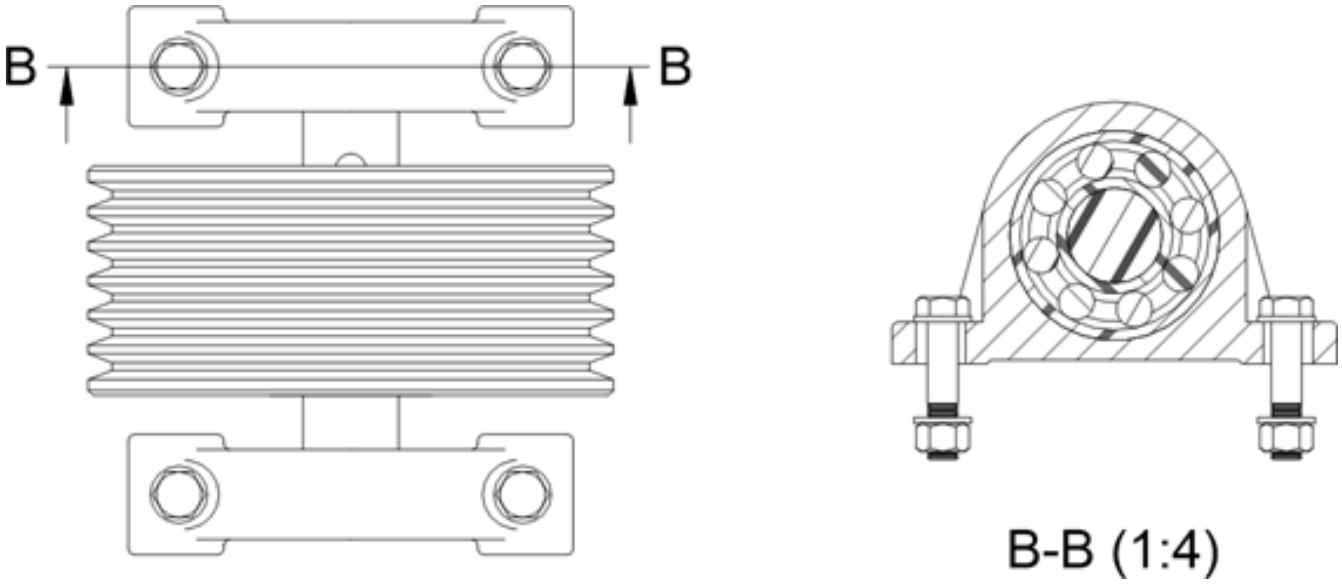


Figure 3.538



3.9.4 3. Non-sectioned components

1. Switch to the **Model Space**.
2. Select **Ball_Bearing_param** in the Mechanical Browser (3.2.2).
3. Click the **Open (6.23.8)**( ) tool button on the right-click menu. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/h
4. Select the **Ball_Bearing_param** component on the Mechanical Browser (3.2.2).
5. On the bottom part of the Mechanical Browser (3.2.2), click the **Sectionable** field and change the parameter value to **No.**
6. Click the **Save (6.27.1)** ( ) tool button on the **Home** | **File** panel on the **Ribbon (1.13.6)**to save the**Ball_Bearing_param.**
file.
7. Switch to the **Ball_Bearing_param.dwg** file.
8. Click the **Update (6.8.25)** ( ) tool button on the **Assembly** | **Tool** panel on the **Ribbon (1.13.6)**.
9. Switch to **Layout1**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041531813/mcecl

Figure 3.539

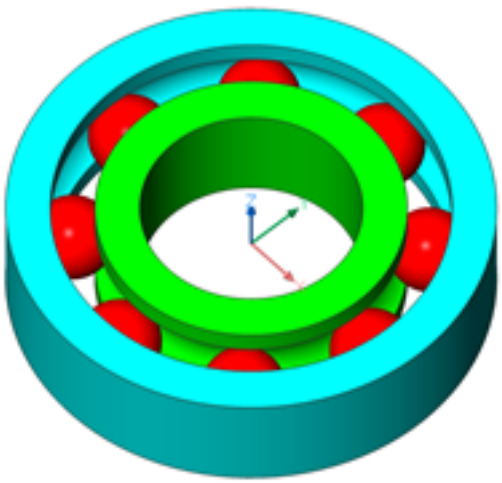
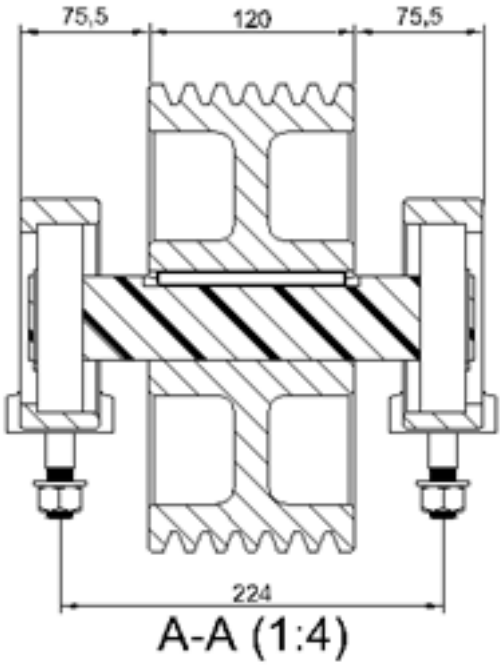


Figure 3.540

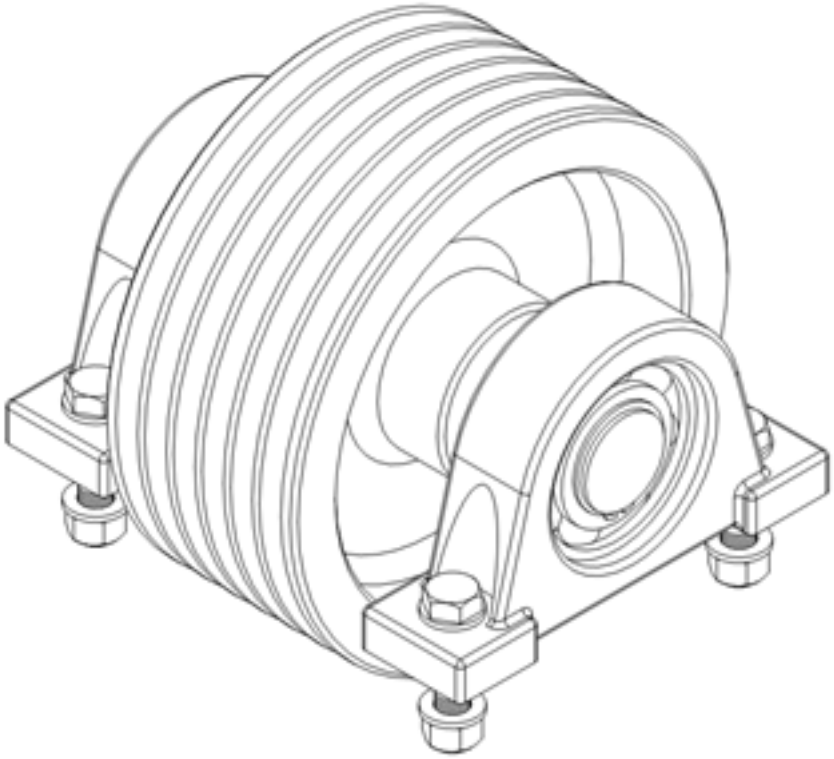


3.9.5 4. Creating Isometric views

1. Switch to the **Model Space**.
2. Click the**Base Views (6.31.3)** ( ) tool button on the **Annotate** | **Drawing Views** panel in the **Ribbon (1.13.6)**.
3. Hit **Enter** to accept **<Entire model>**.
4. Hit **Enter** to accept <**Layout1**>.
5. Select the **Orientation** | **Current** options in the **ViewBase** Prompt menu.
6. Select the**Tangent lines** | **Yes** options in the **ViewBase** Prompt menu.
7. Set the position for the isometric
8. Hit **Enter** to accept <**Done**>. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36004070205

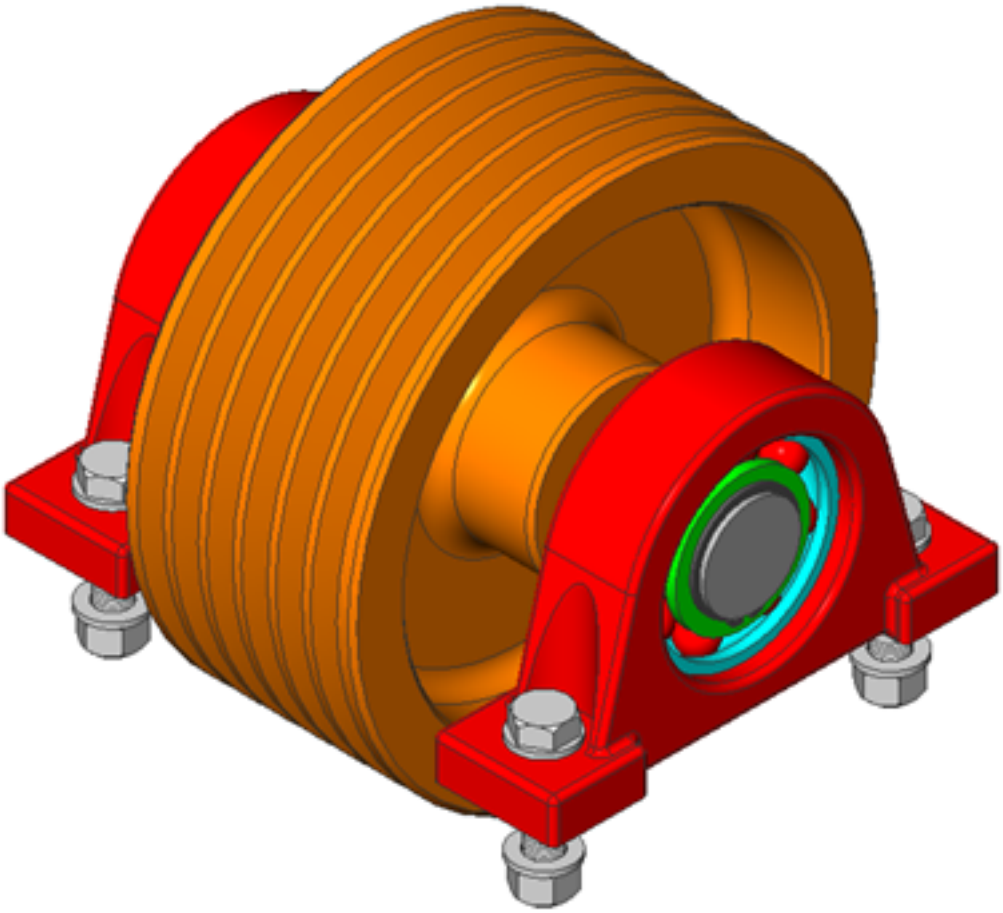
9. Select the **Isometric view.**
10. In the **Drawing view** section of the Properties panel (1.12.4) change **Geometry** to **3D** and in the **Misc** section change **Visual style** to **Mechanical**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36004

Figure 3.541



11. Switch off the **Drafting Viewports** layer in the drop-down **Layers** menu, in the **Home** | **Layers**panel on the **Ribbon (1.13.6)**.

Figure 3.542



3.9.6 5. Bill of materials

1. Click the **Bill of material (6.8.2)** ( ) tool button on the **Annotate** | **Assembly** panel on the **Ribbon (1.13.6)**.
2. Select the **Configure, Material, Mass, Parameters** options on the **BmBom** prompt menu.
3. Click **Back** on the **BmBom** prompt menu to return in the main menu.
4. Select the **Bottom level** option on the **BmBom** prompt menu.
5. Set the Insertion point. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360041531913/mc

Figure 3.543

Bill of Materials Small_Deflector_Sheave_assembly								
No.	Component	Quantity	Material	Mass	Type	Bore	Length	OutsideDiameter
1	Ball_Bearing_param	2	Steel	0.88 kg	6310	50.00		110.00
2	Bearing support	2	Iron	3.49 kg				
3	DIN 471xDSH-50	2	Steel	11.30 g				
4	ISO 4017xM16 X [illegible].5x75	4	Steel	0.15 kg				
5	ISO 4032xM16 X 1.5x0	4	Steel	38.46 g				
6	ISO 7089xM17x0	8	Steel	11.52 g				
7	Pin_16x10x110	1	Steel	0.11 kg				
8	Shaft	1	Steel	4.00 kg			261.00	
9	Small Deflector Sheave	1	Iron	23.38 kg				

3.9.7 6. Balloons placement

1. Click the **Balloon (6.8.1)** () tool button on the **Annotate** | **Assembly** panel on the **Ribbon (1.13.6)**.
2. Select a component insert and set a point to place a balloon.
3. Place the balloons for all components on the **left view** and **Section View A-A**. (\vref {fig:C:/Dev/Task-2448/images/helpbr

4. Select balloon **#8**.
5. On the **Misc** section of the Properties panel (1.12.4) change **Mleader style** to**Dot**. (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040702494/mceclip8.png})

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360040702574/mceclip9.png})

Figure 3.544

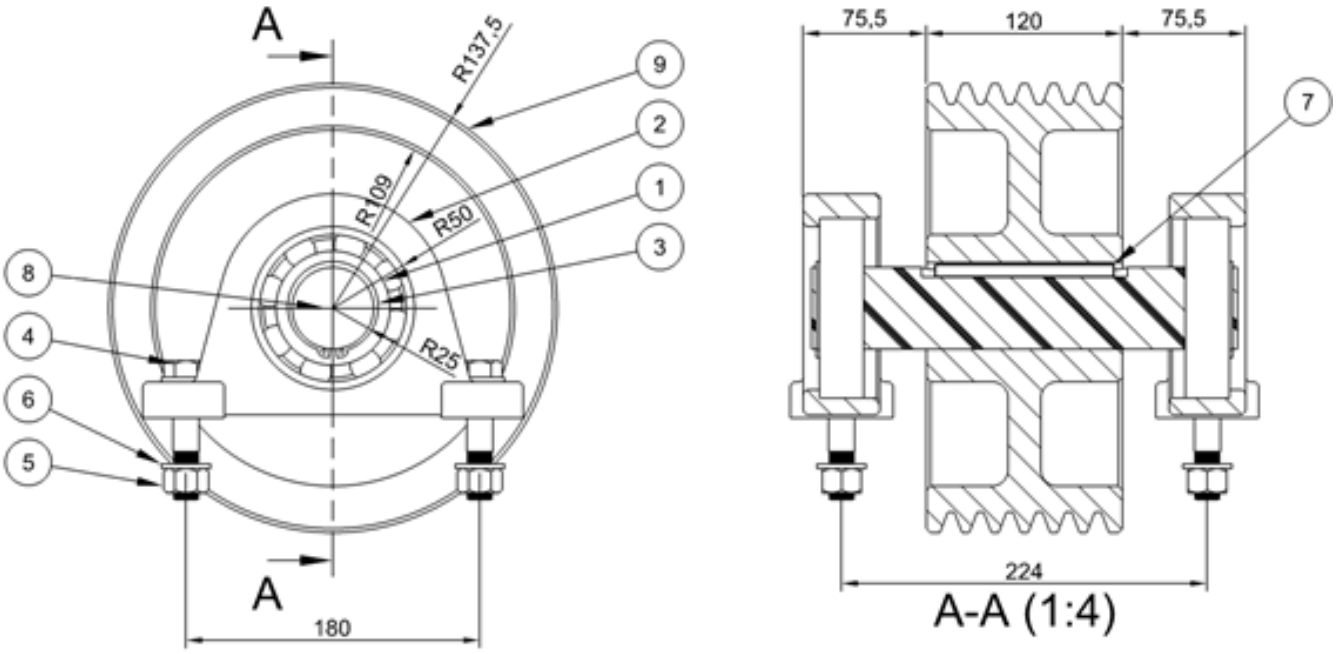
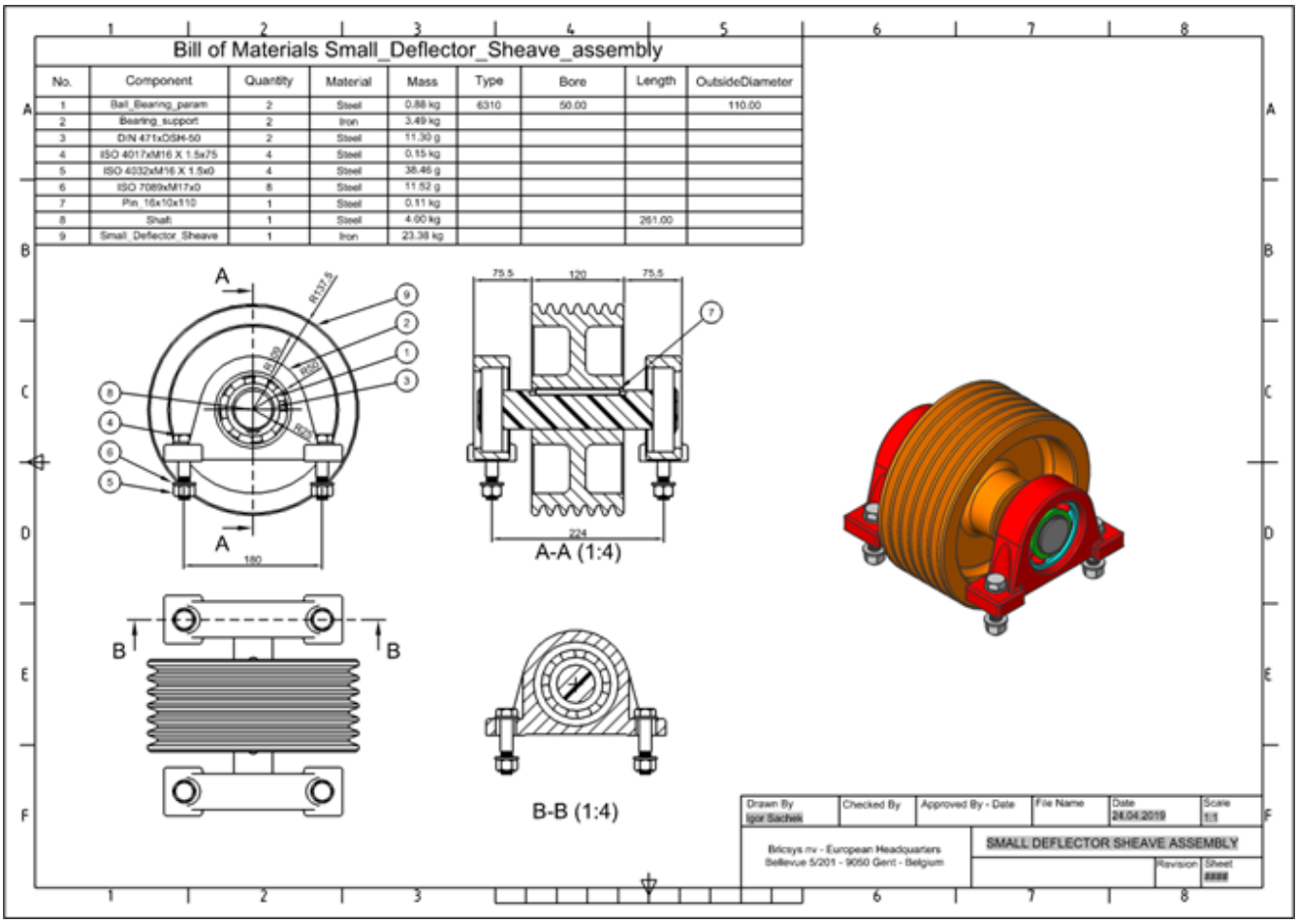


Figure 3.545



Figure 3.546

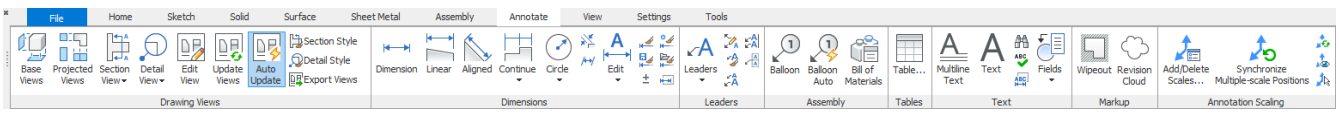

Bill of Materials Small_Deflector_Sheave_assembly
No. Component Quantity Material Mass Type Bore Length OutsideDiameter
1 Ball_Bearing_param 2 Steel 0.88 kg 6310 50.00 110.00
2 Bearing_support 2 Iron 3.49 kg
3 DIN 471xDSH-50 2 Steel 11.30 g
4 ISO 4017xM16 X 1.5x75 4 Steel 0.15 kg
5 ISO 4032xM16 X 1.5x0 4 Steel 38.46 g
6 ISO 7089xM17x0 8 Steel 11.52 g
7 Pin_16x10x110 1 Steel 0.11 kg
8 Shaft 1 Steel 4.00 kg 261.00
9 Small_Deflector_Sheave 1 Iron 23.38 kg
A
A
180
75,5
120
75,5
224
A-A (1:4)
B
B
B-B (1:4)
Drawn By
Igor Sachek
Checked By
Approved By - Date
File Name
Date
24.04.2019
Scale
1:1
Bricsys nv - European Headquarters
Bellevue 5/201 - 9050 Gent - Belgium
SMALL DEFLECTOR SHEAVE ASSEMBLY
Revision
Sheet

3.10 Generated Drawings

3.10.1 Drawing Views

BricsCAD Mechanical provides a rich toolset to generate drawing views from 3D models of parts and assemblies. Drawings views are created in the paper space of the same .dwg file and the 3D model is created in model space. Drawing views can be annotated using dimensions, leaders, text and tables. The views and annotations are associatively updated upon change of the underlying 3D model. The tools to generate, update and annotate drawing views are presented on the **Annotate** ribbon tab of **Mechanical** workspace: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360017834654/mceclip0.png}) Base Views (6.31.3) generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout. Projected Views (6.31.8) generates additional projected views from an existing drawing view. Section View (6.31.10) creates a cross section view based on a standard drawing view generated by the ViewBase command in a paper space layout. Detail View (6.31.4) creates a detail view of a portion of a standard generated drawing at a larger scale. Edit View (6.31.6) allows changing the scale and the hidden line visibility of drawing views. Update Views (6.31.12) updates a selection of drawing views obtained by VIEWBASE and VIEWSECTION when VIEWUPDATEAUTO = 0. There are predefined and user-defined styles for Section (6.31.11) and Detail (6.31.5) views. Export Views (6.31.7) exports the content of drawing views obtained by ViewBase and ViewSection to the Model Space of the drawing or to a new drawing.

Figure 3.547



3.10.2 Drawing Annotations

Creating and editing annotations, review Dimensioning Tools Overview (1.21.2). Dimensions and Multileader (1.21.14) that are in the drawing view are updated upon change of the underlying 3D model. Assembly drawings (3.6.7) can be annotated with BOM balloons. Every balloon refers to a row in a BOM (3.6.6) table generated from the assembly. The BOM table can be placed in paper space (layouts (1.16.15)) with the drawing view of an assembly. Exploded Representations (3.6.7) of assemblies can be used to generate drawing views and annotate them with balloons.

Chapter 4

Communicator for BricsCAD

4.1 Getting Started

4.1.1 BricsCAD Communicator

Commands: COMMUNICATORINFO (6.9.39), EXPORT (6.13.15), IMPORT (6.17.10) BricsCAD Communicator is an optional plug-in for the Pro and Platinum editions of BricsCAD that imports and exports 3D data between major CAD formats. Its high quality translations facilitate the re-use of 3D CAD data, thereby improving productivity and reducing project costs.

Tutorial: Assembly design using BricsCAD Communicator Click here (3.3.9) to watch.

Supported file formats

Export

- STE, STP, STEP: Standard for Exchange of Product data. The following export protocols can be specified: AP203, AP214, AP242. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Communicator/GettingS
- IGS, IGES: Initial Graphics Exchange Specification
- CATIA V4 and V5:
- VDA: VDA-FS
- 3D PDF

You can export BricsCAD assembly with mechanical components as a STEP file with product structure. To do that set the desired value in **Settings** dialog: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Communicato

Figure 4.1

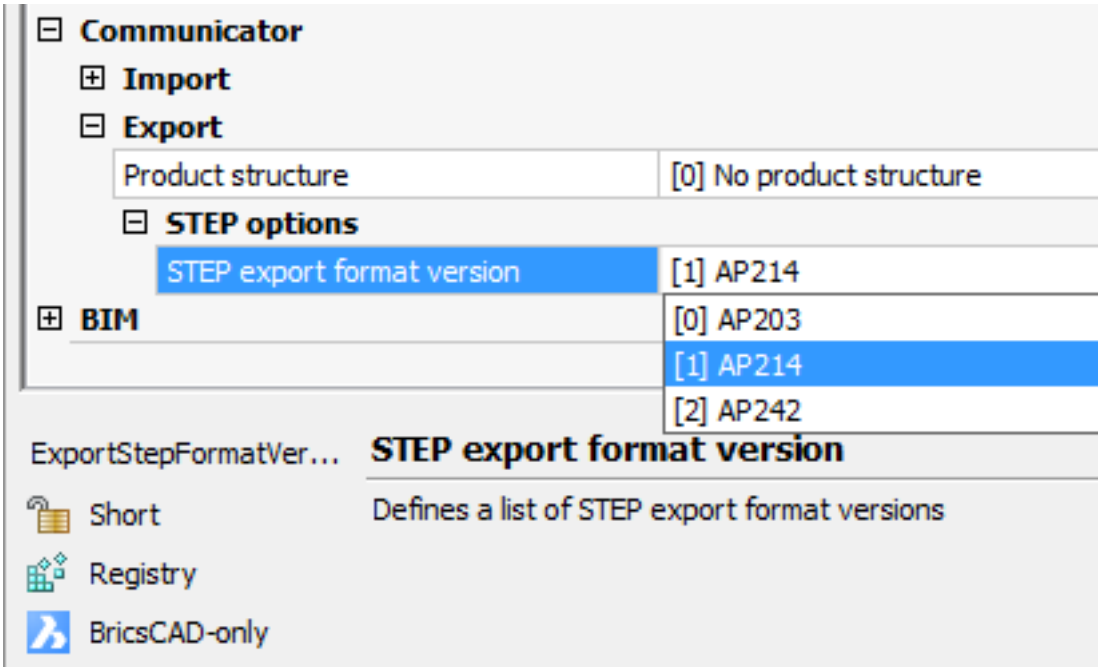


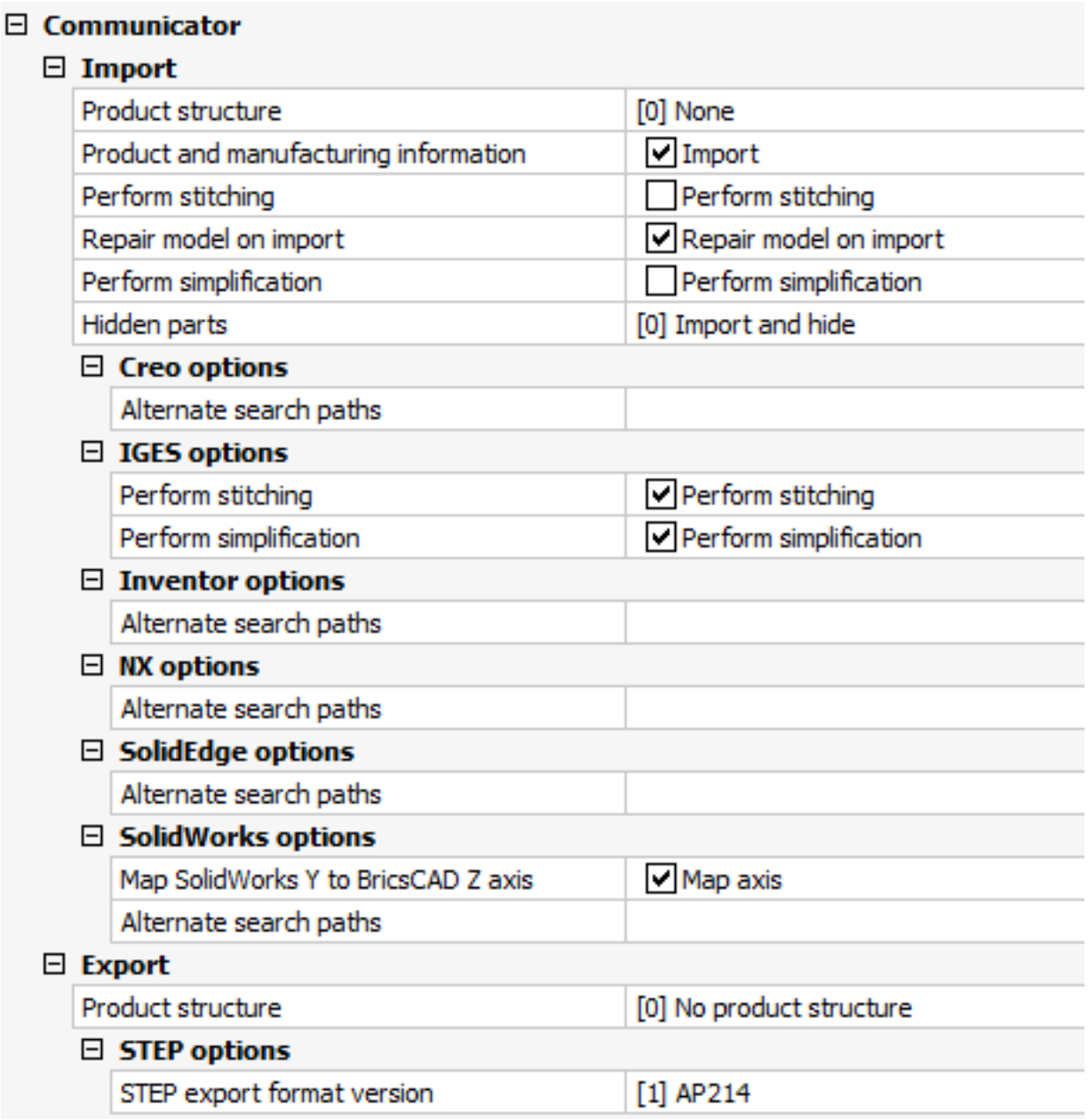
Figure 4.2



Import

- ACIS
- ASM, PRT: Creo Elements / Pro Engineer
- CATIA V4/V5/V6
- IAM, IPT: Inventor
- IGS, IGES: Initial Graphics Exchange Specification
- NX
- PARASOLID
- PAR, PSM: Solid Edge
- Pro/E/Creo
- PRT: NXSTE, STP, STEP: Standard for Exchange of Product data
- SLDASM, SLDPRT: Solidworks
- VDA: VDA-FS
- XCGM

Figure 4.3


Communicator
Import
Product structure
[0] None
Product and manufacturing information
Import
Perform stitching
Perform stitching
Repair model on import
Repair model on import
Perform simplification
Perform simplification
Hidden parts
[0] Import and hide
Creo options
Alternate search paths
IGES options
Perform stitching
Perform stitching
Perform simplification
Perform simplification
Inventor options
Alternate search paths
NX options
Alternate search paths
SolidEdge options
Alternate search paths
SolidWorks options
Map SolidWorks Y to BricsCAD Z axis
Map axis
Alternate search paths
Export
Product structure
[0] No product structure
STEP options
STEP export format version
[1] AP214

When you import an assembly file, you can get its content as a plain geometry (a nonhierarchical set of entities) or map its product structure into native blocks or mechanical components. PMI (product and manufacturing information) is supported for the following file formats: ACIS, CATIA V5, NX, Pro/E / Creo, SolidWorks, XCGM (Communicator license is required). PMI annotations are imported as graphical elements grouped in blocks. If the assembly contains hidden parts, you can decide how to represent them in BricsCAD: import and hide, import and set visible, or just do not import. The desired behavior can be changed in the Settings (1.13.11) dialog: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Communicator/GettingStarted/CommunicatorSettings.png}) You can also set other import options to automatically repair, stitch and simplify imported geometry. If you import a 3D model from SolidWorks, it makes sense to set an option to map Y axis into Z (because in SolidWorks the vertical axis is Y). For SolidWorks and other native CAD formats it is possible to specify alternate search path, which will be used to look for parts absent in the imported assembly folder.

4.2 Settings

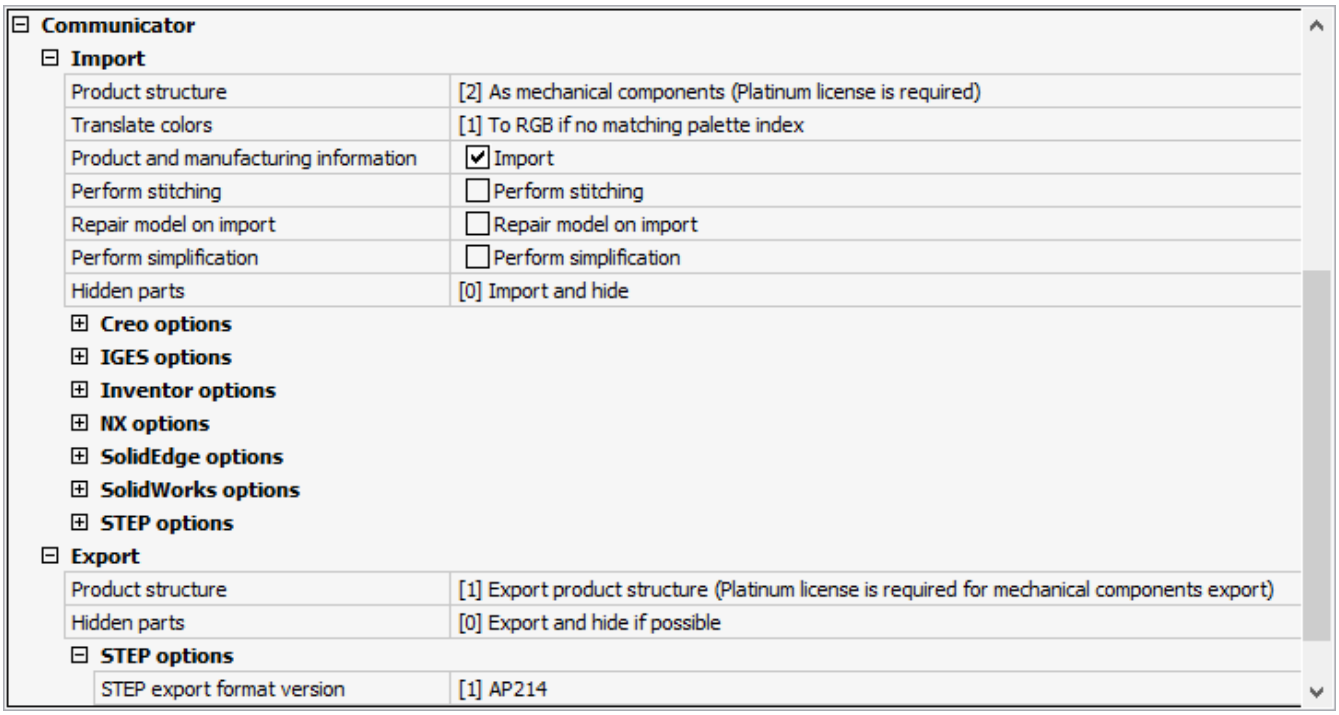
4.2.1 BricsCAD Communicator Settings

Commands: COMMUNICATORINFO (6.9.39), EXPORT (6.13.15), IMPORT (6.17.10), SETTINGS (6.27.23) The Communicator section in the Settings (1.13.11) dialog box allows to customize the behavior of the related commands:

- setting environment options.
- control the import/export of data.
- toggling post-processing operations.

Settings are applicable whe theEXPORT (6.13.15) andIMPORT (6.17.10) commands engage one of Communicator (4.1.1) file formats. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Communicator/Settings/Settings

Figure 4.4



Import Settings

Product Structure **Name**: ImportProductStructure **Description**: Determines how the imported entities will be structured and whether it will form a soup of entities, as a block hierarchy or as anassembly (3.6.2) structure. **Values**:

- **[0] None**:Creates a flat structure without blocks in the modelspace of the target database regardless whether the imported data have an assembly structure or not.
- **[1] As blocks**:The imported data have an assembly structure, which will be converted into a hierarchy of plain blocks; so the structure will be preserved, while the assembly metadata are lost. If the imported data is composed of entities only, they are placed in the modelspace of target database.
- **[2] As mechanical components**:In this mode data will be translated into BricsCAD assembly data, the structure and its properties (i.e. physical materials (2.6.10)). If the imported file has no assembly data, a mechanical component will be created in the BricsCAD target document root anyway.

Translate Colors **Name**: ImportColors **Description**: Determines how thecolors (1.17.4) of the imported entities will be assigned: either as true colors or matching to current color palette. **Values**:

- **[0] To RGB**:All the entity colors will be converted to RGB regardless of current palette.
- **[1] To RGB if no matching palette index**:If the entity color is found in the palette, then the entity gets an index color, else a true color.
- **[2] To nearest palette index**:For any true color of the imported entity the nearest match is searched in the palette and this indec color is assigned to the entity.

Product and Manufacturing Information **Name**: ImportPMI **Description**: Determines whether product manufacturing information is imported. Currently such information is imported as exploded data (lines, text, ...) instead of compound entities (e.g. annotations). **Values**: **on/off** checkbox.

Perform Stitching **Name**: ImportStitch **Description**: In some cases imported geometry representssolid (1.6.17) geometry as a set of separate surfaces. In order to be able to work with solid operations on the imported geometry, theDMSTITCH (6.12.28) command has to be applied. If this settings = ON, the DMSTITCH command is executed automatically when the geometry is imported. **Notes:**

- Stitch operations are time consuming when importing large files.

- Check theImportIgesStitch setting,which can set an override for the IGES file format.

Values: **on**/**off** checkbox.

Repair Model on Import **Name**: ImportRepair **Description**: Applies operations similar to theDMAUDIT (6.12.2)command in order to improve the quality of the imported geometry. Geometry modeled in CAD systems which use a kernel different from ACIS, often needs to be healed because of possible flaws. **Values**: **on**/**off** checkbox.

Perform Simplification **Name**: ImportSimplify **Description**: Applies operations similar to theDMSIMPLIFY (6.12.27)command:

- Convert imported splines into canonic surfaces.
- Simplify topology (remove imprinted edges) if possible.

Note: Also check the ImportIgesSimplifysetting, which can set an override for the IGES file format. **Values**: **on**/**off** checkbox.

Hidden Parts **Name**: ImportHiddenParts **Description**: Defines how hidden (invisible) entities in the imported geometry are processed. **Values**:

- **[0] Import and hide**:All entities are imported; invisible entities are hidden. Note that currently there are no user tools to make these hidden entities visible again.
- **[1] Import and set visible**: All entities are imported and visible with regardless of the visibility in the source file.
- **[2] Do not import**:Hidden entities in the source file are not imported.

Creo Options

Alternate Search Paths **Name**: ImportCreoAlternateSearchPaths **Description**: This setting is taken into account when Creo files are imported. The value is a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations. **Values**: string; if empty, the search is performed in the folder of the imported file only.

IGES options

Perform Stitching **Name**: ImportIgesStitch **Description**:In some cases imported geometry representssolid (1.6.17) geometry as a set of separate surfaces. In order to be able to work with solid operations on the imported geometry, theDMSTITCH (6.12.28) command has to be applied. If this settings = ON, the DMSTITCH command is executed automatically when the geometry is imported. This setting determines whether astitchoperation has to be applied to imported entities when the format of the source file is IGES. **Notes:**

- Stitch operations are time consuming when importing large files.
- For file formats other than IGES, the ImportStich setting applies.

Values: **on**/**off** checkbox.

Perform Simplification **Name**: ImportIgesSimplify **Description**: Determines whether asimplification (6.12.27)operation has to be applied to the imported geometrywhen the source file format is IGES **Note:** For other formats see the ImportSimplifysetting. **Values**: **on**/**off** checkbox.

Inventor Options

Alternate Search Paths **Name**:ImportInventorAlternateSearchPaths **Description**: Setting is taken into account when Inventor files are imported. The value is a set of semi-colon (';') delimited, fully qualified paths used to search for missing assembly references during product structure import operations. **Values**: string; if empty the search is performed in the folder of the imported file only.

NX Options

Alternate Search Paths **Name**: ImportNxAlternateSearchPaths **Description**: Setting is taken into account when NX files are imported. The value is a set of semi-colon (';') delimited, fully qualified paths used to search for missing assembly references during product structure import operations. **Values**: string; if empty search is performed in the folder of the imported file only.

SolidEdge Options

Alternate Search Paths **Name**: ImportSolidEdgeAlternateSearchPaths **Description**: Setting is taken into account when SolidEdge files are imported. The value is a set of semi-colon (';') delimited, fully qualified paths used to search for missing assembly references during product structure import operations. **Values**: string; if empty search is performed in the folder of the imported file only.

SolidWorks Options

Map SolidWorks Y to BricsCAD Z axis **Name**: ImportSolidWorksRotateYZ **Description**: Corrects the difference between the SolidWorks and BricsCAD coordinate systems. If unchecked, imported geometry in BricsCAD is rotated by 90 degrees compared to its position in SolidWorks. **Values**: **on**/**off** checkbox.

Alternate Search Paths **Name**: ImportSolidWorksAlternateSearchPaths **Description**: Setting is taken into account when SolidWork files are imported. The value is a set of semi-colon (';') delimited, fully qualified paths used to search for missing assembly references during product structure import operations. **Values**: string; if empty search is performed in the folder of the imported file only.

STEP Options

Map Y to BricsCAD Z axis **Name**: ImportStepRotateYZ **Description**:Corrects the difference between coordinate systems in different coordinate systems, i.e. SolidWorks and BricsCAD. This can be helpful to prevent undesired rotation when the origin CAD of STEP file is known, i.e. if it is SolidWorks. **Values**: **on**/**off** checkbox.

Export Settings

Product Structure **Name**: ExportProductStructure **Description**: Determines whether theassembly (3.6.2) structure will be preserved in the target document. **Values**:

- **[0] No product structure**:Exports a flat structure without components in the target document whether the BricsCAD document has a product structure or not.
- **[1] Export product structure**: Exports the BricsCAD product structure data (if existing) to the target document.

Hidden Parts **Name**: ExportHiddenParts **Description**: Determines if invisible entities are exported or not. Entities can be invisible because of:

- the result of theHIDEOBJECTS (6.16.11) command.
- sitting on a hidden layer. (6.20.2)
- owned by aninvisible (3.6.3) component.

Values:

- **[0] Export and hide if possible**: Exports hidden entities. If the target format supports hidden entities, hidden entities in the source document will be hidden also in target document.
- **[1] Do not export**: Invisible entities are skipped.

STEP Options

STEP Export Format Version **Name**: ExportStepFormatVersion **Description**: Allows to select available specifications of the STEP format. **Values**:

- **[0] AP203**: Selects AP203 protocol.
- **[1] AP214**: Selects AP214 protocol.
- **[2] AP242**: Selects AP242 protocol.

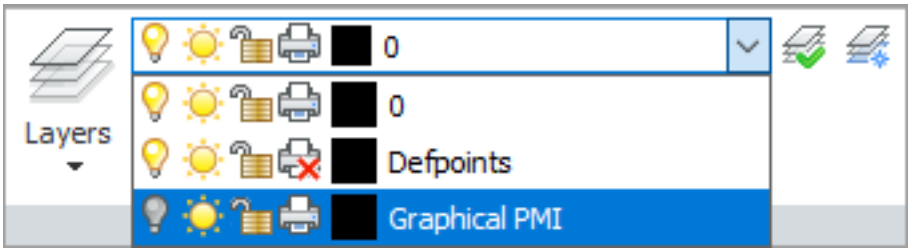
4.3 Tutorials

4.3.1 Import PMI

You need**BricsCAD Platinum**and **Communicator** licenses for this tutorial.

1. Create a new file.
2. Launch theImport (6.17.10)command.
3. Select the Geared_Traction_Machine_2_PMI.SLDPRfile in the **Import file** dialog box. Click the **Ok** button.
4. Switch off the **Graphical PMI** layer in the drop-down **Layers** menu, in the **Home**| **Layers**panel, on theRibbon (1.13.6). (\vref{fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360038743253/mceclip0.png})
5. Click the **Audit (6.12.2)** ( ) tool button on the**Sheet Metal** | **Heal** panel on theRibbon (1.13.6).
6. Click **Entire model** on the **DmAudit** prompt menu or press**Enter.**
7. Click **Fix** on the **DmAudit** prompt menu or press **Enter**.
8. Save asGeared_Traction_Machine_2_PMI.dwgfile.

Figure 4.5



Chapter 5

BrisCAD Shape

5.1 Getting Started

5.1.1 The Shape user interface

https://www.youtube-nocookie.com/embed/s1IIETo2E3U Let's explore the Shape user interface. The toolbar at the top of your screen gives you fast access to the most used tools. Left-click on any icon in the toolbar to use the tool. Many top-level tools offer sub-menus. Hover over the icon, left click and hold. The sub-menu will fly out to show these additional tools. The icons on the right side of your screen give you access to a set of collapsible panels The top-most icon opens the Tips panel. You'll find more details on how to use the tools in Shape here. We'll tell you more about the other panels in later tutorials.

5.1.2 How to navigate in BricsCAD Shape

https://www.youtube-nocookie.com/embed/Ce9KIjYbarE Model navigation in BricsCAD Shape can be done in three ways. The Pan command slides your viewpoint. You can click and hold your mouse wheel to Pan.Press and hold CONTROL + SHIFT and the right mouse button to Pan. Or you can start the Pan command. The Zoom command moves your viewpoint in and out. You can scroll your mouse wheel to Zoom in and out. You can press and hold CONTROL + SHIFT and the left mouse button to Zoom. Double-click the mouse wheel to Zoom Extents. The second option is tostart the Zoom command. You can rotate your viewpoint around the model. Press and hold SHIFT and the mouse wheel, then move the mouse to rotate your viewpoint. Or you can start the real-time rotate command. Finally, the “lookfrom” widget lets you select preset 3D views quickly.

5.1.3 How to select elements in BricsCAD Shape

https://www.youtube-nocookie.com/embed/ROAIqPfdiNE Model elements in BricsCAD Shape are solids. They are made up of edges and faces. You can select faces by moving your cursor over the face and left-clicking your mouse. Press the ESC key to clear your selection. You can select edges by moving your cursor over the edge. Then press & hold CTRL and left-click your mouse. To select an entire solid, move the cursor over any face of the solid, press & hold CTRL and left-click your mouse. To select multiple faces, just left-click them, one-by-one. To select multiple edges or multiple solids, press & hold CTRL and left-click them, one-by-one. Sometimes, the element or elements you want to select are hidden behind others. To select hidden elements, press the TAB key while hovering over the element. Each time you press TAB, you'll cycle to the next element underneath the cursor. When the element you want is highlighted, just left-click to select it.

5.1.4 How to use the Quad cursor & Rollover tips in BricsCAD Shape

https://www.youtube-nocookie.com/embed/SecgCMi2LiI

In BricsCAD Shape, the user interface includes everything that you need and nothing that you don't. Hover over a model element, and a Rollover Tip will display that element's properties. Different types of elements will display different property sets. The Quad cursor is a context-sensitive tool panel. Right-click on any empty space on your screen, and you'll see the Quad in its "No Selection" state. Move your cursor onto the displayed icon, and the Quad expands. The most relevant commands are on the top bar, and the pulldowns will show all available commands by category. Next, move your cursor over any model element. The Rollover Tip will appear. Then, move your cursor onto the Rollover Tip. The Quad opens, and shows you the tools that you can use on the selected element.

5.1.5 How to work with layers in BricsCAD Shape

https://www.youtube-nocookie.com/embed/p1QFV9sdf5g

Using layers to group model elements is a time-tested way to organize 3D models. Open the Layer panel by left-clicking the “stacked sheets” icon. It is located on the vertical toolbar, on the right-hand edge of your screen. Turn layers on and off in the Layer Panel. Do this by moving the cursor over the lightbulb icon to the right of each layer name, and left-clicking the mouse. Toggling layers is a simple way to isolate model elements, to help with navigation and viewing. You can create a new layer by left-clicking the green “plus sign” at the top of the layer panel. The current layer is indicated by the blue dot, shown in the column to the left of the layer names. To change the current layer, just left-click in that column. To change the layer of a model element, hover over the element, move the cursor onto the Rollover Tip, and click the desired new layer name in the “Layer” pull-down list.

5.1.6 How to hide and isolate objects in BricsCAD Shape

https://www.youtube-nocookie.com/embed/NHJpHaZvC9o

When model elements block your view, you can HIDE them easily. To hide one or more elements, just highlight them, raise the Quad and left-click 'Hide Entities'. The opposite of hiding an element is isolating it. Highlighting one or more entities, and selecting 'Isolate Entities' from the Quad will hide all the non-highlighted elements in your model. Hiding and Isolating are temporary functions that you can use to simplify your model while working on it. When you want to see all the elements in your model, just raise the "No Selection" Quad and left-click "Show Entities".

5.1.7 How to change the visual style in BricsCAD Shape

https://www.youtube-nocookie.com/embed/PIqsRBwYsvk

BricsCAD Shape uses Visual Styles to control the appearance of your model. Open the Visual Styles panel by left-clicking the “camera” icon, located on the vertical toolbar, on the right-hand edge of your screen. Each defined Visual Style, when selected, will change the look of your entire model. In the Visual Styles panel, you can change the perspective view. Moving the perspective slider all the way to the left will turn perspective off. The Quality slider controls the on-screen rendering quality of your model.

5.2 Getting Started

5.2.1 The basic concepts of modelling

Navigation

Useful shortcuts:

- Hold down the **middle mouse button** to **pan**
- Hold down **shift** and the **middle mouse button** to **rotate**
- Scroll the **mouse wheel** to **zoom in or out**
- Click any of the locations on the 'LookFrom' widget to view your model from a certain angle

Highlighting vs Selecting

When hovering over an object, it will be highlighted. Left-clicking when an entity is highlighted will select this entity. It is not always necessary to select an entity to perform an operation on it; for example you can highlight a surface, which will trigger the Quad to open. Then you can find the command you want to perform on that surface, without having to select the surface first. You can only **highlight** one entity at a time, whereas you can **select** as many entities as you want. For this reason, highlighting is particularly useful when performing an operation on one single entity.

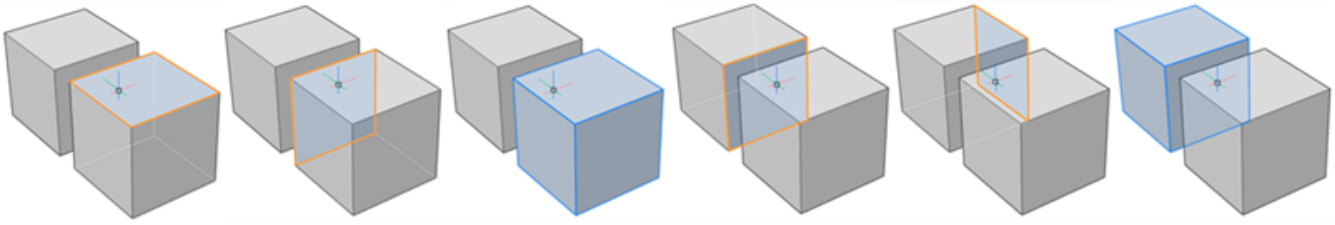
Selection modes

A solid has different subentities: faces and edges. It is possible to select these faces or edges individually to perform operations on them, for example offsetting an edge or extruding a face. BricsCAD Shape allows you to select any of these subentities using the **Ctrl** and **Tab** keys.

To highlight a solid **face**, just hover over it with your cursor. The face will be displayed in orange. To select it, left-click it while it is highlighted. You can select multiple solid faces by left-clicking them one by one. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e	To highlight a **solid**, hover over one of its faces while holding down the **Ctrl** key. The solid will be displayed in blue. To select it, left-click it while it is highlighted. You can select multiple solids by left-clicking them one by one. (\vref {fig:C:/Dev/Task-	To highlight an **edge**, hover over the edge while holding down the **Ctrl** key. The edge will be displayed in orange. To select it, left-click it while it is highlighted. You can select multiple edges by left-clicking them one by one. (\vref {fig:C:/Dev/Task-

Pressing the **Tab** key while keeping the cursor at the same position will cycle through all entities or subentities that are 'underneath' the cursor. This can be used for example to select an entity that is not visible because it is behind another element. For instance, to select the back face of the front cube in the image below, or one of the faces of the second cube, you can keep the cursor at the same location while pressing **Tab** to cycle through the different entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/GettingStarted/ShapeBasicCycle.png Thus, using a combination of the **Ctrl** and **Tab** keys you can highlight and select any entity type or subentity type.

Figure 5.1



Selection windows

A selection window can be used to select one or multiple elements at once. To use a selection window, just left-click on any empty space in your model area, and move the cursor until the window covers whatever you want to select. There are two types of selection windows:

- Blue selection window: appears when you create the window from **left to right**. This selection window will only select elements that are entirely covered by this selection window.
- Green selection window: appears when you create the window from **right to left.** This selection window will select all elements that are entirely covered by the selection window, plus all the elements that intersect with one of the edges of the window.

In this example, the blue selection window will only select the left cube, whereas the green selection window will select both cubes. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/GettingStarted/ShapeBasicSe
A selection window will by default only select entire elements (i.e. solids, lines, polylines, blocks, ...) but not subentities (such as solid faces or solid edges). By pressing **Ctrl** once while drawing the selection window, you will switch to selecting only solid faces. The selection mode of the window will be indicated by a small icon in the bottom right corner: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/GettingStarted/ShapeBasicSelectionW
By pressing **Ctrl** twice while drawing the selection window, you will switch to selecting only solid edges: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/GettingStarted/ShapeBasicSelectionWindowEdges.

Figure 5.2

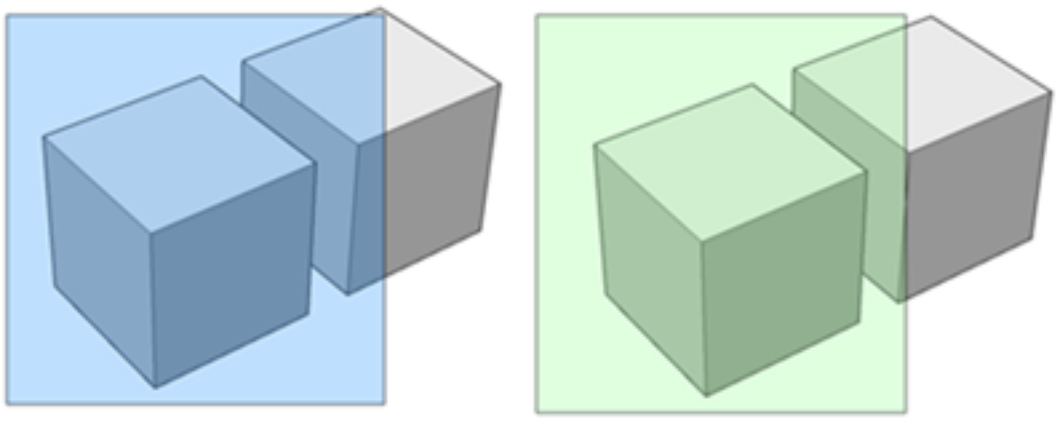


Figure 5.3

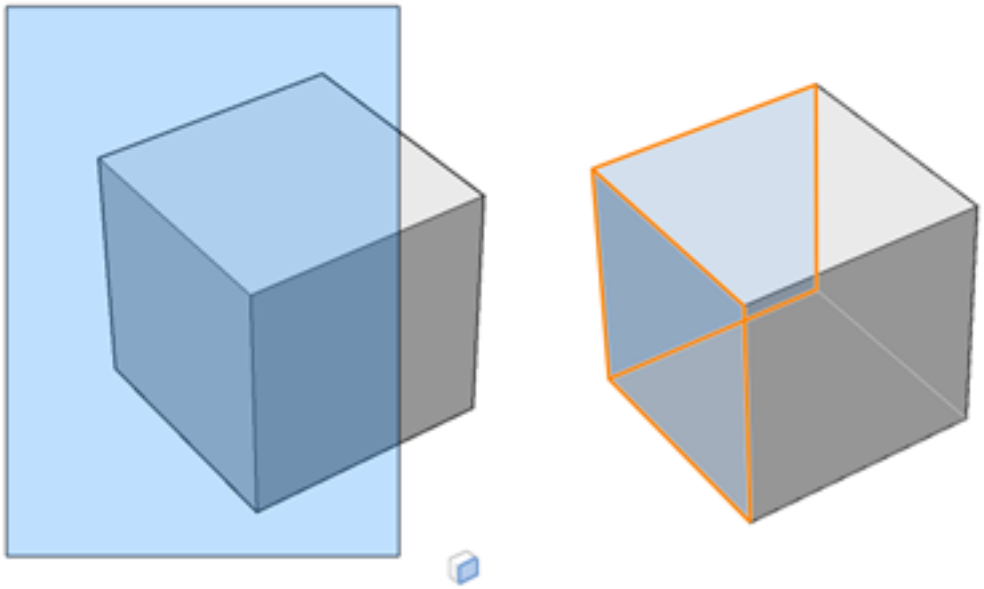
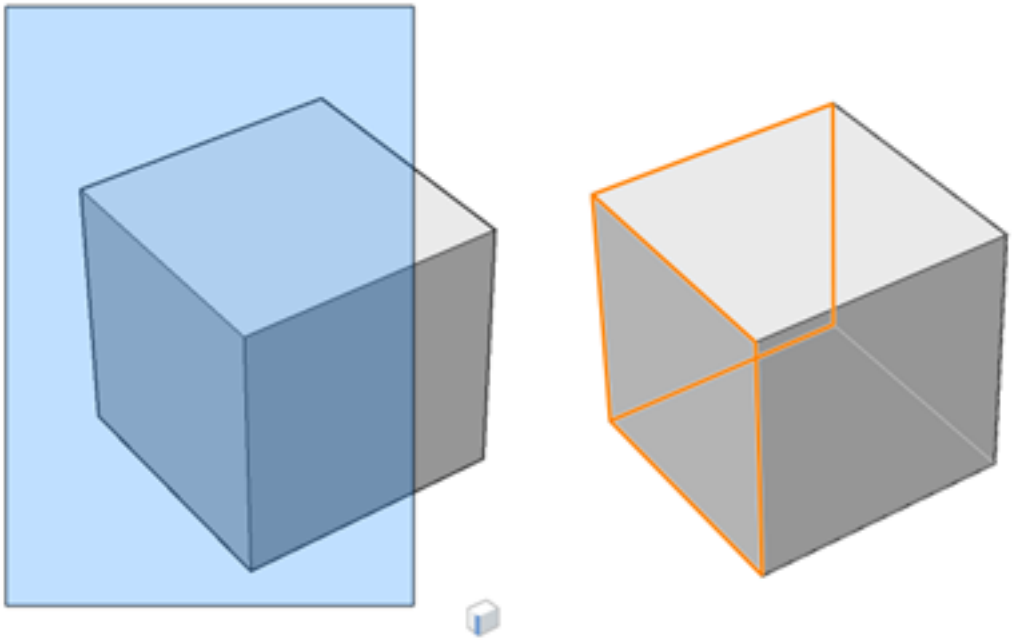


Figure 5.4



Boundaries vs Faces

Boundaries on top of planar faces will be detected automatically. A boundary can be created by a set of 2D entities (lines, arcs, circles…) on top of a surface (e.g. a solid face): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
The area marked in green is an automatically detected boundary. This boundary can be used to create a new solid, extrude a hole in the existing solid, create an offset, … Another way a boundary can be formed is by multiple solids that have touching faces: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/GettingStarted/Shap
The area marked in green is an automatically detected boundary. If you want to select the entire face of the box instead of this boundary, you can press the **Tab** key to cycle through subentities, as explained earlier.

Figure 5.5

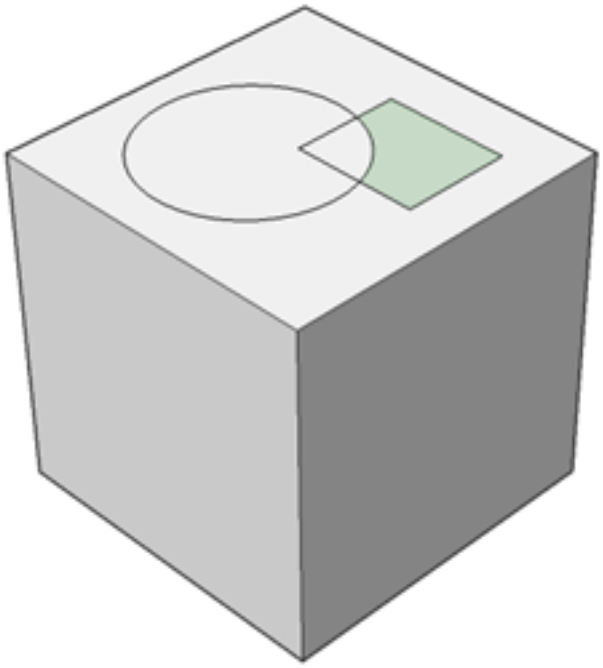
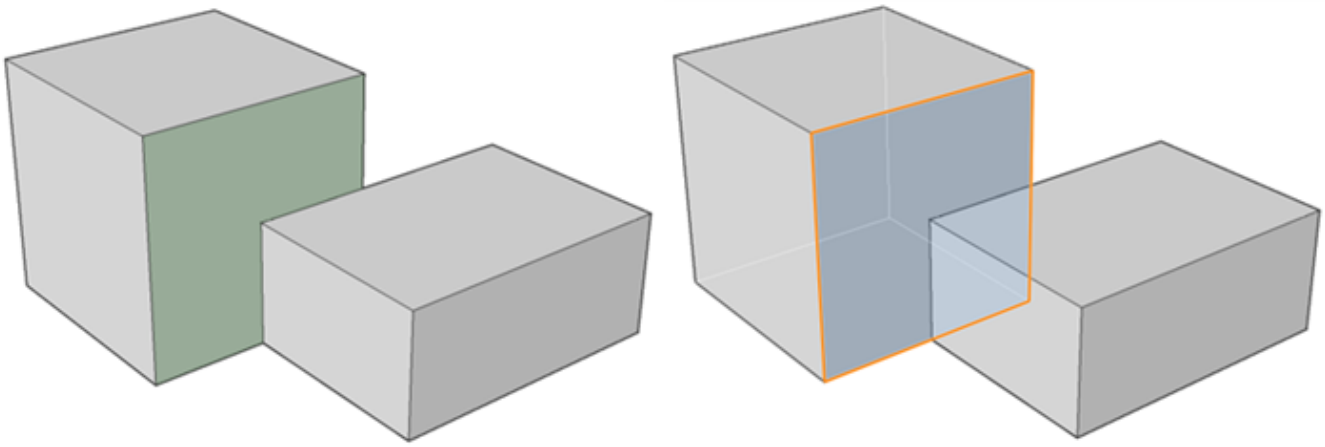


Figure 5.6



Using the Quad & Rollover Tips

When highlighting an entity, a 'toolbar' appears at your cursor. This is called the Rollover Tip, and consists of:

- An overview of **what you have highlighted** or selected (in the example below: a 'solid face')
- The icon of the **last used command on this entity type** (in the example below: 'copy')
- Some relevant **properties** (in the example below: color, layer and area)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/GettingStarted/ShapeBasicRT.png})
When moving the cursor **over the icon** (without clicking it), the toolbar expands. This expanded toolbar is called the **Quad**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/GettingStarted/ShapeBasicQuad.png})
The top row of icons displays the most recently used commands on this entity type. Note that the Quad will adapt to the entity type you have highlighted or selected. When highlighting a solid face, it will display **different tools** than when highlighting a solid edge, a line or a block reference. It will only show tools that are relevant for the selected entity type. Beneath the row of icons are different tabs. Hovering the cursor over one of these tabs will expand the Quad even further, showing even more tools that might be relevant for the selected entity type. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/GettingStarted/ShapeBasicQuadExpanded.png})
The Quad can also be called on a **selection set**. Say you want to use the Copy command on 3 solids, you can either:

- **select** the first two solids and **highlight** the third solid to open the Quad
- **select** all three solids and hover over one of the solids to open the Quad
- **select** all three solids and **right-click** somewhere on an empty space on the canvas to open the Quad

Finally the Quad can also be called when **nothing** is highlighted or selected. Just right-click on an empty space on the modelling canvas to open the **No-selection Quad**. Alternatively to the Quad you can also use the Command Line or the Toolbar at the top of your screen. Icons that display a little triangle in the bottom right corner are expandable toolbars containing similar tools. Expand these toolbars by holding the left mouse button down slightly longer on the icon.

Figure 5.7

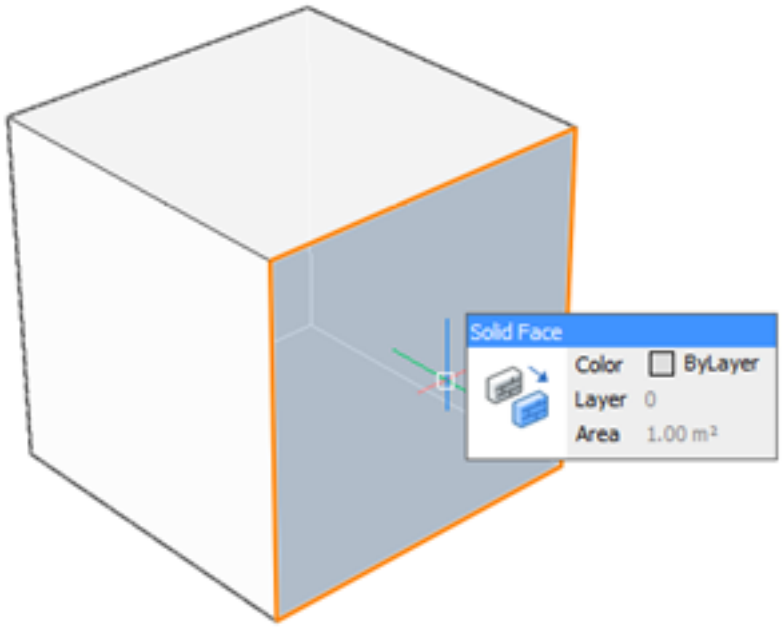


Figure 5.8

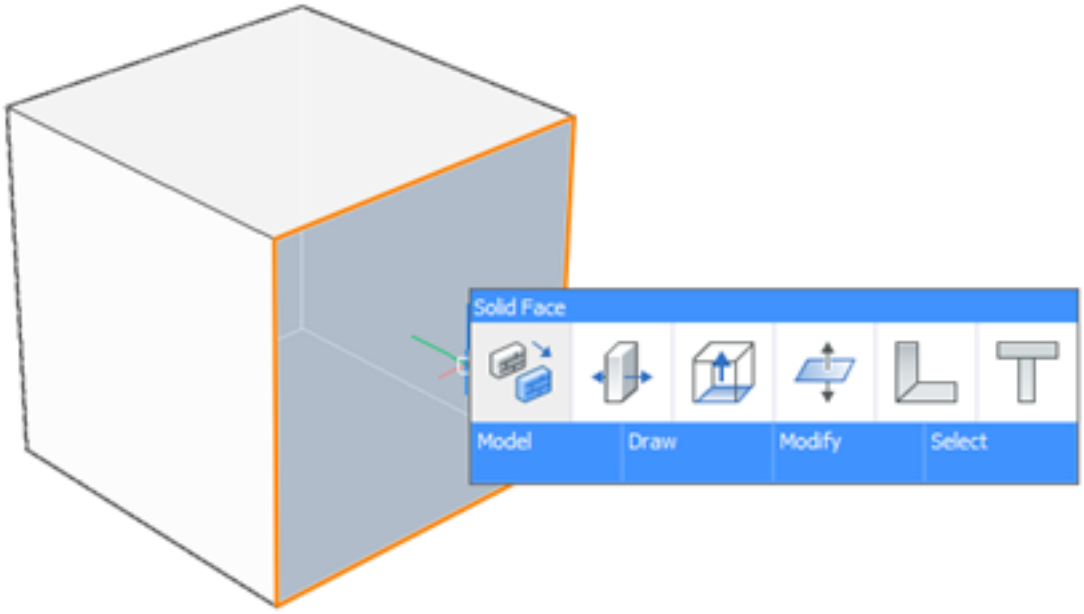
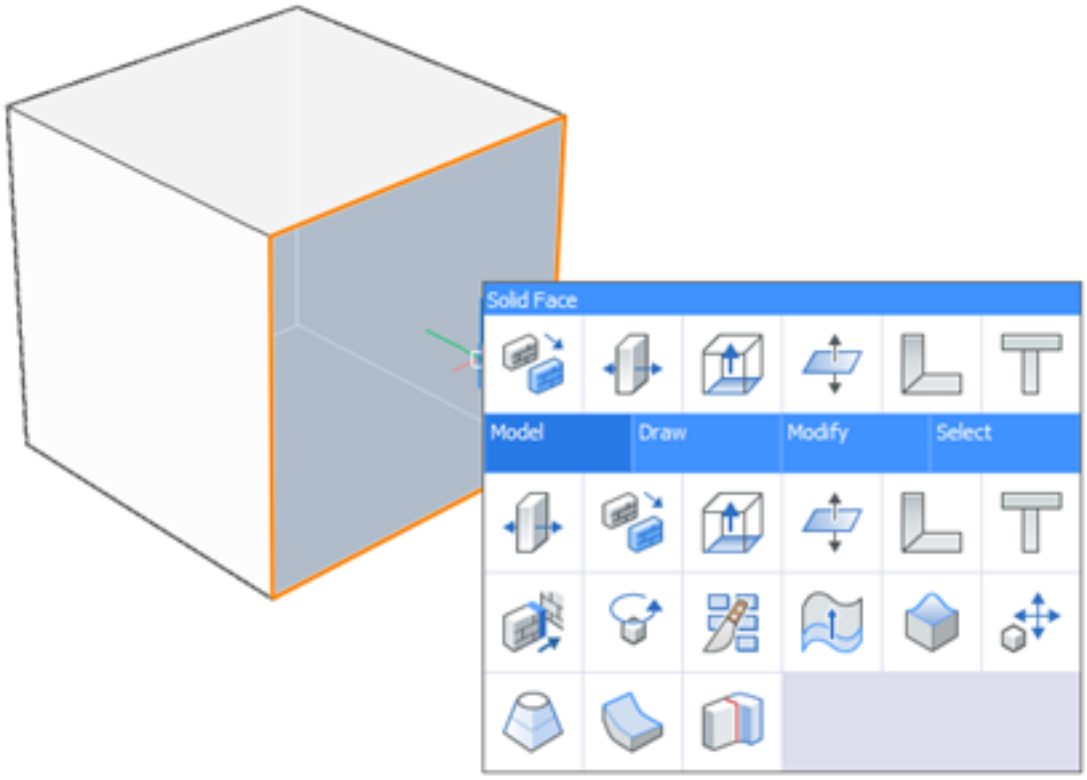


Figure 5.9



Exercise on basic concepts

Procedure: Drawing a box

1. Open the No-Selection Quad by right-clicking on an empty place in the canvas and moving the cursor over the icon
2. Under the Model tab, click the **BOX** icon
3. You are prompted: **Set corner of box or [Center]:**
4. Type in **0,0,0** and press **Enter**
5. Two dynamic dimensions are displayed that control the length and width of the box (see image). The dynamic dimension that is highlighted in blue is the active one. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
6. Type in a value of **1000** and press the **Tab** key to switch to the other dynamic dimension

Figure 5.10

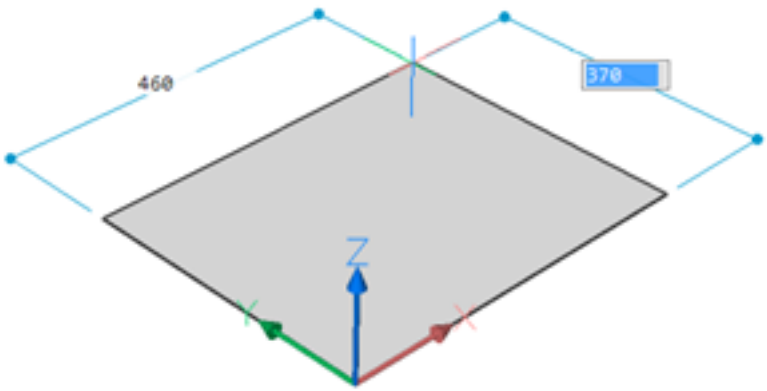


Figure 5.11

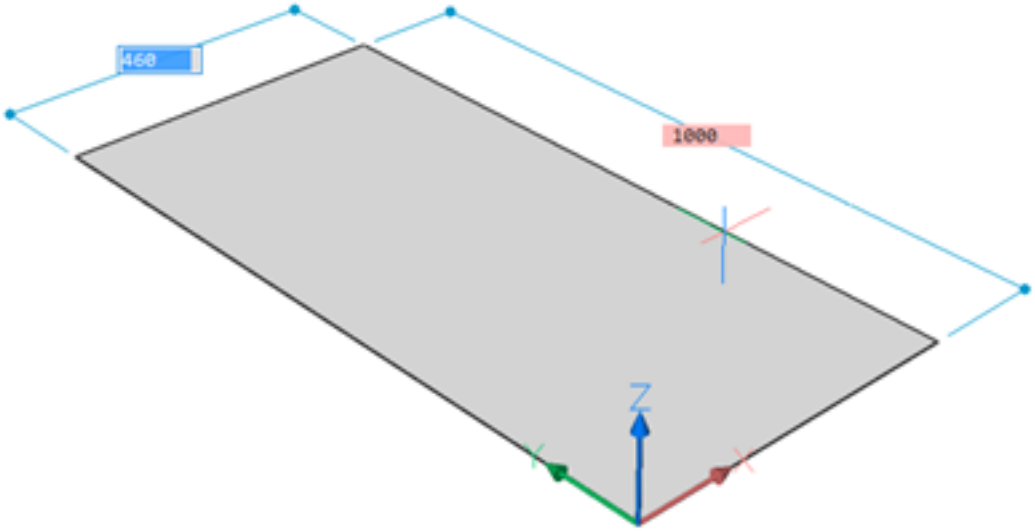
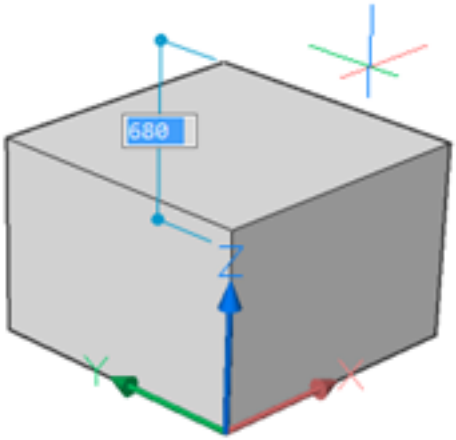


Figure 5.12

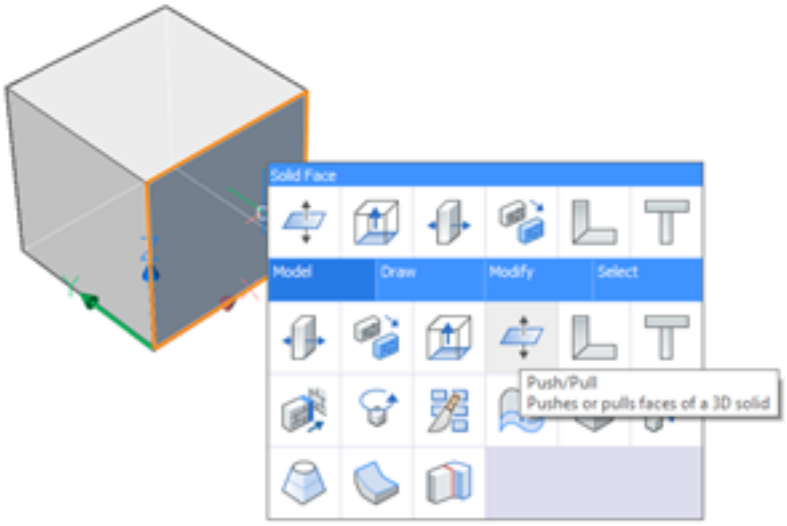


7. Note that the first dimension is now highlighted in pink. This is to indicate that a value was entered here, and this value is now locked on this dynamic dimension. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
8. Type in a value of **1000** and press **Enter**. Alternatively, you can press **Tab** again to switch back to the first dynamic dimension
9. Finally a dynamic dimension appears that controls the height of the box. Type in a value of **1000** and press **Enter**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/GettingStarted/ShapeBasicDynDimI

Procedure: Manipulating a face

1. Move your cursor over a side face of the box until it is highlighted
2. Open the Quad by moving your cursor over the icon
3. Under the Model tab of the Quad, click the **PUSHPULL** icon (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
4. Type in a value in the dynamic dimension box and press **Enter**, or left-click on the location where you want to pull this face to.

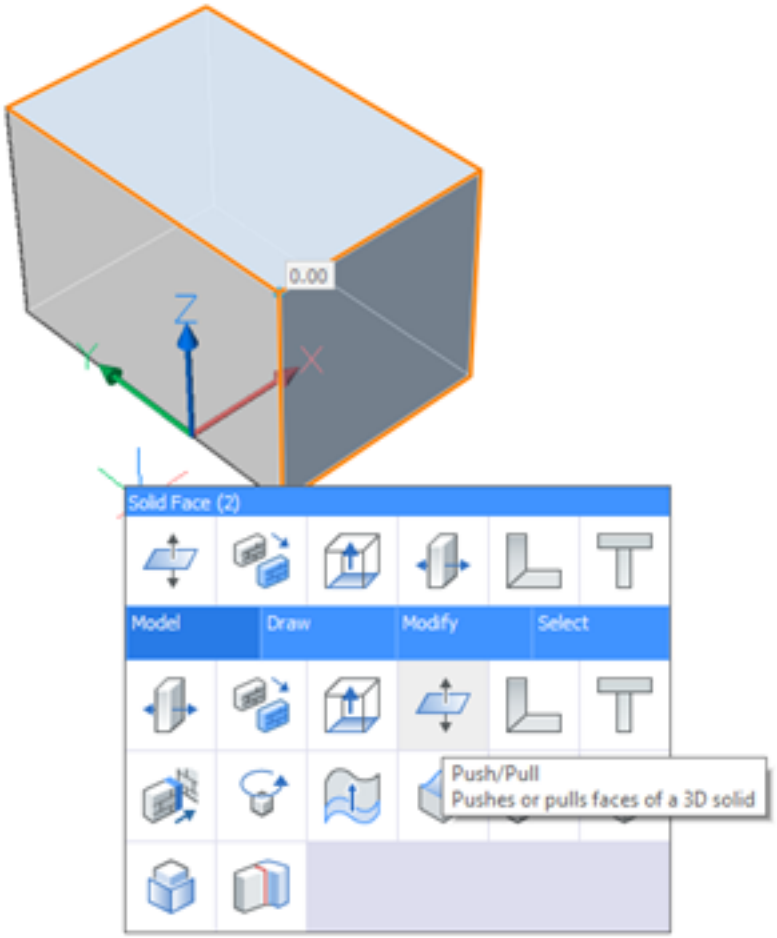
Figure 5.13



Procedure: Manipulating multiple faces

1. Select a side face of the box by left-clicking it
2. Select the top face of the box
3. When you have both faces selected, right-click anywhere on an empty space in the canvas to open the Quad
4. Under the Model tab, click the **PUSHPULL** icon (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
5. Both faces will now be pushed in or pulled out over equal distances.
6. Type in a value and press **Enter** or left-click.

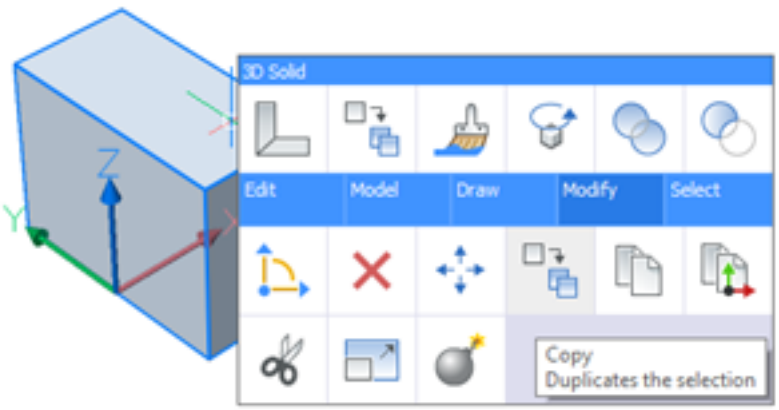
Figure 5.14


0.00
Z
Solid Face (2)
Model
Draw
Modify
Select
Push/Pull
Pushes or pulls faces of a 3D solid

Procedure: Copying a solid

1. Highlight the box by holding the **Ctrl** key and hovering over one of the faces of the box. Instead of the face, the entire solid should be highlighted now.
2. Open the Quad by moving your cursor over the icon, and under the Modify tab, click the **COPY** icon (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/GettingStarted/ShapeBasicQuadCopy.png})
3. You are prompted for a **Base Point.** Left-click one of the corners of the box.
4. You are prompted for the **Second point.** Left-click where you want the copy of the box to be.
5. You can continue placing copies, or press **Enter** or **Esc** to exit the command.

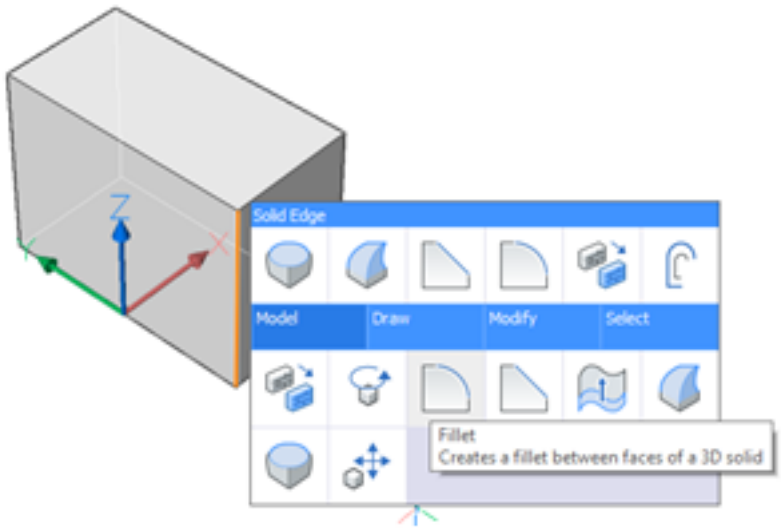
Figure 5.15



Procedure: Manipulating an edge

1. Highlight an edge of the box by holding the **Ctrl** key and hovering over one of the edges of the box. Instead of the face, the edge should now be highlighted.
2. In the Quad under the Model tab, click the **FILLET** icon (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
3. Type in a fillet radius and press **Enter.**

Figure 5.16



Procedure: Manipulating multiple edges

1. Instead of giving one edge a fillet, we will fillet all the edges of the box at once
2. Press **Ctrl + Z** to undo your previous action, so the fillet of the previous step is removed.
3. Create a selection window around the entire box, and while drawing this window, press the **Ctrl** key twice until a small icon indicating 'edge selection' appears. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/S
4. You should now have selected all 12 edges of the box. In the Quad under the Model tab, click the **FILLET** icon
5. Type in a fillet radius and press **Enter.** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Shape/Ge

Figure 5.17

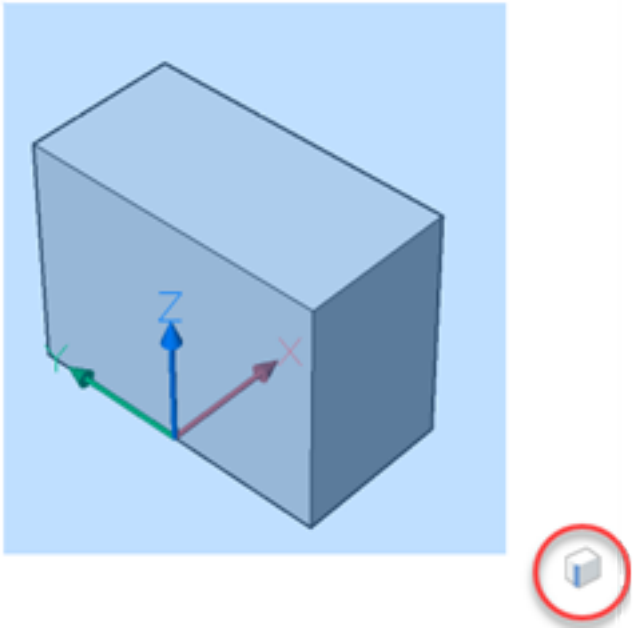


Figure 5.18

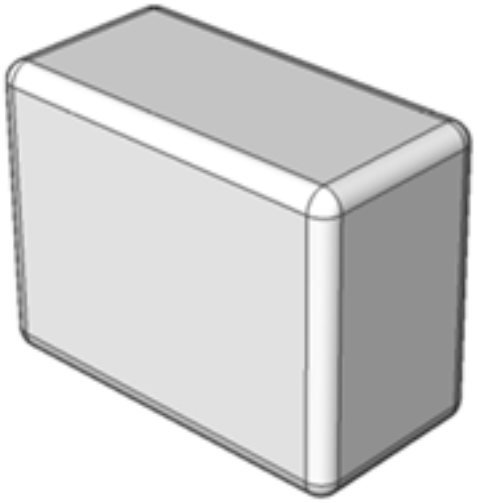


Figure 5.19

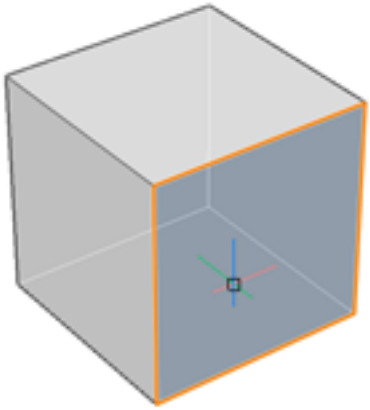


Figure 5.20

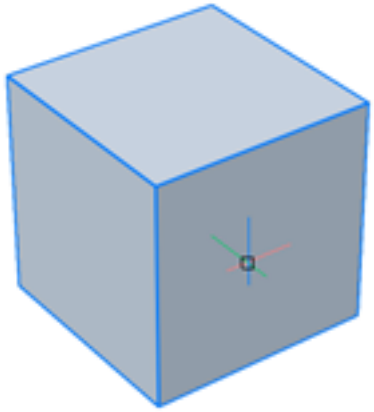
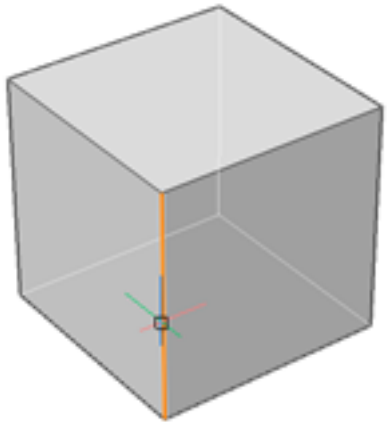


Figure 5.21



5.2.2 Welcome to BricsCAD Shape

https://www.youtube-nocookie.com/embed/QCsp_NKtOlo

What is BricsCAD Shape?

BricsCAD® Shape is the new conceptual modeling tool from Bricsys. It's made to help architects and engineers streamline their 3D schematic design work. Shape is the fastest way to get designs out of your head and in front of your clients. It's CAD-accurate, using a powerful direct modeling engine. You can use it as a live presentation tool, too. Potentially best of all, Shape offers a clean, simple user interface that you can learn in 30 minutes or less.

BricsCAD Shape lets you create without limits

Shape's smart modeling tools make it easy to capture in 3D. The multi-functional "Shape" tool can be used to create walls, slabs, structural elements and more. Direct modeling features let you drag, connect, push/pull and extrude as you like. Wall connections are automatically mitered for you. BricsCAD Shape also offers everything you need to present your designs. It comes with a library of materials, textures and pre-built 3D components. You can use the included collection of doors and windows, or create your own custom versions easily. Make your model look the way you want using the Visual Styles panel in Shape.

How is BricsCAD Shape different from other tools?

First, BricsCAD Shape is CAD-accurate from the start. Under the simple UI is the same fast and proven parametric, 3D direct modeling engine that's in BricsCAD Platinum. Shape creates solid models, stored in industry-standard DWG – just like BricsCAD® BIM. Unlike competing products, Shape's solid models are fully accurate – never approximated. Every element that you create in Shape can be modified deeply, anytime, on the fly. Your concept models open directly in BricsCAD BIM. You'll never lose time starting over again. Make the smart move – directly from concept to BIM with BricsCAD Shape with no loss of detail.

BricsCAD Shape accelerates your BIM workflow

The Bricsys BIM workflow starts in 3D with Shape, and stays in 3D – speeding 2D construction documentation with the world's best drafting tools. No breaks or transitions in your workflow, all in DWG, with a friendly, familiar user interface that you'll appreciate. It won't take long for you to be productive in BricsCAD Shape or BricsCAD BIM.

Stop sketching. Start shaping!

Get your free copy of BricsCAD Shape! BricsCAD Shape delivers the tools you need, and none that you don't. Even the learning materials are clean and simple – check out "30 minutes to Shape" on YouTube.

5.2.3 Further documentation

Many of the modelling tools that are used in BricsCAD and BricsCAD BIM are also available in Shape. For this reason we refer to the following articles for a more in depth explanation of the different tools: BricsCAD - Direct Modeling Operations (1.18.2) BricsCAD BIM - Modeling Techniques Furthermore, you can get an overview of all available commands in Shape by typing in COMMANDS in the command line.

5.3 3D Modeling

5.3.1 How to create a wall in BricsCAD Shape

https://www.youtube-nocookie.com/embed/EuAnBK8BdVI Select the start point of the wall. Move the cursor to the next point, and left-click it. You can continue creating joined wall segments. Simply right-click your mouse, or press ENTER, to stop creating additional wall segments. Move your cursor up to set the wall height, or type in a value. Right-click to accept the default wall width, or press the TAB key to type in a new value. Values you enter here become the new defaults for wall height and width. In Shape, it's easy to use new or existing linework as a guide. The cursor snaps to line endpoints automatically. The TOOLTIP at the bottom of the screen reminds you to press the CTRL key to cycle through the wall justification options: left, center or right.

5.3.2 How to create a layout in BricsCAD Shape

https://www.youtube-nocookie.com/embed/QdFOvGo_AKs You will use the Shape tool. Select your start point, move the cursor to the next point and left-click again. Shape automatically creates wall connections. Press the enter key or right-click once to accept. You can move your cursor to set the wall height, or simply type in a new height. Then, you can press the TAB key to change the width, or just right-click to accept the default value. You can also use an existing 2D plan to start your layout. Use the Shape tool. Press E, for “Entity” and select the outside border. Move up to set the height, type in a value, or right-click to accept the default value. You can press the CTRL key to cycle through the wall justification options: left, center or right. Finally, let’s use the Shape tool with the “E” option to finish the interior walls.

5.3.3 How to use selection boxes in BricsCAD Shape

https://www.youtube-nocookie.com/embed/h4wMXoQ4Ozw

You can select multiple model elements using a selection box. By default, selection boxes see and select entire solids. If you press the CTRL key ONCE while the selection box is active, you will only see faces. The tooltip at the bottom of the screen shows these different modes. The CTRL key cycles between these modes. If you press CTRL once more, you will only see edges. Draw a selection box from left to right, it will display in blue. Blue selection boxes will only see elements that are completely inside the box. Draw a selection box from right to left, it will display in green. Green selection boxes will see all elements that lie inside the box, AND all elements that cross the boundaries of the box.

5.3.4 How to detect boundaries - BricsCAD Shape

https://www.youtube-nocookie.com/embed/wsaOK8Q8ctc BricsCAD Shape automatically detects and highlights closed boundaries, as you hover over different model elements. When a boundary is detected, it will be highlighted. To select it, just left-click inside the highlighted boundary. After you select a boundary, you can use it to create new elements or modify existing elements. Remember, if you don't want to select the boundary but for example the face or the solid, just press TAB until the element you want is highlighted.

5.3.5 How to connect walls in BricsCAD Shape

https://www.youtube-nocookie.com/embed/n8gtD2HrNbQ

Connecting walls is easy Select two walls, raise the Quad and click the “CONNECT” tool. The TOOLTIP at the bottom of your screen reminds you that there are different connection options available.You can cycle through the three connection options by pressing the CTRL key. We can use the CONNECT tool to connect disjoint walls. Highlight any face, and the “Connect with Nearest” tool makes it happen.

5.3.6 How to extrude in BricsCAD Shape

https://www.youtube-nocookie.com/embed/5iJJ00KdEq8

The EXTRUDE tool creates 3D elements from 2D line work. You can EXTRUDE (6.13.21) a selected boundary, for example, to create a floor slab between existing walls. EXTRUDE can subtract geometry, also. Note the TOOLTIP that appears, reminding you of the CTRL key options for the tool. Pressing the CTRL key cycles through all of the available EXTRUDE options.

5.3.7 Using the Manipulator in BricsCAD Shape

https://www.youtube-nocookie.com/embed/6P1rlvp2nHE

The manipulator is a visual tool that rotates, moves, or mirrors solid elements, around or along a single axis. Press and hold the CTRL key, move the cursor to the element of choice and left-click. By holding down the left mouse button slightly longer, the manipulator should appear. Move the cursor onto one of the three manipulator arcs and left-click to rotate your solid. Then, move the cursor onto one of the three axis shafts and left-click for a constrained move along that axis. To move your selection in 2D, constrained to a plane, left-click one of the three white plane indicators. To mirror your selected solid, left-click one of the blue axis arrows. Precise manipulation is always available through the manipulator's dynamic dimensions. You can rotate and re-locate the manipulator on the selected solid by left-clicking on the white orientation dots.

5.3.8 Using components in BricsCAD Shape

https://www.youtube-nocookie.com/embed/a0rr3q7nfpI

You can drag and drop content from the right-side panel directly into your BricsCAD Shape session. Open the component library by left-clicking the “hot-air balloon” icon. It is located on the vertical toolbar, on the right-hand edge of your screen. The pull-down list box shows the component categories. Left-click on the category name to select it, or search for specific types of components in the Search box. Let’s place some furniture in our model. To select a component from the panel, hover the cursor over the component that you want to use, then click and hold the left mouse button. Next, simply drag the component into your model. Move it to the desired position and release the left mouse button. Hover over the icon, left-click and hold.

5.3.9 How to create windows and doors in BricsCAD Shape

https://www.youtube-nocookie.com/embed/oUUGQRZ70RA

BricsCAD Shape includes a collection of doors and windows in the components library. Open the components library by left-clicking the “hot-air balloon” icon. It is located on the vertical toolbar, on the right-hand edge of your screen. Click the “Doors” icon on the Components panel. To select a door from the panel, hover the cursor over the door that you want to insert, then click and hold the left-mouse button. Drag the door onto a wall. When releasing the left mouse button, you will be able to position the window in the wall precisely using Dynamic Dimensions. You can enter a precise value for a distance, or press TAB to go to the next value. The TOOLTIP at the bottom of your screen reminds you of your options. Press CTRL to toggle between door positioning and door size. Enter the desired values for the height and width of the door by pressing the TAB key. When you’re done sizing and placing the door, press the ENTER key. This is thesame for a window. Change “Doors” to “Window” in the pull-down list box. Drag a window into the wall. You can change the size and location of a window just as you did with the door.

5.3.10 Create a custom window in BricsCAD Shape

https://www.youtube-nocookie.com/embed/KQmMNQhh5Kg

You can create custom windows easily in BricsCAD Shape. Use the POLYLINE (1.17.12) tool to draw the outline of the window on the wall face. Use the POLYGON (6.24.43) tool to draw the outline of the window on the wall face. Use the RECTANGLE (1.17.19) tool to draw the outline of the window on the wall face. Move the cursor inside the window outline and, when the boundary is highlighted, choose “CREATE WINDOW” from the Quad. The available window styles are shown in a dialog box. Move your cursor over the window design of your choice, and left-click it. The window is automatically created. You can create a custom window from a wall boundary. Select face of the outer wall and use the “OFFSET (6.23.3)” tool from the Quad. Move the cursor to re-size the new boundary, or key in a value and press ENTER. Then, highlight the offset boundary and select “CREATE WINDOW” from the Quad. Once again, the available window styles are shown in a dialog box.

5.3.11 Using materials in BricsCAD Shape

https://www.youtube-nocookie.com/embed/wj1SbA7Nz7o

BricsCAD Shape includes a collection of materials and color swatches in the materials library. Open the materials library by left-clicking the “paintbrush” icon. It is located on the vertical toolbar, on the right-hand edge of your screen. To assign a material to an element in your model, hover the cursor over the material or color swatch that you want to insert, then click and hold the left-mouse button. Drag the material onto the model element, and release the mouse button. You can filter by selecting a category from the drop-down list It is possible to apply a material to multiple elements at once by preselecting them.

5.3.12 How to create sections in BricsCAD Shape

https://www.youtube-nocookie.com/embed/bV56_fdn7MU You can use Sections to cut through your model dynamically. A Section can be created by opening the Quad and left-clicking 'define section', found under the Model tab. Then, selecting any solid face in your model will create the Section, parallel to the selected solid face. If you don't choose a surface to attach the Section, a horizontal Section plane will be created. A Section can be toggled on and off by double-left-clicking its handle, or by highlighting the Section plane and using 'Clip Display' from the Quad. A Section element has three grip edit points. Selecting the Section's handle turns the grips on for editing. You can move the start and end grips to change the orientation and width of the Section. You can move the midpoint grip to reposition the section plane. If you left-click the arrow displayed near the midpoint grip, you will flip the clipping direction of the Section.

5.3.13 How to create steel structural profiles in BricsCAD Shape

https://www.youtube-nocookie.com/embed/MwYysLRnuzQ BricsCAD Shape offers standard structural steel profiles to speed the creation of structural elements. Included are American, British and European standards. Use the Profile tool to create a beam or a column. Select the start point of the column. Move the cursor to the next point and left-click, or type in a value and press Enter. Continue to create elements, or press enter when you're done. You can change the profile using the Profile panel. Open the profile panel by left-clicking the "profile" icon. It is located on the vertical toolbar, on the right-hand edge of your screen. Select a standard and choose a profile. You can also search for specific profiles. To select a profile, hover over profile you want, then click and hold the left mouse button. Drag the profile into the beam or column and release the left mouse button. To connect elements, select them, raise the quad and use the Connect tool, found on the Model tab. You can toggle between different connection types by pressing the CTRL key. You can also create your own profiles from scratch. We'll draw a square, and offset it a given distance to create the new profile shape. To change the profile of an existing beam, select the beam and use "apply profile" from the quad. Select the new profile. The beam shape updates. Press enter to apply the change. Left-click to place the component, or use Dynamic Dimensions to precisely position it. You can enter a precise value for a distance, or press TAB to go to the next value. Press enter to accept the final position.

5.3.14 How to create a roof in BricsCAD Shape

https://www.youtube-nocookie.com/embed/YSHL0uXLyzU Need a gable style roof? It's easy with BricsCAD Shape. Using the Shape tool we will create a pitched roof slab. Align the drawing plane to the wall surface, by pressing SHIFT while hovering over the wall surface. You can change the alignment of the slab by pressing the CTRL key. To fix the angle, press TAB and enter a value. Press enter to accept. To change the slab profile thickness, press the TAB key. Let's repeat the process for the roof profile on the other side. To have the slabs join at the top of the roof, select both elements and select the CONNECT tool from the quad. Use the CTRL key to cycle through the available connection options. Using "BIM drag" you can extend the slabs to have the appropriate length. Now that the roof creation is completed, let's extend the walls to reach the roof. Highlight the top face of one wall, then from the quad, click the "select aligned faces" tool. Next, select "connect with nearest". To view the inside of the house, hide the roof slabs.

5.3.15 How to push or pull solids in BricsCAD Shape

https://www.youtube-nocookie.com/embed/7yWFzyk7BBU

Push/Pull is a basic modelling tool. It's a local editor for single solids. Push/Pull requires a positive distance and a direction. It won't let you create a void. Move in the positive direction, and push/pull lets us increase the width of this solid. Simplicity is power with push/pull editing in BricsCAD Shape.

5.4 FAQ

5.4.1 What are the Shape minimum system/hardware requirements?

1. Operating systems:

- Windows 10 or higher (64 bits)
- Windows 8 or higher (64 bits)
- Windows 7 (64 bits)
- Windows Vista with Service Pack 2 or later (all editions except Starter Edition)
- Windows Server 2008 R2
- Windows Server 2008 with Service Pack 2 or later
- Mac OSX 10.9 or higher
- Ubuntu 14.04 or higher

2. Processor: 1 GHz processor or faster, or compatible processor. 3. RAM: Next to the memory required by the operating system, BricsCAD requires 256 MB **(1 Gb or more recommended)**. 4. Hard Drive: 250 MB for program files + 1 GB free space. 5. Display:

- 1024 x 768 XGA with true color (minimum)
- Supported GPU (*): see below for more information on supported chipsets for GPU acceleration.

More Information

(*) BricsCAD's non-wireframe render modes are based on RedSDK technology from Redway3D®. GPU-acceleration for these render modes requires a supported GPU (graphic processing unit), either on a video card or on the motherboard.

1. On Windows, all graphic cards from NVIDIA, ATI and INTEL are supported by RedSDK
2. On Linux, 3D graphics hardware acceleration in RedSDK is supported on the following video chipsets:

- NVIDIA cards. It is required to use the latest recommended driver from NVIDIA.
- ATI cards. It is advised to use the latest recommended driver from ATI/AMD.

3D graphics hardware acceleration in RedSDK on Linux is NOT supported on:

- INTEL cards. As opposed to ATI and NVIDIA, 3D hardware acceleration in linux drivers for INTEL cards is relatively new. So far we do not yet officially support 3D hardware acceleration on INTEL cards, although we are working on it.
- Laptops with dual graphics, although it happens to work on some dual graphics systems using recent versions of proprietary NVIDIA drivers.

Chapter 6

Command Reference

6.1 Getting Started

6.1.1 Command Prefixes

The following prefixes can be used with the BricsCAD commands:

Prefix	Type	Result
' (single quote or apostrophe)	Transparency	Allows a command to be executed inside another command. E.g. If you are drawing a line, type **'zoom** at the command prompt to temporarily interrupt th
- (dash)	Command line	Executes a non-dialog version of a command. All command options are available in the co E.G. Type **-layer** at the command prompt to suppress the Drawing Explorer / Layers dialog. This is especially useful if you need to execute such commands in a script.
_ (underscore)	Global	Executes the original English command (global command) in translated version of BricsCA It is recommended to use the global command prefix in scripts, or when defining keyboard
. (period)	None redefined	Executes commands that have been temporarily removed from BricsCAD using the UNDEFINE (6.30.4) command.

Command prefixes can be combined. E.g. if the ZOOM command is temporarily undefined '**.zoom** executes the ZOOM command transparently.

6.1.2 Prompt Option Format

   User preference which controls how command options display in the command line prompts. **The command prompt format is:** Required input [FirstOption/SecondOption/ThirdOption] <what happens if you just press ENTER>: **e.g. the first prompt of the COPY command:** Enter base point [Displacement/mOde] <Displacement >:

Accessing the preference

command bar: **promptoptionformat** settings dialog: **Program Options / Display / Prompt Option Format** : **promptoptionformat Prompts you in the command bar:** New value for PromptOptionFormat <0>: (**Type 0, 1, 2, 3 or 4)**

Options

A command option has a keyword , a description and a shortcut . The shortcut is the keyword, without its lowercase characters. e.g. the third option of the CIRCLE (6.9.20) command: Keyword = TanTanRad Description = Tangent-Tangent-Radius Shortcut = TTR

Option	Description
0	Description only; adjust casing to mark shortcut: **Select center of circle or [2 Point/3 Point/Tangent-Tangent-Radius/turn Arc into circle/Multiple circles]:** This is the default prompt option format in the English version. In other versions the default prompt option format depends on local standards.
1	Show keyword only: **Select center of circle or [2Point/3Point/TanTanRad/Arc/Multiple]:**
2	Show description and keyword in brackets: **Select center of circle or [2 Point(2Point)/3 Point(3Point)/Tangent-Tangent-Radius(TanTanRad)/Turn arc into circle(Arc)/Multiple circles(Multiple)]:**
3	Show Description and shortcut in brackets: **Select center of circle or [2 Point(2P)/3 Point(3P)/Tangent-Tangent-Radius(TTR)/Turn arc into circle(A)/Multiple circles(M)]:**
4	Show local keyword and global keyword in brackets; this is relevant in localized versions only. **Select center of circle or [2Point/3Point/TanTanRad/Arc/Multiple]:**

NOTE	The PromptOptionTranslateKeywords (1.13.11) user preference controls whether translations of command option keywords are loaded or not. If disabled, the local keyword will be a copy of the global (English) keyword. As a result global shortcuts can be used without an underscore.

6.2 2

6.2.1 2dIntersection

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the apparent intersection entity snap, which snaps to the intersections of entities, even when they only appear to be intersecting in 3D space.

Figure 6.1

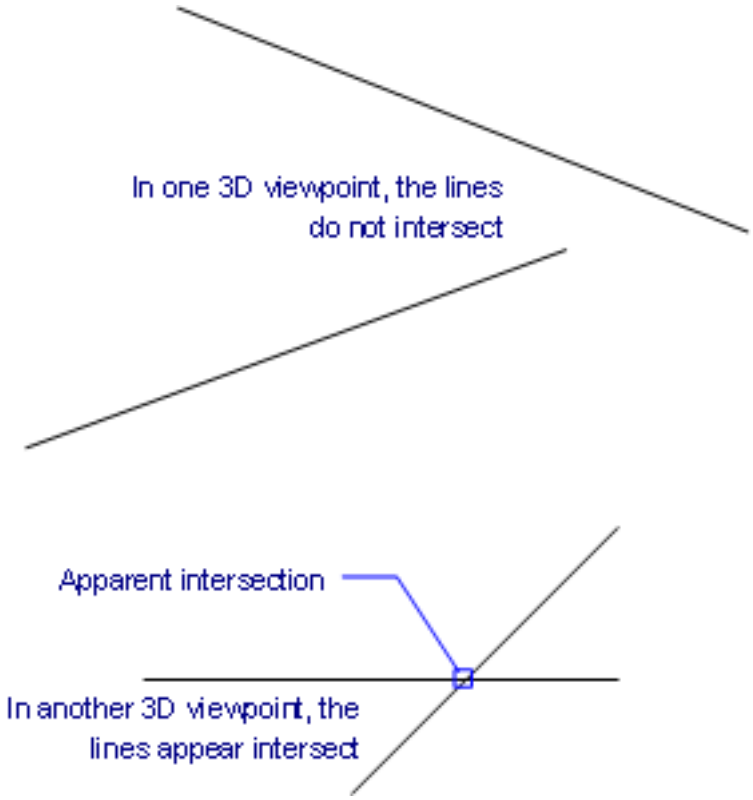


Accessing the Command



command bar: **2dintersection** transparent: **2dintersection** aliases: **apparent** **planviewint** toolbar: **Entity Snaps |**  ribbon:**Home | Entity Snaps | Snap to Apparent** : **2dintersection** Displays no prompt in the command bar. Each time you enter the command, it toggles the entity snap: on, then off, then on, and so on. Keep an eye on the Entity Snaps toolbar to see if this entity snap is on or off. This entity snap snaps to intersecting entities, including those that appear to intersect in 3D space: (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V19/Commands/2/2dintersection.png}) When 2dIntersection entity snap is on, the cursor snaps to the real and apparent intersections of entities.

Figure 6.2



Command Option

Option	Description
2dIntersection	Entering this command toggles apparent intersection entity snap on and off. This command can be entered transparently during commands, but must be entered without the transparency apostrophe prefix (').

Procedures

How to set Entity Snaps (1.15.15)

Related Commands

OSnap (6.23.12) - toggles entity snap modes through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command prompt. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode, like 2dIntersection. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - enters coordinates relative to a reference point. GCenter (6.15.4) - toggles geometric center entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Mtp (6.21.36) - specifies a point midway between two other points. Midpoint (6.21.16) - toggles midpoint snap mode. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

6.3 3

6.3.1 3D

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws basic 3D mesh entities, including boxes, cones, cylinders, dishes, domes, meshes, pyramids, spheres, tori, and wedges(short for ”three dimensional”). **Note:** Separate commands are available for each type of entity, such as AI_Box for drawing mesh boxes.

Figure 6.3



Accessing the Command

command bar: **3d** : **3d Prompts you in the command bar:** [Wedge/Pyramid/Box/Cone/cyLinder/Sphere/DIsh/DOme/Torus/Me
(**Enter an option)**

Command Options

Option	Description
Wedge	Draws 3D wedges. See Ai_Wedge (6.5.16) command. (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/image29.jpg})
Pyramid	Draws 3D pyramids. See Ai_Pyramid command. (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V19/Commands/3/3dPyramid.png})
Box	Draws 3D boxes. See Ai_Box (6.5.3) command. (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/image8.png})
Cone	Draws 3D cones. See Ai_Cone command. (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/image11.png})
cyLinder	Draws 3D cylinders See Ai_Cylinder (6.5.5) command. (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/image5.jpg})
Sphere	Draws 3D spheres. See Ai_Sphere (6.5.13) command. (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/image25.jpg})
DIsh	Draws 3D dishes (bottom half of spheres). See Ai_Dish (6.5.7) command. (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/image8.jpg})
DOme	Draws 3D domes (top half of spheres). See Ai_Dome (6.5.8) command. (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/image10.jpg})
Torus	Draws 3D tori (donuts). See Ai_Torus (6.5.15) command.
Mesh	Draws 3D meshes; prompts you: (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/image708.png}) **First of four corners for polygon mesh** - **(pick a point, or enter x,y,z coordinates.) Second corner** - **(pick a point, or enter x,y,z coordinates.) Third corner** - **(pick a point, or enter x,y,z coordinates.) Last corner for mesh** - **(pick a point, or enter x,y,z coordinates.) Number of divisions along primary axis (between 2 and 256)** - **(enter a number.) Number of divisions along secondary axis (between 2 and 256)** - **(enter a number.)**

Figure 6.4

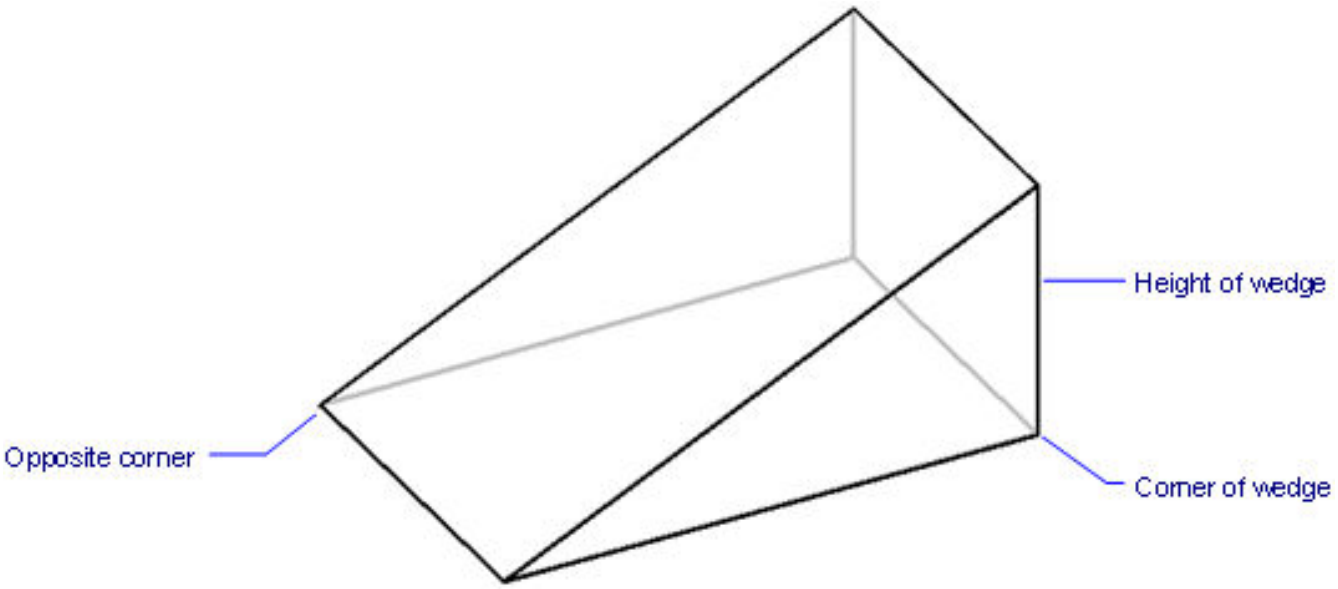
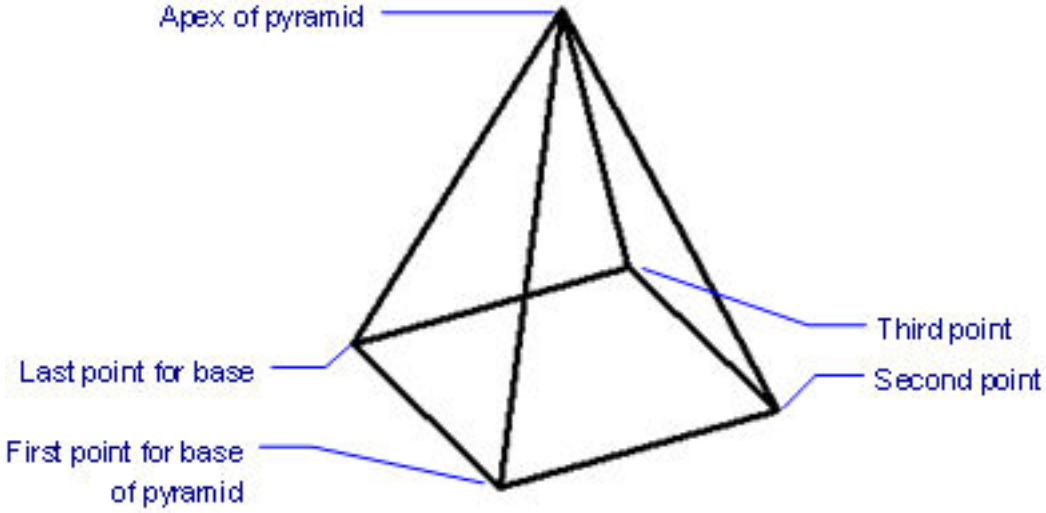


Figure 6.5



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. Ai_EdgeSurf (6.5.9) and EdgeSurf (6.13.2) - draws edge-defined surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.6

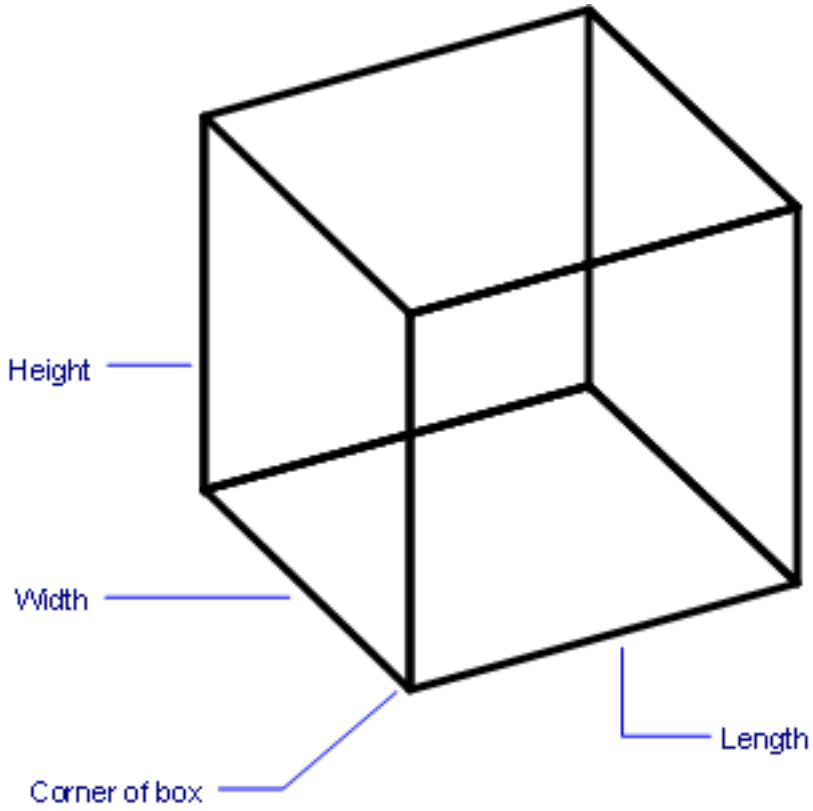


Figure 6.7

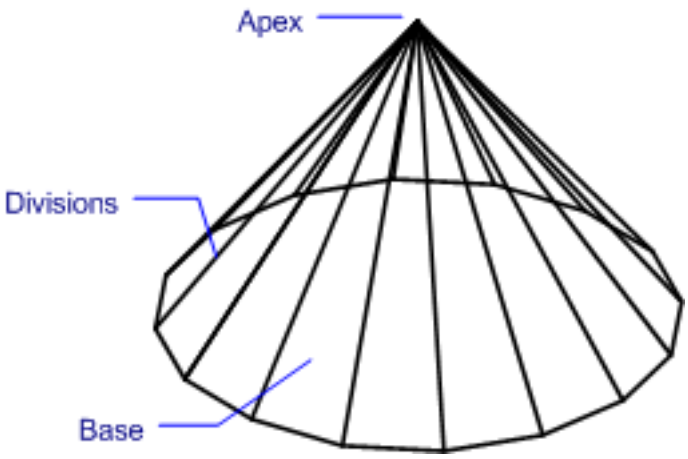


Figure 6.8

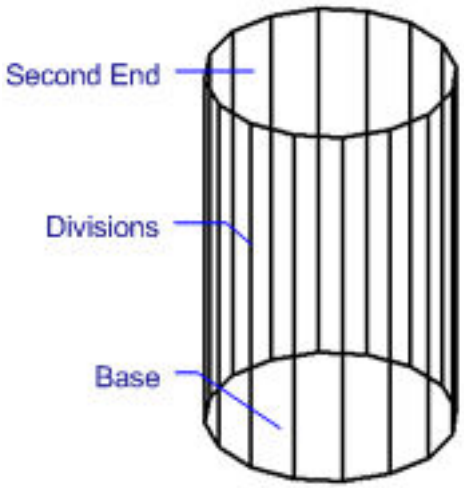


Figure 6.9

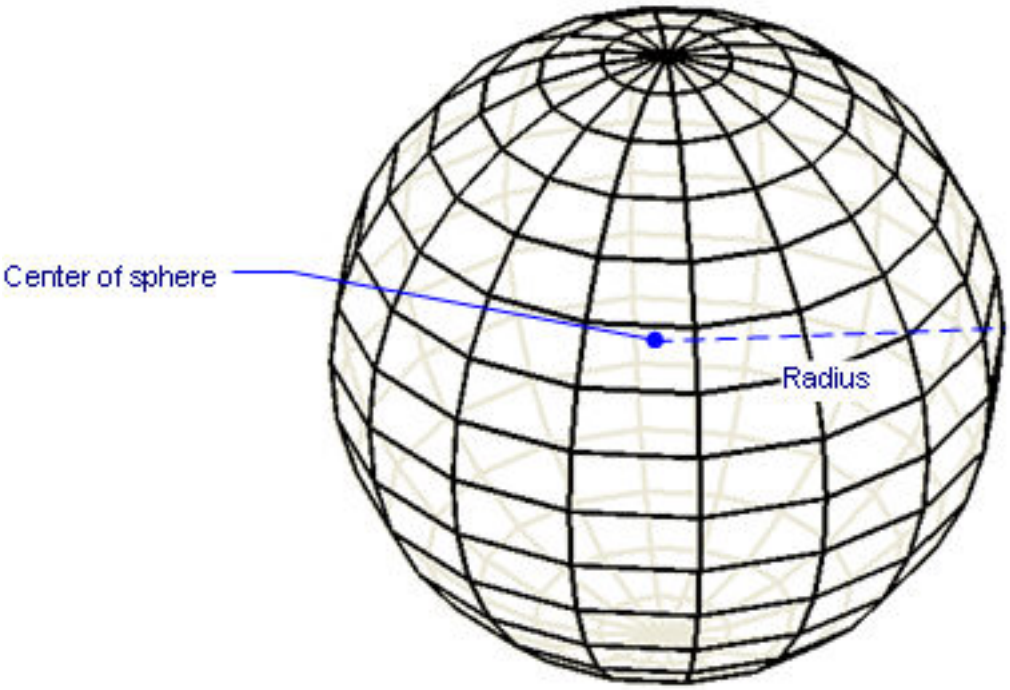


Figure 6.10

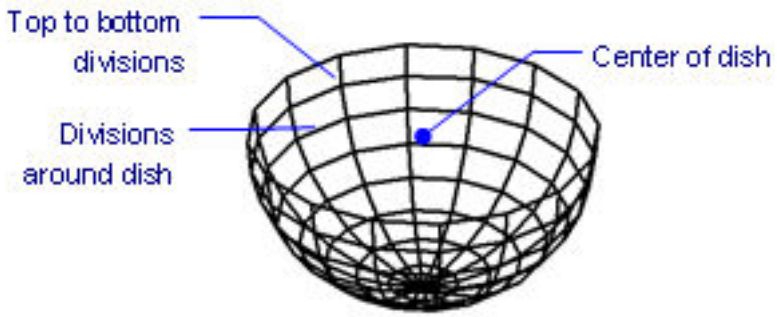


Figure 6.11

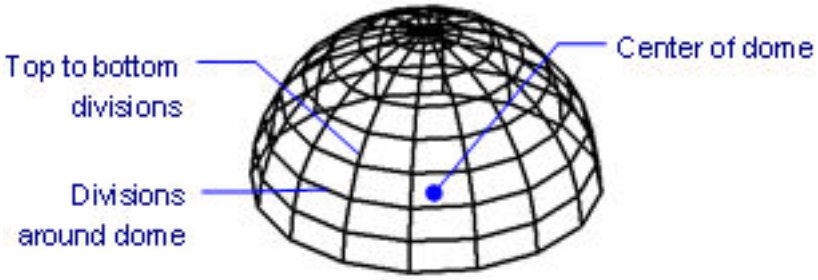


Figure 6.12

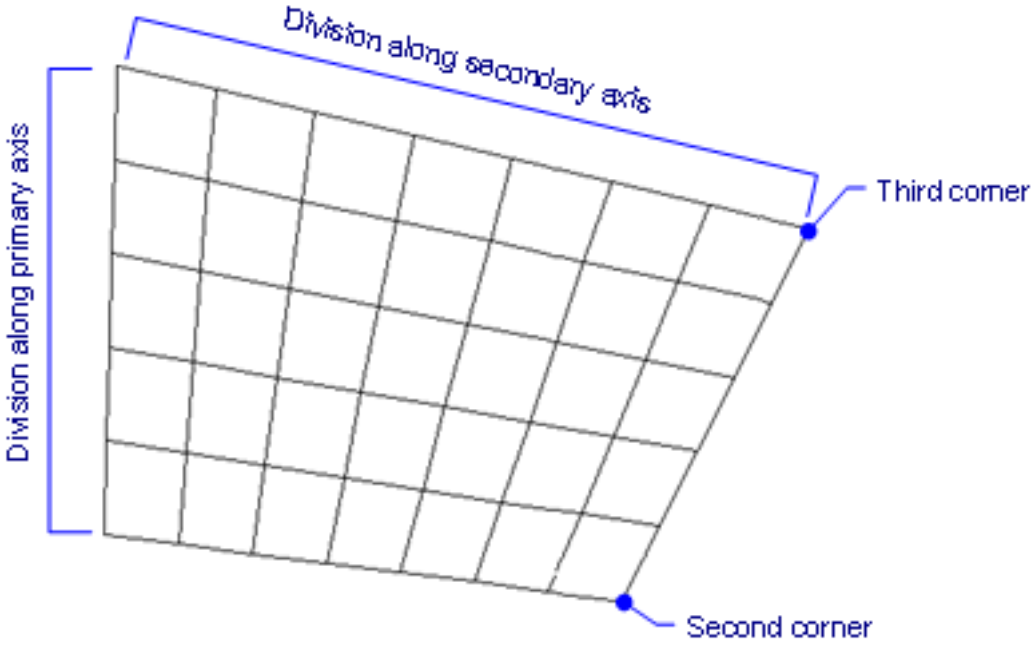

Division along secondary axis
Division along primary axis
Third corner
Second corner

Figure 6.13



6.3.2 3dArray

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Constructs static rectangular and polar arrays in rows, columns, and levels (short for "three dimensional array").

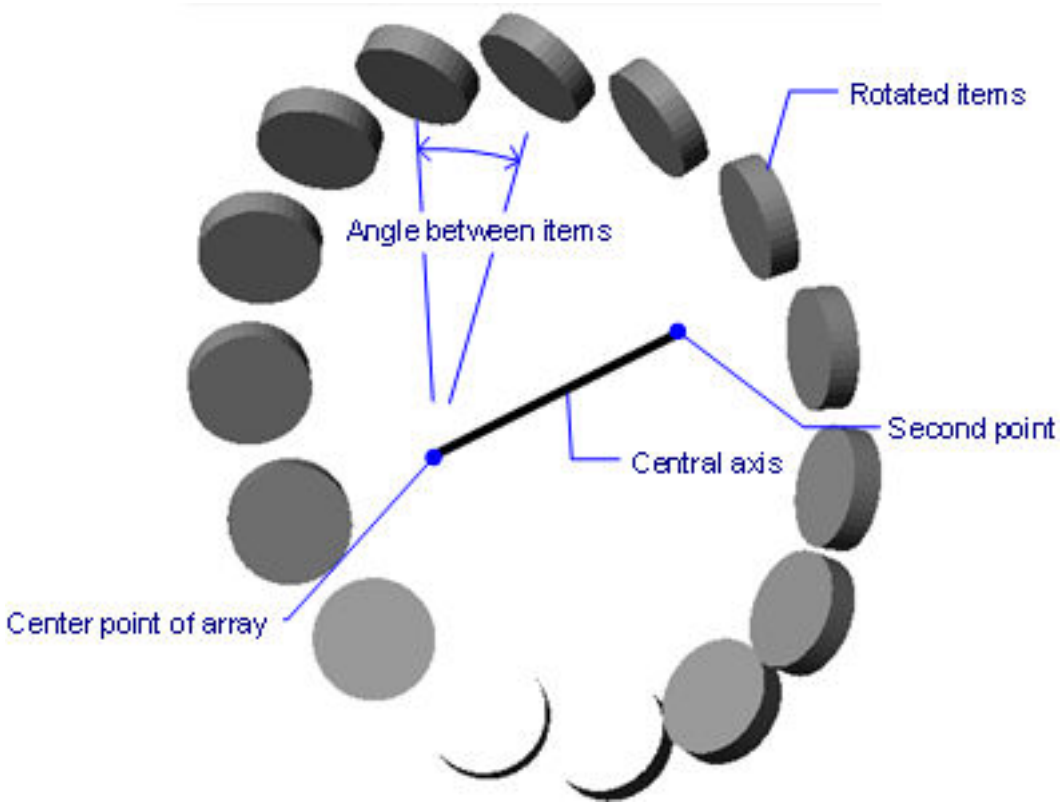
Accessing the Command


command bar: **3darray** aliases: **3a, array3d** menu bar: **Modify | 3D Array** toolbar: **Modify | Arrays |** 
:**3darray Prompts you in the command bar:** Select entities to array: **(Choose one or more entities)** Entities in set: 1 Select entities to array: **(Choose more entities, or press Enter to continue)** Type of array: Polar/<Rectangular>: **(Type P or R)** Prompts for creating 3D rectangular arrays: Number of rows in the array <1>: **(Enter a number, or press Enter to accept the default)** Number of columns <1>: **(Enter a number, or press Enter to accept the default)** Number of levels <1>: **(Enter a number, or press Enter to accept the default)** Vertical distance between rows: **(Enter a number, or press Enter to accept the default)** Horizontal distance between columns: **(Enter a number, or press Enter to accept the default)** Depth between levels: **(Enter a number, or press Enter to accept the default)**

- **Prompts for creating 3D polar arrays:**

(\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/image383.jpg}) ENTER to specify angle between items/<Number of items to array>: **(Enter a number, or press Enter)** Angle to array (+ for ccw, - for cw) <360>: **(Enter a number, or press Enter to accept the default)** Rotate entities around the array? No/<Yes>: **(Type Y or N)** Center point of array: (**Pick a point, or enter x,y,z coordinates)** Specify second point along central axis of array: (**Pick a point, or enter x,y,z coordinates)**

Figure 6.14



Command Options

Option	Description
Select entities to array	Specifies the entities to be arrayed. Use any entity selection method; press Enter when you are done.
Polar	Creates polar (circular) arrays; see Polar Array Options, below.
Rectangular	Creates linear, rectangular, or square arrays; see Rectangular Array Options, below.

Rectangular Array Options	Description
Number of rows in the array	Specifies the number of rows (x direction); for a linear array, enter 1. To draw the array downwards, enter a negative number.
Number of columns	Specifies the number of columns (y direction).To draw the array to the left, enter a negative number.
Number of levels	Specifies the number of levels (z direction).To draw the array downwards, enter a negative number.
Vertical distance between rows	Specifies the distance between rows of entities.
Horizontal spacing between columns	Specifies the distance between columns of entities.
Depth between levels	Specifies the distance between levels of entities.

Polar Array Options	Description
ENTER to specify angle between items	Specifies the angle between each copy.
Number of items to array	Specifies the number of items in the array; enter a number larger than 1.
Angle to array	Determines the extent and direction of the array. • Enter 360 for a full circle of copies. • Enter a smaller number to draw a partial polar array.

Related Commands

ArrayClassic (6.4.17) - creates a static polar or rectangular array of entities through a dialog box. -Array (6.4.15) - creates a static polar or rectangular array of entities; displays prompts at the command bar. Array (6.4.16) - associatively distributes entity copies in an evenly spaced rectangular, polar or path array. ArrayPath (6.4.21) - associatively distributes entity copies evenly along a path into multiple rows, and levels. ArrayPolar (6.4.23) - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. ArrayRect (6.4.25) - associatively distributes entity copies into any number of rows, columns, and levels. ArrayEdit (6.4.20) - edits associative array entities and their source entities. MInsert (6.21.17) - inserts arrays of blocks.

Figure 6.15



6.3.3 3dCompare

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) * Command only available on the Windows platform Performs a geometric comparison of solids and surfaces betweentwo drawing files. The result is shown in a new drawing and reported in the Mechanical Browser (3.2.2) panel.

Accessing the Command

 command bar: **3dcompare** toolbar: **3D Compare|Compare 3D** ribbon: **tools| Compare 3D|Compare 3D** :**3dcompare** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18
Do the following:

1. Click the **Browse** button (...) of the **Model 1** and **Model 2** fields and select the two DWG drawing files you want to compare.
2. Set the options.
3. Click the **OK** button.

The selected files are attached to a new drawing that is named **Compare_<File_1>_<File_2>.dwg**.In this drawing, a layout named ”Comparison” is created automatically with two viewports, one for each drawing, illustrating the differences between the drawings using colors. (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/3dCompareResult.png}) At the same time, the Mechanical Browser panel opens automatically, and lists the differences between the two drawings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

Figure 6.16

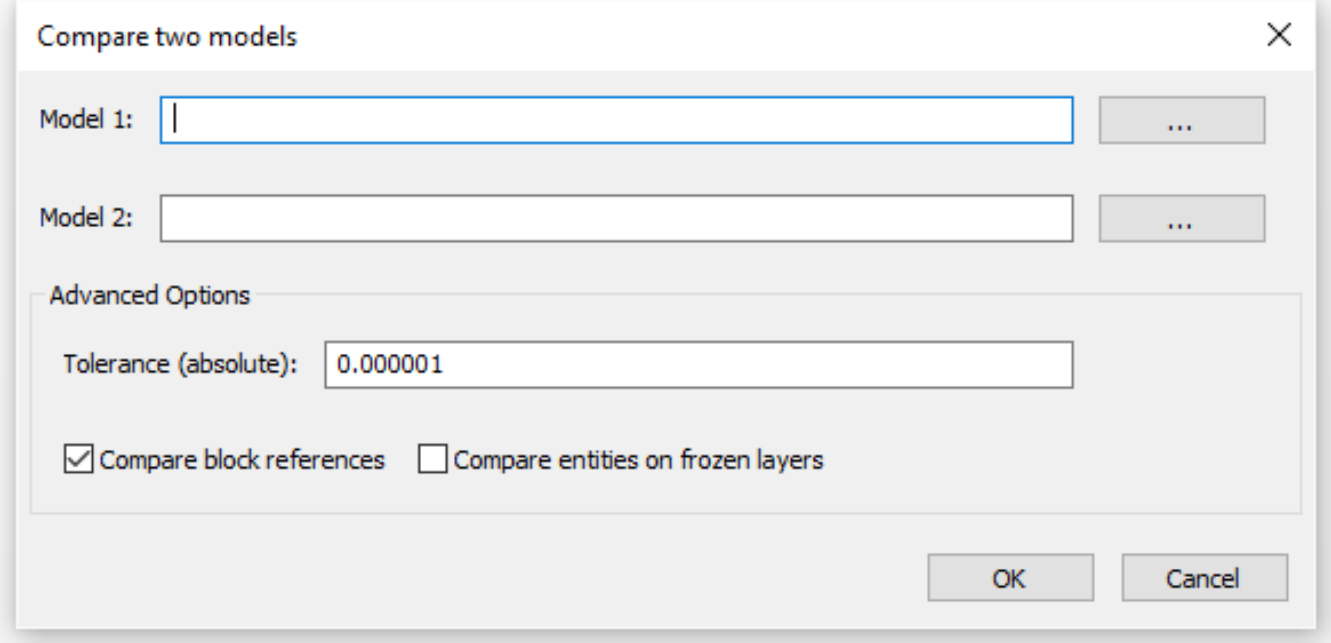


Figure 6.17

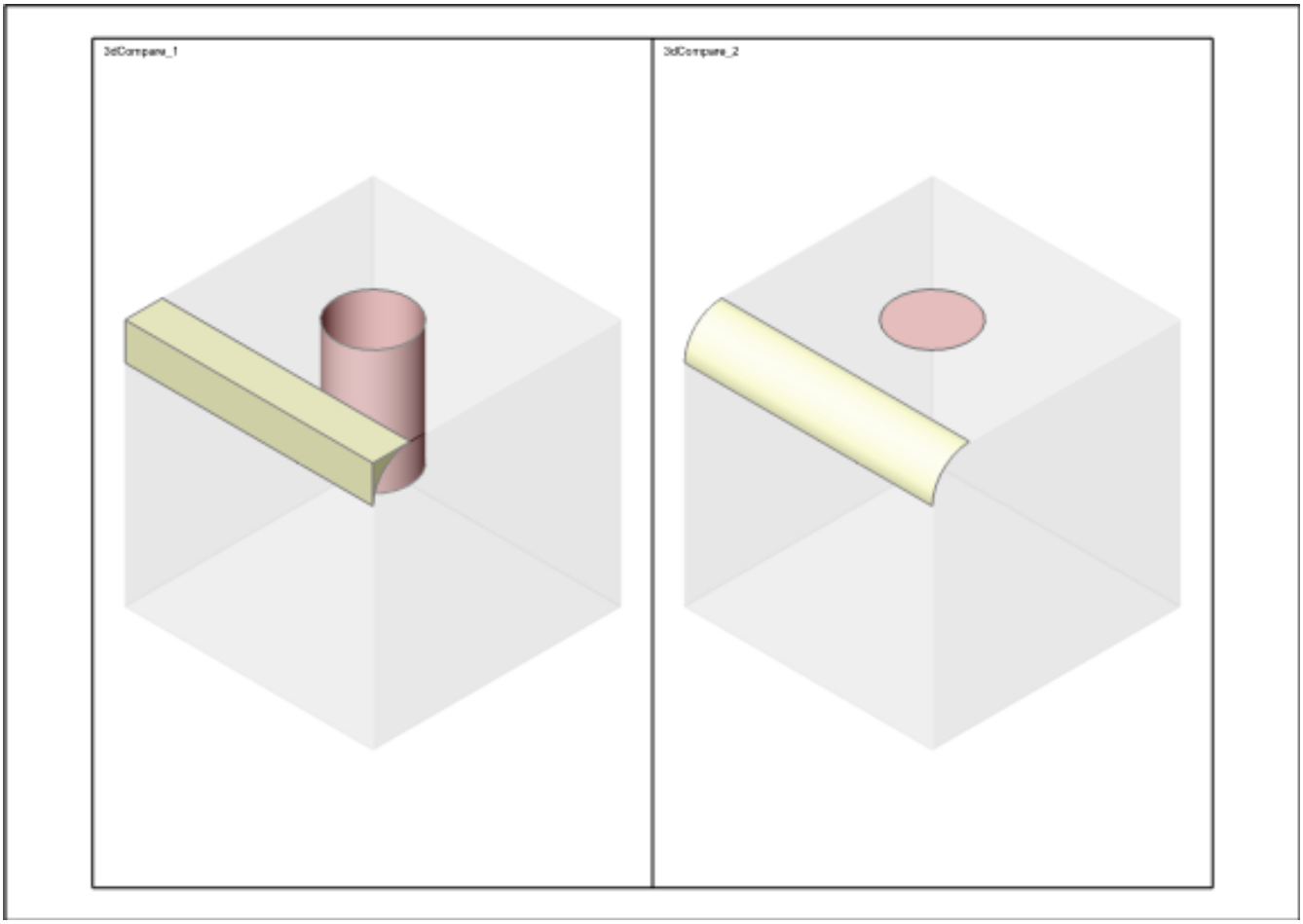
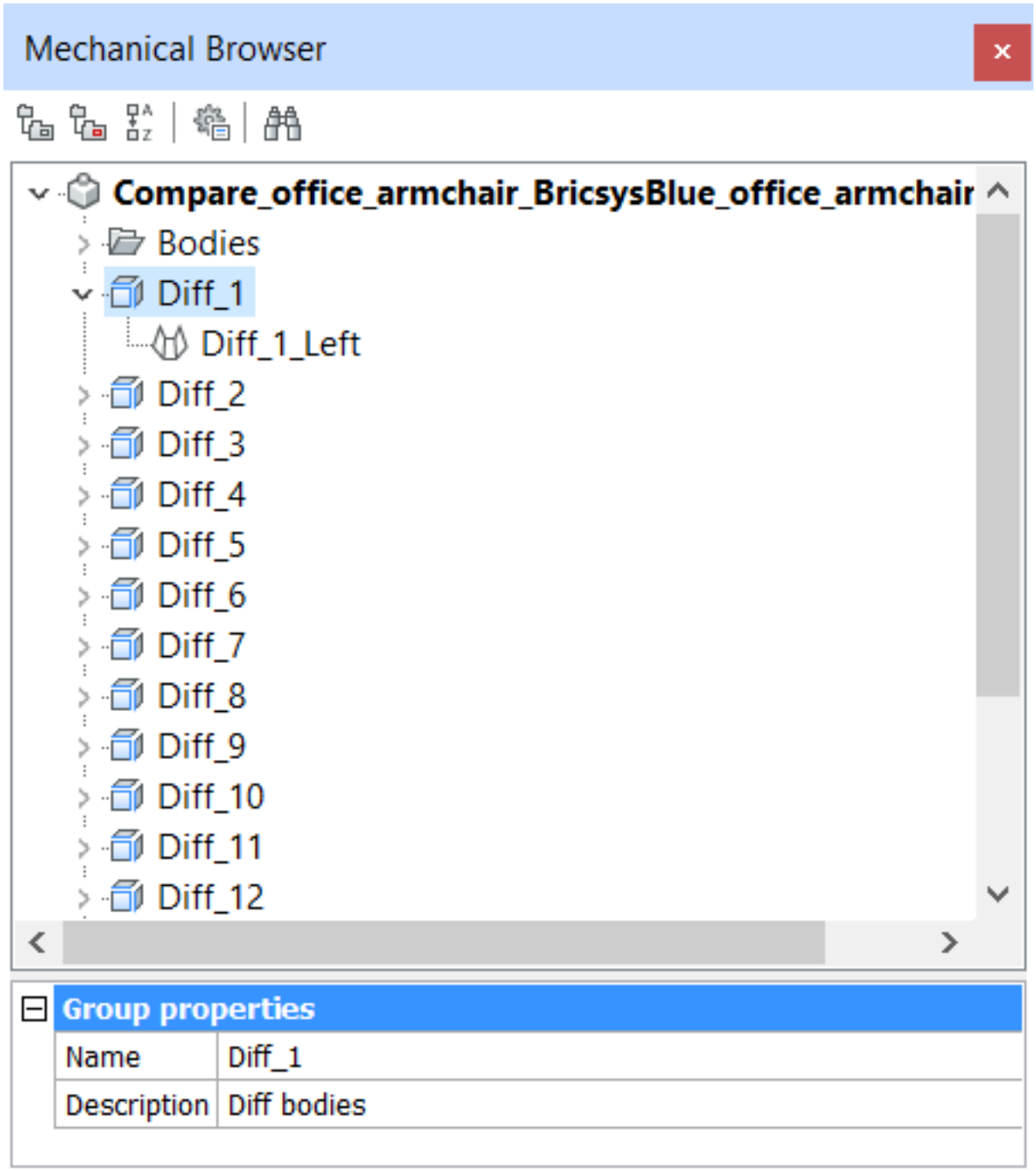


Figure 6.18


Mechanical Browser
Compare_office_armchair_BricsysBlue_office_armchair
Bodies
Diff_1
Diff_1_Left
Diff_2
Diff_3
Diff_4
Diff_5
Diff_6
Diff_7
Diff_8
Diff_9
Diff_10
Diff_11
Diff_12
Group properties
Name
Diff_1
Description
Diff bodies

Compare Two Models dialog box

Option	Description
Model 1	Specifies the file name of the first drawing to compare.
Model 2	Specifies the file name of the second drawing to compare.
Tolerance	Threshold value for geometry comparison: determines how close two models must be to be considered unchanged.
Compare block references	Determines if the content of blocks should be compared: • **Yes** - compares solids and surfaces in blocks • **No** -ignores solids and surfaces in blocks This command works with regular blocks and inserts of arbitrary depth, which contain solids or surfaces, including assemblies that use block structures.
Compare entities on frozen layers	Determines if entities on frozen layers should be compared: • **Yes** - solids and surfaces on frozen layers are compared • **No** -solids and surfaces on frozen layers are ignored

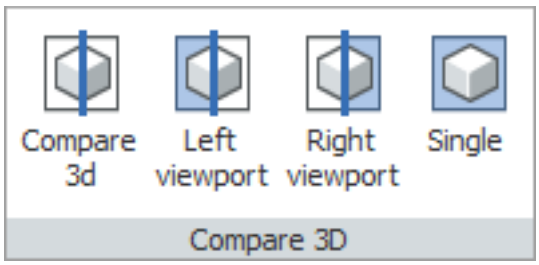
The value of the 3DCOMPAREMODE (1.13.11) system variable controls the display in the viewports of the Comparison layout; in the Settings dialog box, search for "3dcomparemode."

Compare 3D buttons

The buttons in the 3D Compare toolbar orCompare 3D ribbon panel (of the Tools tab) switch between view modes. (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V18/CmdRef/source/3dCompareToo

Icon	Name	Description
	Compare3D	Displays the Compare Two Models dialog box; see the description below.
	Left Viewport	Shows the differences between the two drawings in the left viewport of the Comparison layout.

Figure 6.19



	Right Viewport	Shows the differences between the two drawings in the right viewport of the Comparison layout.
	Single	Shows the differences in model space.When switching to this option, the layout viewport displays options are disabled.

Related Commands

MechanicalBrowserOpen (6.8.29)- displays the Mechanical Browser. DwgCompare (6.10.41) - compares a selected drawing with the current drawing during a comparison session.

Figure 6.20



6.3.4 3dConvert

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Converts 3D solids to polyface meshes.

Accessing the Command

command bar: **3dconvert** : **3dconvert Prompts you in the command bar:** Select ACIS entities to convert: **(Choose one or more 3D solids)** Select ACIS entities to convert: **(Choose more 3D solids, or press Enter to continue)** The 3D model looks no different following conversion.

Command Options

Option	Description
Select ACIS entities to convert	Chooses the 3D solids to be converted to polyface meshes.

Related Commands

Box (6.6.17) - creates 3D boxes as solid models. Cone (6.9.63) - creates 3D cones as solid models. Cylinder (6.9.60) - creates 3D cylinders as solid models. Dish (6.10.30) - creates the lower half of 3D balls as solid models. Dome (6.10.34) - creates the upper half of 3D balls as solid models. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. Revolve (6.26.30) - revolves 2D entities into 3D solid models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Sphere (6.27.64) - creates 3D balls as solid models. Subtract (6.27.56) - removes one solid model from a second. Torus (6.29.22) - creates 3D donuts as solid models. Union (6.30.6) - joins two or more solid models into a single entity. Wedge (6.33.7) - creates 3D wedges as solid models.

Figure 6.21



6.3.5 3dFace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws 3D 4-edged faces with optional invisible edges.

Accessing the Command


command bar: **3dface** aliases: **3f, face** menu bar: **Model | Meshes | 3D Face** toolbar: **Meshes | 3D Faces |**


ribbon:**Model | Meshes | 3D Face** : **3dface Prompts you in the command bar:** First point of 3D face [Invisible]: **(Pick a point, or type I)** Second point [Invisible]: **(Pick a point, or type I)** Third point of 3D face [Invisible]: **(Pick a point, or type I)** Last point [Invisible]: **(Pick a point, or type I)** Third point of 3D face [Invisible]: **(Pick a point, or type I, or press Enter to stop)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image384.jpg This command repeats the **Invisible edge/<Third point of 3D face [Select Invisible edge(Invisible)]>**prompt so that adjoining faces can be drawn: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image

Figure 6.22

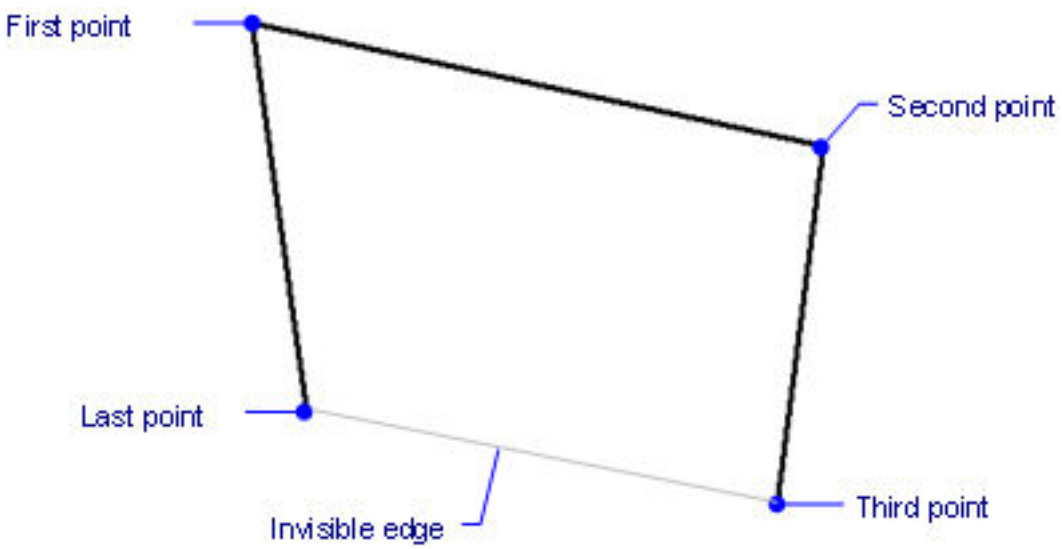
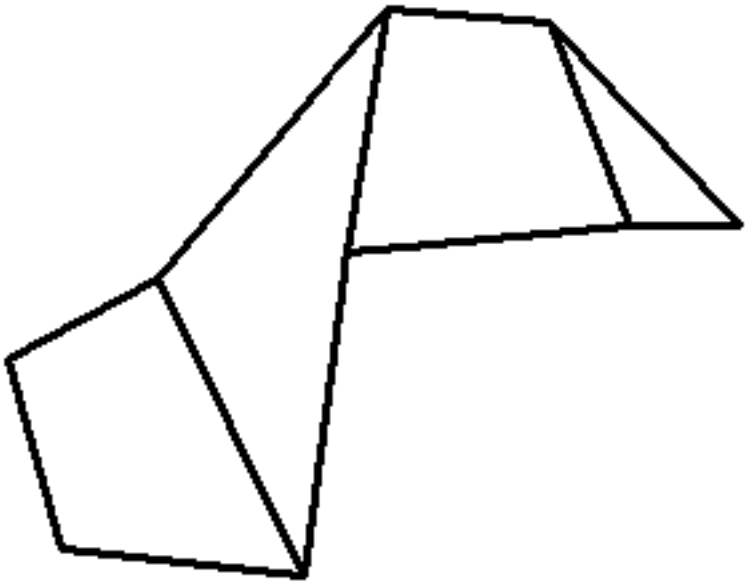


Figure 6.23



Command Options

Option	Description
First point of 3D face	Specifies the location of the first corner of the face.Pick a point, or enter x,y,z coordinates.
Second point	Specifies the location of the second corner of the face.Pick a point, or enter x,y,z coordinates.
Third point of 3D face	Specifies the location of the third corner of the face.Pick a point, or enter x,y,z coordinates.
Last point	Specifies the location of the fourth corner of the face.Pick a point, or enter x,y,z coordinates.
Invisible edge	Makes the next edge invisible; prompts you: **Third point of 3D face** - **(specify a point for the next vertex).**

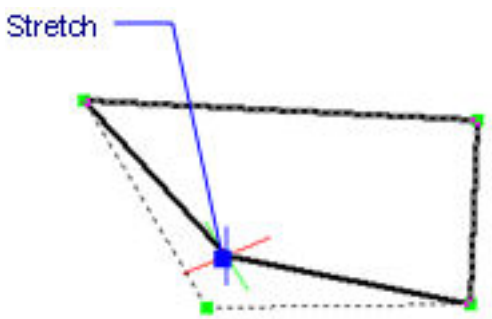
Grips Editing

3D faces can be edited directly through grips:

1. Select the 3D face. Notice that it has several grips.
2. Drag a grip to stretch the face.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image450.jpg})

Figure 6.24



Related Commands

3dMesh (6.3.7) - draws 3D surface meshes of size m x n. PEdit (6.24.20) - edits 3D meshes. PFace (6.24.23) - draws 3D multi-sided meshes.

Figure 6.25



6.3.6 3dIntersection

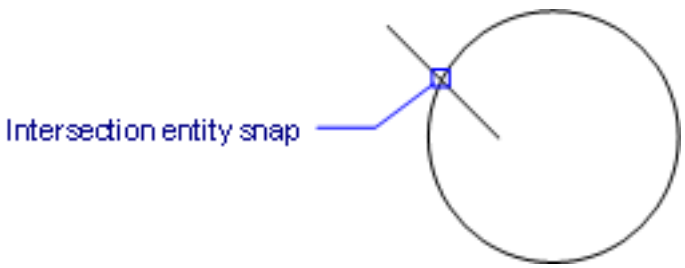
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles Intersection entity snap, which snaps to the intersection of two entities.

Accessing the Command



command bar: **3dintersection** transparent: **3dintersection** alias: intersection (6.17.20)toolbar: **Entity Snaps |** ribbon:**Home | Entity Snaps | Snap to Intersection** : **3dintersection** Displays no prompts in the command bar. Each time you enter the command, it toggles the entity snap, either on or off. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image227.png}) This entity snap snaps to entities that physically intersect each other; to snap to apparent intersections, use 2dintersection entity snap.

Figure 6.26



Command Options

Option	Description
3dIntersection	Entering this command toggles Intersection entity snap mode.This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Procedure

How to set Entity snaps (1.15.15)

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.27



6.3.7 3dMesh

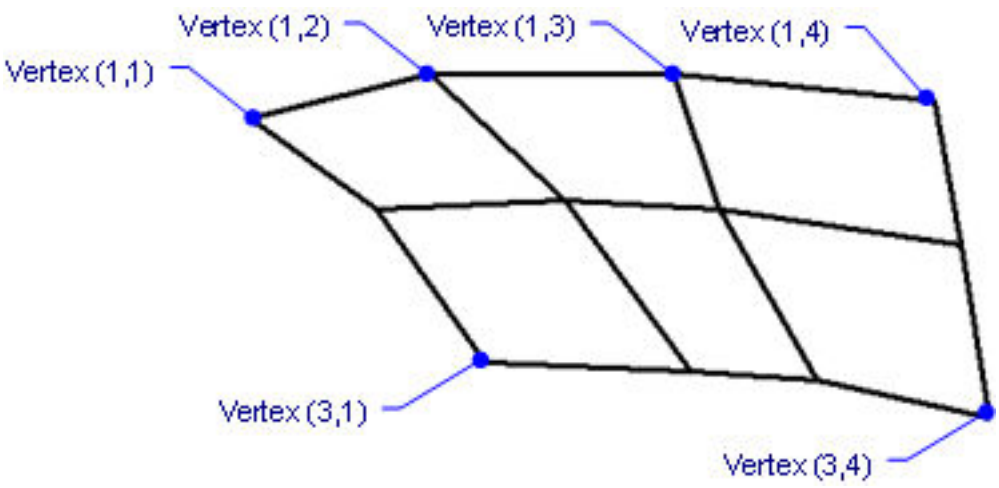
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws 3D surface meshes.

Accessing the Command



command bar: **3dmesh** alias: **mesh** menu bar: **Model | Meshes | Polygon Mesh** toolbar: **Meshes | 3D Face** |  ribbon:**Model | Meshes | Polygon Mesh** : **3dmesh Prompts you in the command bar:** Number of vertices in M direction (between 2 and 256): **(Enter a number)** Number of vertices in N direction (between 2 and 256): **(Enter a number)** Position of vertex (0, 0): **(Pick a point, or enter x,y,z coordinates) Command continues prompting you for the coordinates of vertices** Vertex (3, 4): (Pick another point, or enter x,y,z coordinates.) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image385.jpg})

Figure 6.28



Command Options

Option	Description
Number of vertices in M direction	Specifies the number of vertices in the primary direction.Enter a number between 2 and 256.A single face is made from a 2x2 vertex. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Number of vertices in N direction	Specifies the number of vertices in the secondary direction.Enter a number between 2 and 256.
Position of vertex (n, n)	Specifies the position of the vertex.Pick a point, or enter x,y,z coordinates.

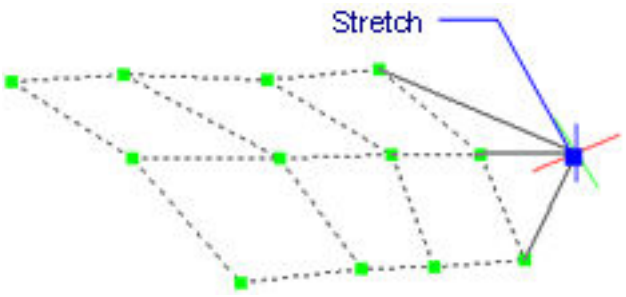
Grips Editing

3D meshes can be edited directly through grips:

1. Select the 3D mesh . Notice that it has several grips.
2. Drag a grip to stretch the mesh.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image452.jpg})

Figure 6.29



Related Commands

3dFace (6.3.5) - draws 3D 4-edged faces with optional invisible edges. PEdit (6.24.20) - edits 3D meshes. PFace (6.24.23) - draws 3D multi-sided meshes.

Figure 6.30

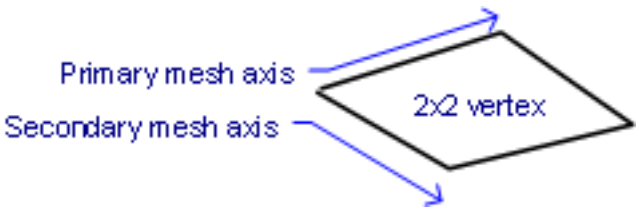


Figure 6.31



6.3.8 3dOsnap

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the entity snap modes for 3D entities through the Settings dialog box (short for ”3D object snap”).

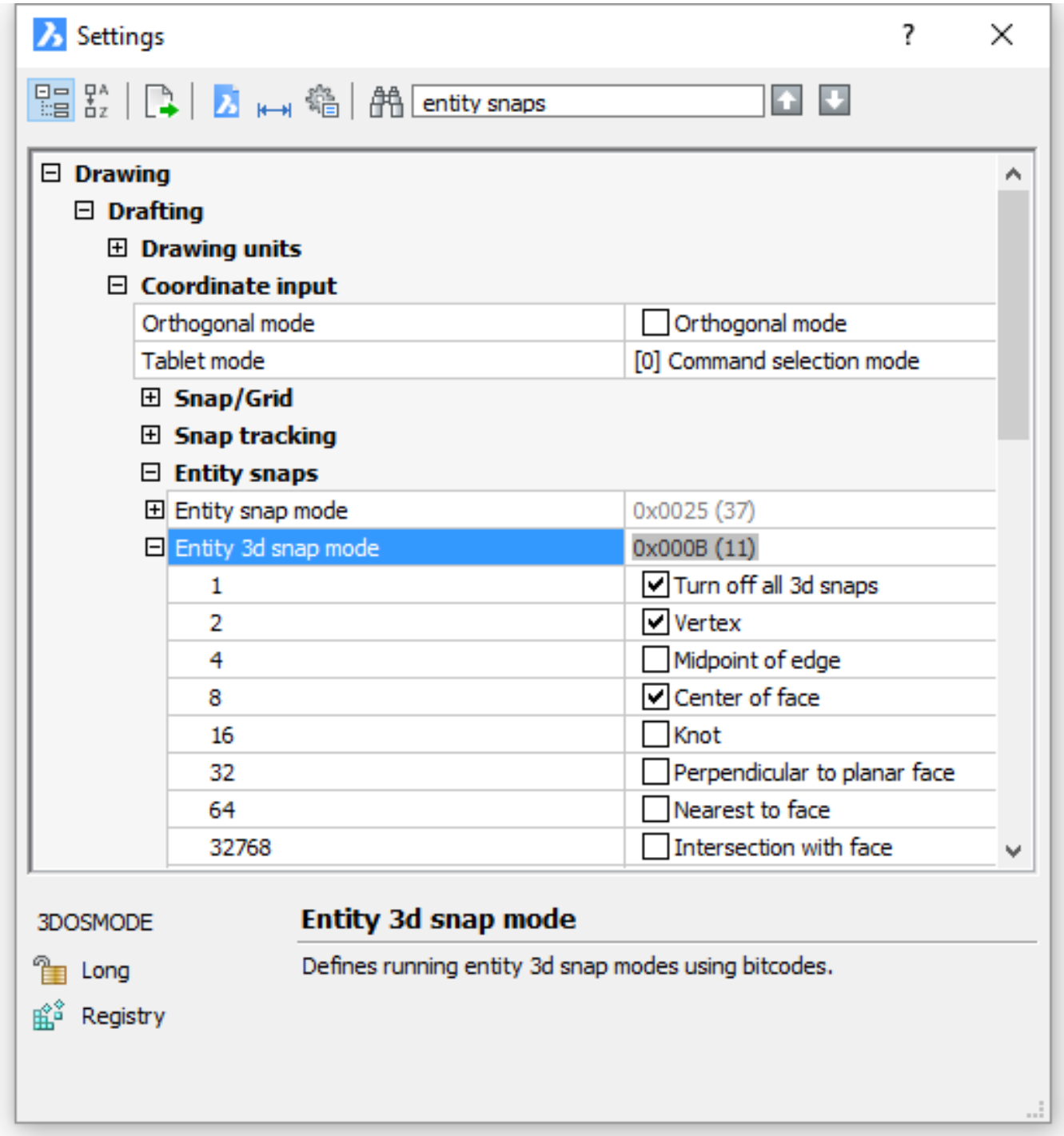
Accessing the Command

command bar: **3dosnap** transparent: **'3dosnap** toolbar: **3D Entity Snaps | (\vref {fig:C:/Dev/Task-2448/images/storagegoogle static/help/enUS/V19/Common/osnapbuttons.png})** : **3dOsnap** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3dOsnapDialog.png})

Figure 6.32



Figure 6.33


Settings
entity snaps
Drawing
Drafting
Drawing units
Coordinate input
Orthogonal mode
Orthogonal mode
Tablet mode
[0] Command selection mode
Snap/Grid
Snap tracking
Entity snaps
Entity snap mode
0x0025 (37)
Entity 3d snap mode
0x000B (11)
1
Turn off all 3d snaps
2
Vertex
4
Midpoint of edge
8
Center of face
16
Knot
32
Perpendicular to planar face
64
Nearest to face
32768
Intersection with face
3DOSMODE
Long
Registry
Entity 3d snap mode
Defines running entity 3d snap modes using bitcodes.

Dialog Box Options

Option	Description
Turn off all 3D snaps	Toggles the currently selected 3D snap modes between on and off; equivalent to the ZNone (6.35.6) command.
Vertex	Snaps to the closest vertex of a 3D entity, equivalent to the ZVertex (6.35.9) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3
Midpoint of edge	Snaps to the midpoint of a face edge, equivalent to the ZMidpoint (6.35.4) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3
Center of face	Snaps to the center of planar or curved 3D faces; equivalent to the ZCenter (6.35.1) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3
Knot	Snaps to a knot on a spline, equivalent to the ZKnot command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3
Perpendicular to planar face	Snaps to a point perpendicular to a face; equivalent to the ZPerpendicular (6.35.8) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3
Nearest to face	Snaps to a point on the face of a 3D entity that is nearest to the cursor, equivalent to the ZNearest (6.35.5) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3
3D Intersection	Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces; equivalent to the ZIntersection (6.35.2) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3

Related Commands

-3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. OSnap (6.23.12) - sets entity snaps through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command line . Aperture (6.4.10) - sizes the aperture cursor for entity snaps. Zcenter (6.35.1) - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces. Zintersection (6.35.2) - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces. Zknot (6.35.3) - toggles the 3D knot entity snap. Snaps to a knot on a **spline**. Zmidpoint (6.35.4) - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge. Znearest (6.35.5) - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor. Znone (6.35.6) - disables all 3d snap modes. Zperpendicular (6.35.8) - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face. Zvertex (6.35.9) - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Figure 6.34

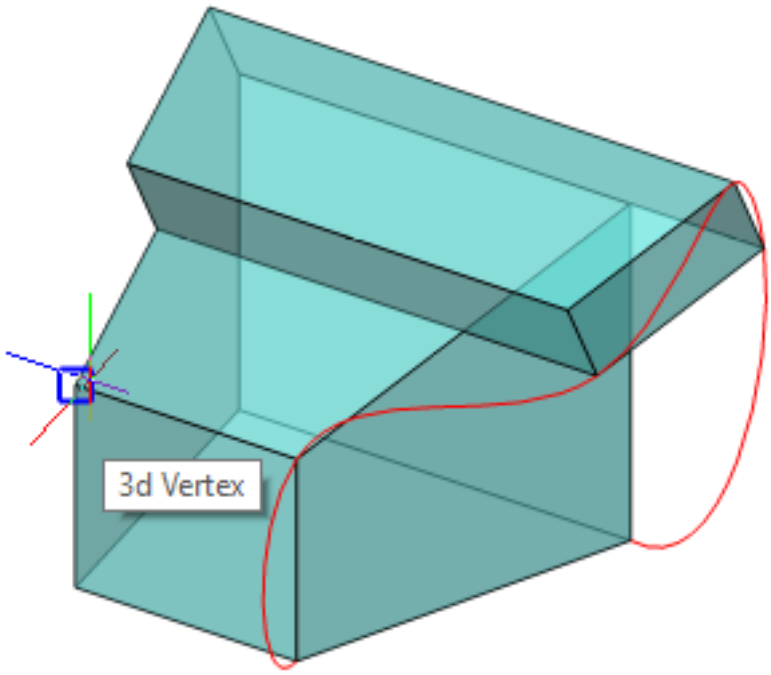


Figure 6.35

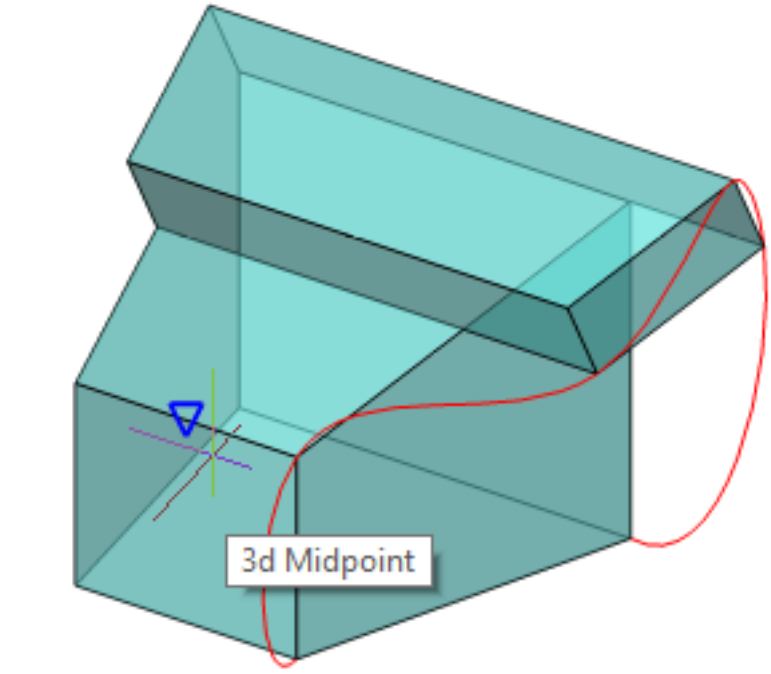


Figure 6.36

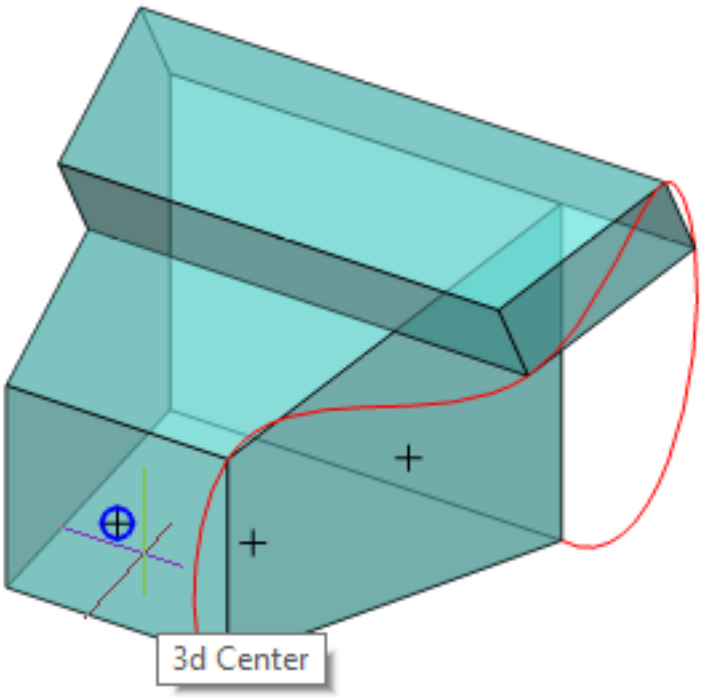


Figure 6.37

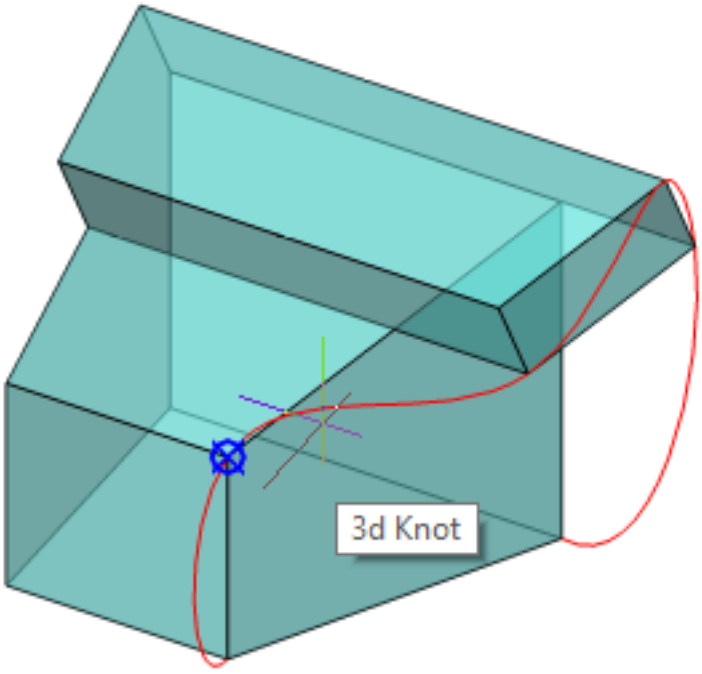


Figure 6.38

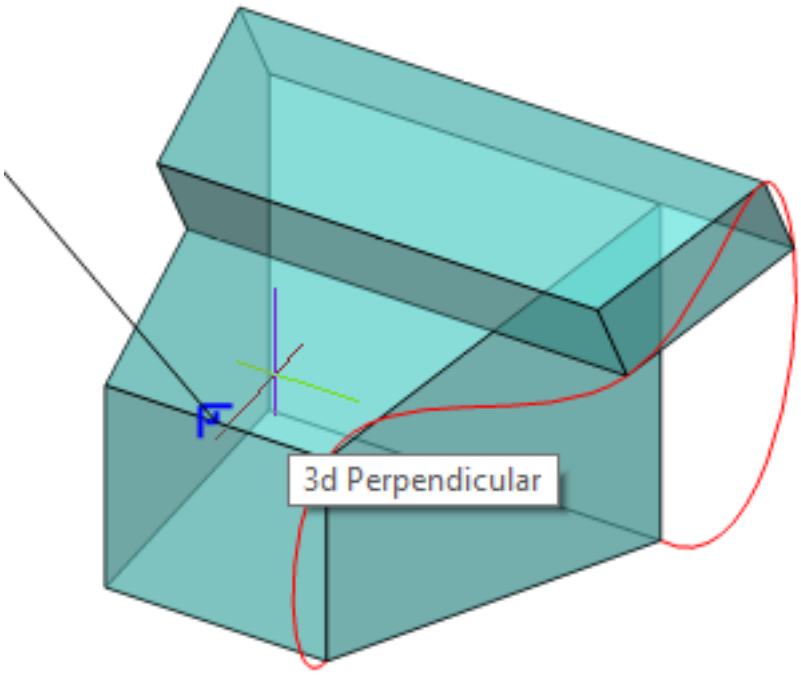


Figure 6.39

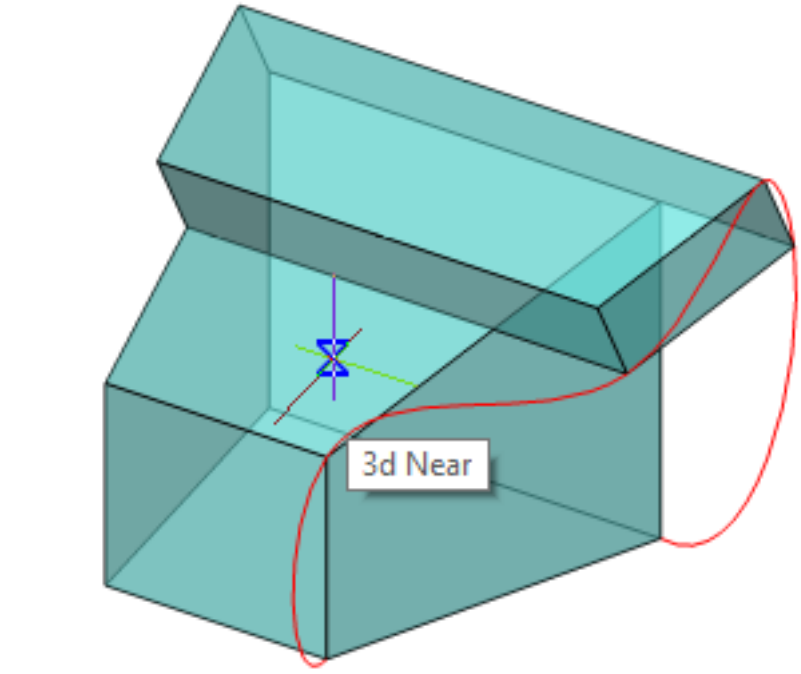


Figure 6.40

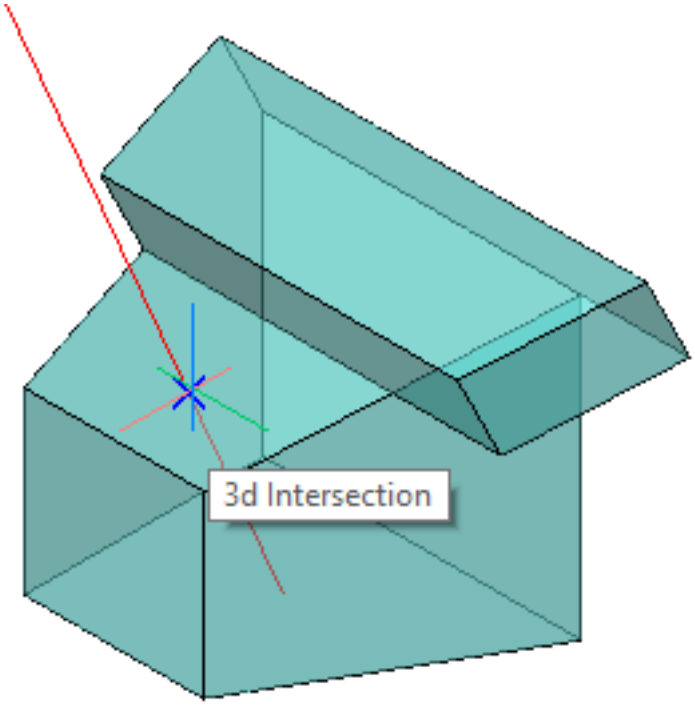


Figure 6.41



6.3.9 -3dOsnap

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles entity snap modes for 3D entities at the command line (short for "3D object snap").

Accessing the Command

command bar: **-3dosnap** transparent: **'-3dosnap** : **-3dosnap Prompts you in the command bar:** 3d Snaps: ZNEArest/ZVERtex/ZMIDpoint/ZCENter/ZPERpendicular/ZKNOt/ZINTersection/ON/<OFF>: **(Enter an option.)** To set more then one snap mode at the same time, separate their names with commas; you need only type the uppercase portion of each snap name. ZNEArest/ZVERtex/ZMIDpoint/ZCENter/ZPERpendicular/ZKNOt/ZINTersection/ON/<OFF>:**(znea**

Command Options

Option	Description
ZNEArest	Snaps to a point on the face of a 3D entity that is nearest to the cursor,equivalent to the ZNearest (6.35.5) command.
ZVERtex	Snaps to the closest vertex of a 3D entity, equivalent to the ZVertex (6.35.9) command.
ZMIDpoint	Snaps to the midpoint of a face's edge, equivalent to the ZMidpoint (6.35.4) command.
ZCENter	Snaps to the center of a flat or curved 3D face, equivalent to the ZCenter (6.35.1) command.
ZPERpendicular	Snaps to a point perpendicular to a face,equivalent to the ZPerpendicular (6.35.8) command.
ZKNOt	Snaps to a knot on a spline, equivalent to the ZKnot (6.35.3) command.
ZINTersection	Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces,equivalent to the ZIntersection (6.35.2) command.
ON	Enables all currently set 3D snap modes,equivalent to the ZNone (6.35.6) command.
OFF	Disables all currently set 3D snap modes,equivalent to the ZNone (6.35.6) command.

Related Commands

3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. OSnap (6.23.12) - sets entity snaps through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command line. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. Zcenter (6.35.1) - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces. Zintersection (6.35.2) - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces. Zknot (6.35.3) - toggles the 3D knot entity snap. Snaps to a knot on a **spline**. Zmidpoint (6.35.4) - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge. Znearest (6.35.5) - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor. Znone (6.35.6) - disables all 3d snap modes. Zperpendicular (6.35.8) - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face. Zvertex (6.35.9) - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Figure 6.42



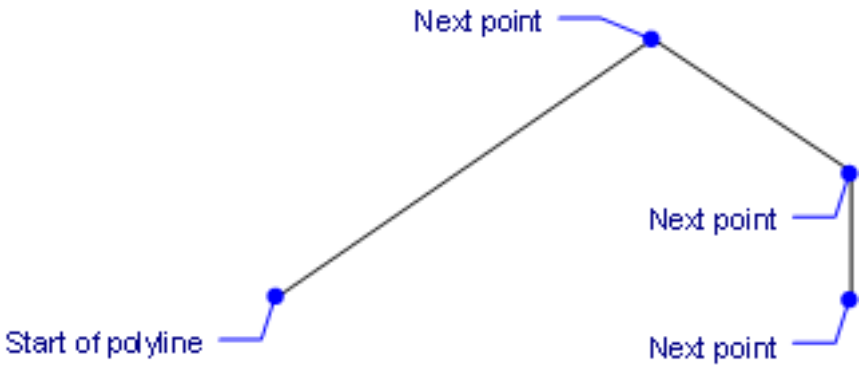
6.3.10 3dPoly

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws 3D polylines (short for "three dimensional polylines").

Accessing the Command

 command bar: **3dpoly** alias: **3p** menu bar: **Draw | 3D Polyline** toolbar: **Draw |Polylines |**  ribbon:**Draw | Draw | Polylines | 3D Polyline** : **3dpoly Prompts you in the command bar:** Select start of polyline or [Follow]<ENTER to use last point>: **(Pick a point, or enter x,y,z coordinates, or press Enter to select the last point entered)** Next point: **(Pick another point, or enter x,y,z coordinates)** Next point or [Undo]: **(Pick a point, or choose an option)** Set next point or [Close/Undo]: **(Pick a point, or choose an option)** Set next point or [Close/Undo]: **(Pick a point, or choose an option, or press Enter to end the command)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18 You edit 3D polylines with the PEdit (6.24.20) command or with grips.

Figure 6.43



Command Options

Option	Description
ENTER to use last point	Uses the last point as the start point of the 3D polyline.
Select start of polyline	Specifies the starting point of the polyline.Pick a point, or enter x,y,z coordinates.
Follow	Draws the first segment at the same angle as the previously drawn segment, whether a line or 3D or 2D polyline; move the mouse to indicate the length.
Next point	Specifies the next vertex of the polyline: • Pick a point • Enter x,y,z coordinates • Press Enter to exit the command
Undo	Removes the last-drawn segment.
Close	Closes the polyline and ends the command.

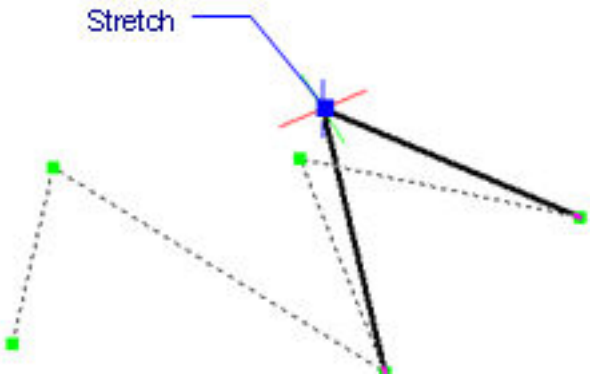
Grips Editing

3D polylines can be edited directly through grips:

1. Select the 3D polyline. Notice that it has a grip at the end of each segment.
2. Drag a grip to stretch the segment.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image458.jpg})

Figure 6.44



Related Commands

Helix (6.16.7) - creates a 2D spiral or a 3D helix. PLine (6.24.27) - draws 2D polylines. PEdit (6.24.20) - edits polylines. Donut (6.10.35) - draws fat circles out of polylines. Polygon (6.24.43) - draws regular polygons out of polylines. Rectang (6.26.6) - draws rectangles out of polylines.

6.4 A

6.4.1 About

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Opens the About dialog box, with version and copyright information

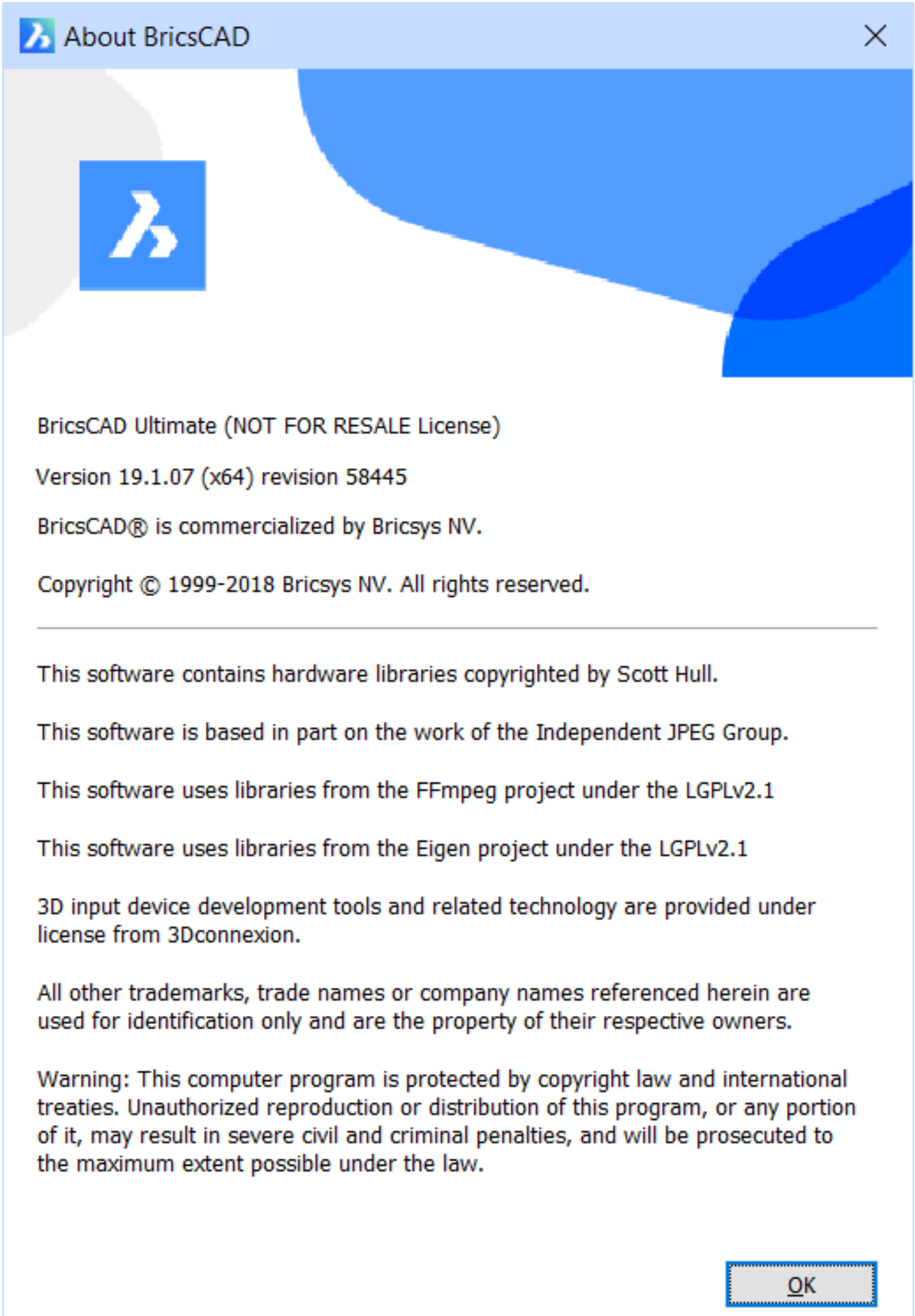
Figure 6.45



Accessing the Command

 command bar: **about** transparent: **'about** menu bar: **Help | About BricsCAD** ribbon: **Home | Help | About BricsCAD** : **about** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A dlg.png}) Click **OK** to close the dialog box.

Figure 6.46


About BricsCAD
BricsCAD Ultimate (NOT FOR RESALE License)
Version 19.1.07 (x64) revision 58445
BricsCAD® is commercialized by Bricsys NV.
Copyright © 1999-2018 Bricsys NV. All rights reserved.
This software contains hardware libraries copyrighted by Scott Hull.
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This software uses libraries from the FFmpeg project under the LGPLv2.1
This software uses libraries from the Eigen project under the LGPLv2.1
3D input device development tools and related technology are provided under license from 3Dconnexion.
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Warning: This computer program is protected by copyright law and international treaties. Unauthorized reproduction or distribution of this program, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the maximum extent possible under the law.
OK

Command Options

Option	Description
OK	Dismisses the dialog box.

Related Commands

Status (6.27.46) - reports on the state of the drawing. LicProperties (6.20.24) - reports the license number and owner.

Figure 6.47



6.4.2 AcisIn

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Imports SAT and SAB files in ACIS format (as exported by other CAD systems), and then creates 3D solid models from the data.

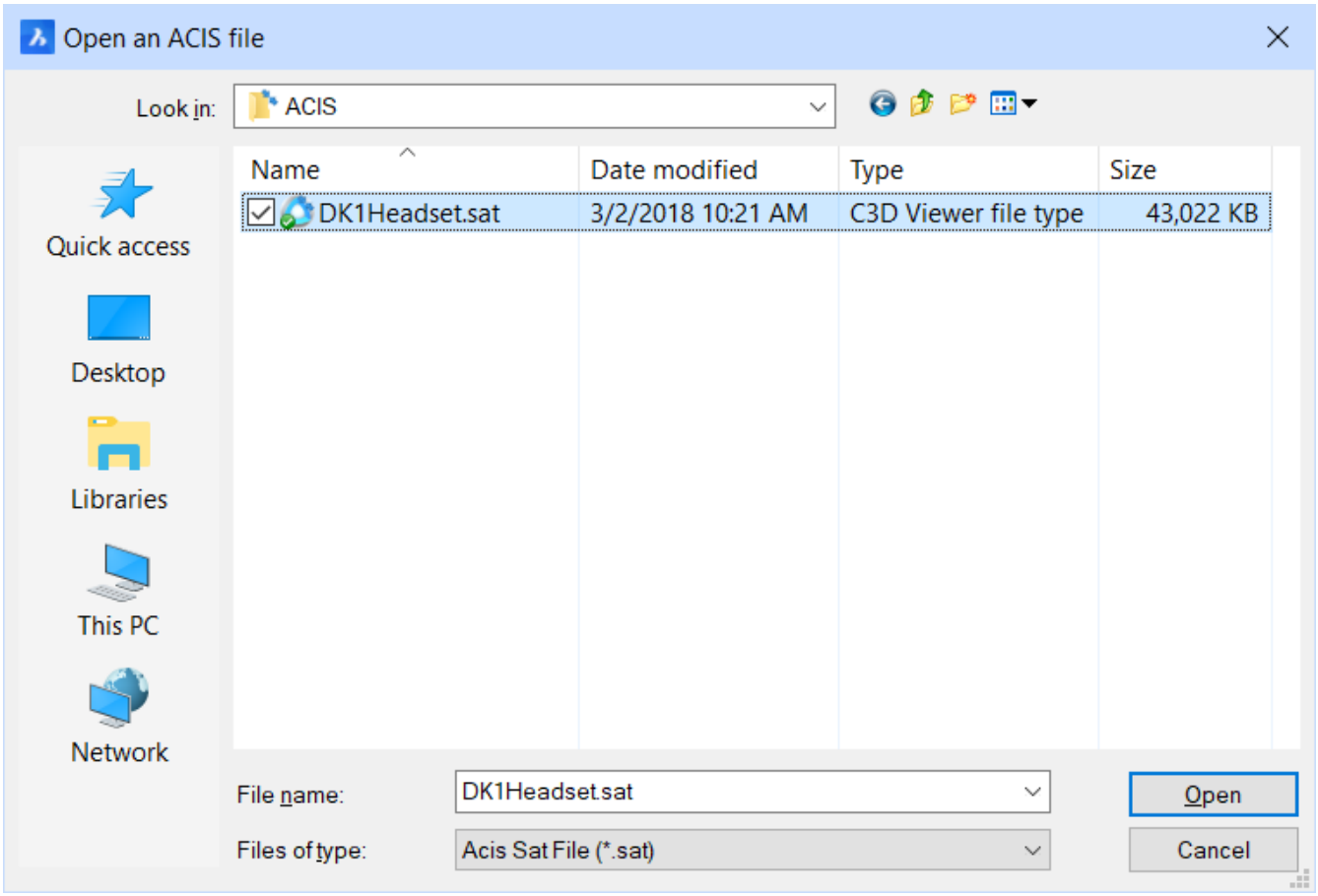
Accessing the Command

 command bar: **acisin** menu bar: **Insert | Acis In** : **acisin** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/acisin-dlg.png})

1. Choose a SAT or SAB file.
2. Click **Open**.The data in the file is converted to 3D solid entities, and then placed on layer 0.

Use this command to import 3D solids from other CAD packages. Use the AcisOut (6.4.3) command to create SAT or SAB files.

Figure 6.48



Dialog Box Options

Option	Description
File name	Specifies the name of the ACIS file.
Files of type	Choose an ACIS format: • SAB- ACIS binary format;short for ”save as binary.” • SAT- ACIS ASCII format;short for ”save as text.”
Open	Opens the 3D solid in the current drawing.If you cannot see the entity, use theZoom (6.35.7)All command.

Related Commands

AcisOut (6.4.3) - exports 3D solids and surface entities in SAT format. ConvToSolid (6.9.48) - converts 3D entities into solid models.

Figure 6.49



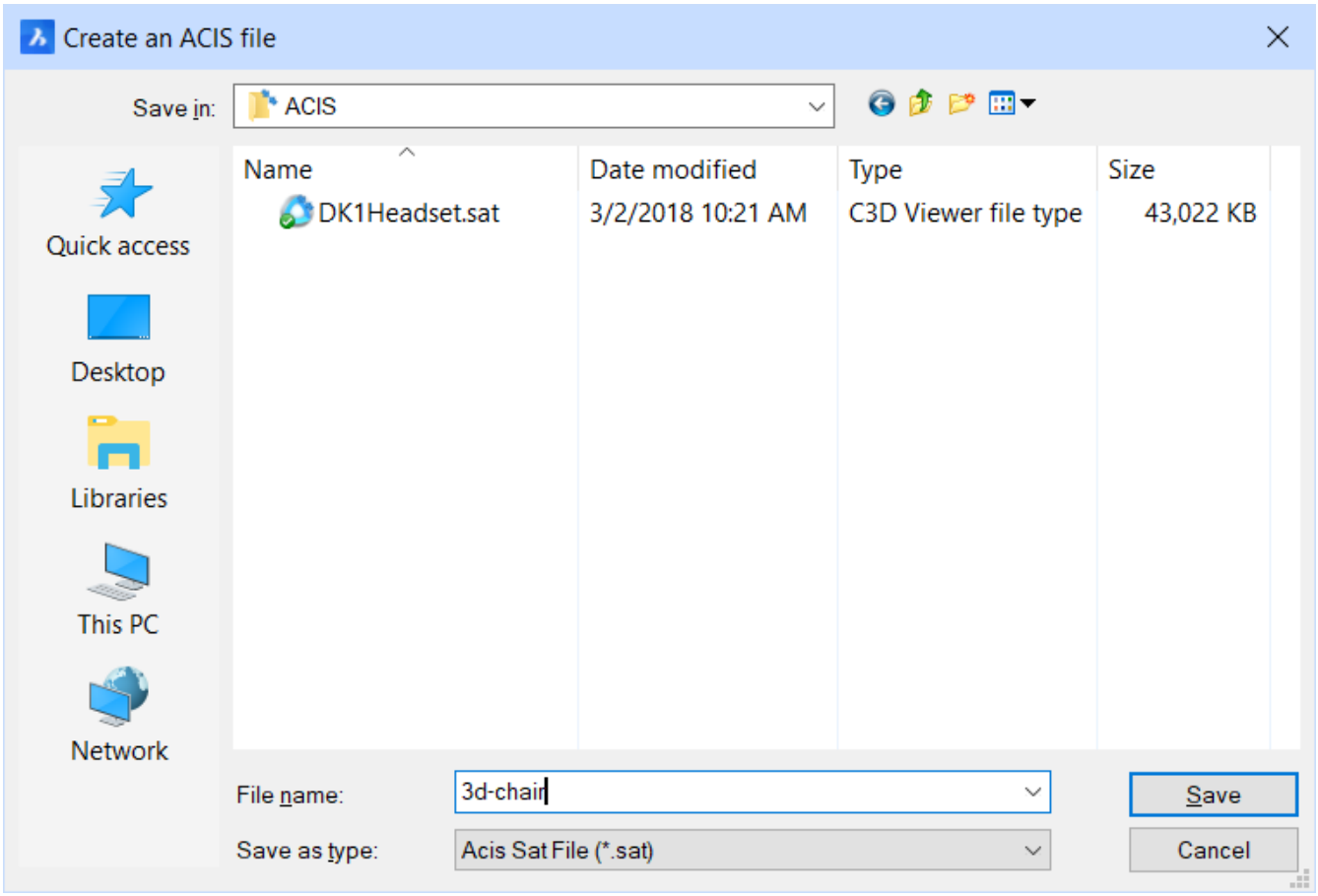
6.4.3 AcisOut

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Exports 3D solids and surface entities in ASCII or binary ACIS format. Exported files can be shared with other 3D CAD programs capable of solid modeling.

Accessing the Command


command bar: **acisout** menu bar: **Insert | Acis Out** : **acisout Prompts you in the command bar:** Select ACIS entities to save: **(Choose one or more 3D solids or surface entities)** Select ACIS entities to save: (**Press Enter to continue the command)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comm dlg.png}) Name the SAT file, chose a format, then click **Save**.The 3D solid entities are converted to binary or ASCII data in the file. Use the AcisIn (6.4.2) command to import SAT files.

Figure 6.50



Command Options

Option	Description
File name	Names the ACIS file.
Files of type	Choose an ACIS format: • **SAB** - ACIS binary format;short for "save as binary." • **SAT** - ACIS ASCII format;short for "save as text."
Save	Saves the selected 3D solid entities.

Related Commands

AcisIn (6.4.2) - imports ACIS solids into the drawing. ConvToSolid (6.9.48) - converts 3D entities into solid models.

Figure 6.51



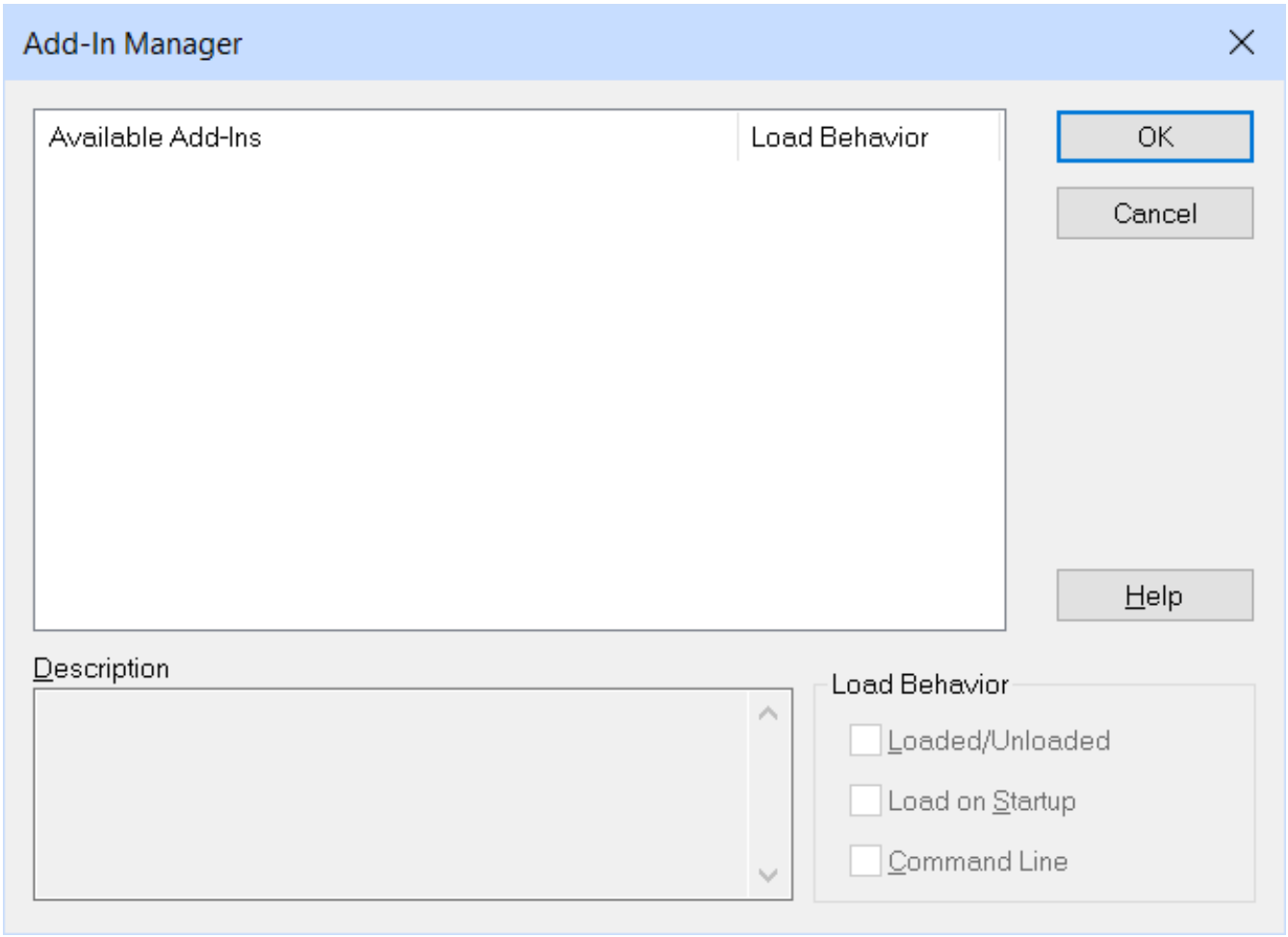
6.4.4 AddInMan

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Displays the Add-In Manager for controlling how Visual Basic for Applications modules are loaded.

Accessing the Command

command bar: **addinman** shortcut: **hift+F11** alias: **Alias** menu bar: **Tools | VBA | Add-in Manager** : **addinman** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/addinmanager-dlg.png}) Change the load behavior of add-ins programs, and then click **OK**.

Figure 6.52


Add-In Manager
Available Add-Ins
Load Behavior
OK
Cancel
Help
Description
Load Behavior
Loaded/Unloaded
Load on Startup
Command Line

Command Options

Option	Description
Available Add-Ins	Lists the names of VBA COM modules available to BricsCAD. ”VBA COM” is short for Visual Basic for Applications Common Object Model.
Load Behavior	Determines how modules are loaded: • **Loaded/Unloaded** - toggles the module between loaded and unloaded. • **Load on Startup** - loads modules when BricsCAD starts up.
OK	Saves the changes, and then dismisses the dialog box.
Cancel	Undoes the changes, and then dismisses the dialog box.

Related Commands

AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Commands (6.9.37) - lists the names of internal and external commands. CuiLoad (6.9.56) and CuiUnload (6.9.57) - loads and unloads menu files. Customize (6.9.58) - customizes the user interface. MenuLoad (6.21.14) and MenuUnload (6.21.15) - loads and unloads menu files. Netload (6.22.3) - loads a .NET application. Security (**??**) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not. VbaLoad (6.32.2) and -VbaLoad (6.32.3) - loads VBA projects VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaMan (6.32.4) - manages VBA projects. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.53



6.4.5 AddSelected

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates new entities based on an existing entity that you select.

Accessing the Command

 command bar: **addselected** menu bar:**Edit | Add Selected** toolbar: **Standard |**  toolbar: **Modify |**  ribbon:**Home | Settings | Select | Add Selected** : **addselected Prompts you in the command bar:** Select an entity: **(Select an entity in the drawing)** Launches the command that creates the selected entity type.For instance, select a spline and the program immediately starts the Spline command.Some entities are not yet supported.

Command Options

Option	Description
Select an entity	Select one entity in the drawing: • Entity type is supported - the program starts the drawing command to create that entity. • Entity type is not supported - the program reports ”This entity type is not yet supported”

Related Commands

All entity creation commands.

Figure 6.54



6.4.6 Align

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Aligns one entity with another through moving, rotating, and scaling.

Accessing the Command

 command bar: **align** alias: **al** menu bar: **Modify | Align** toolbar: **Modify** |  ribbon:**Change | Arrange | Align** : **align Prompts you in the command bar:** Select entities: **(Select one or more source entities; do not select "destination" entities)** Entities in set: 1Select entities: **(Select more entities, or else press Enter to end entity selection)** Specify first source point: **(Pick a point on or near the source entity)** Specify first destination point: **(Pick the matching point on or near the destination entity)** Specify second source point: **(Pick another point on or near the source entity)** Specify second destination point: **(Pick the matching point on or near the destination entity)** Specify third source point: **(Press Enter for 2D entities; pick source and destination points for 3D alignment)** Scale objects based on alignment points [Yes/No] <No>: **(Enter Y or N)**

Command Options

Option	Description
Select entities	Selects entities that are to be aligned; do not select destination entities, otherwise they are also moved.

To move entities, specify one pair of source and destination points to move entities: (\vref {fig:C:/Dev/Task-2448/images/static (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image54.jpg})

Option	Description
Specify first source point	Specifies the source alignment point, such as #1a in the figure above.
Specify first destination point	Specifies the destination alignment point, such as #1b in the figure.

To move and rotate entities, specify a second pair of source and destination points to move and rotate entities: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image55.jpg}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image47.png})

Option	Description
Specify second source point	Specifies the second source alignment point, such as #2a in the figure above. If you press Enter at this prompt, the source object moves.
Specify second destination point	Specifies the matching destination point, such as #2b in the figure.

To move and rotate 3D entities, specify a third pair of source and destination points to move and rotate 3D entities.

Option	Description
Specify third source point	Specifies the source point for 3D alignment; press Enter to move and rotate 2D entities.
Specify third destination point	Specifies the matching destination point.

To scale the source entities:

Option	Description
Scale objects based on alignment points	Optionally scales the source object: • **Yes** - scales objects relative to the picked points

Tutorial: How to Align Entities Click here (1.4.9) to watch

Figure 6.55

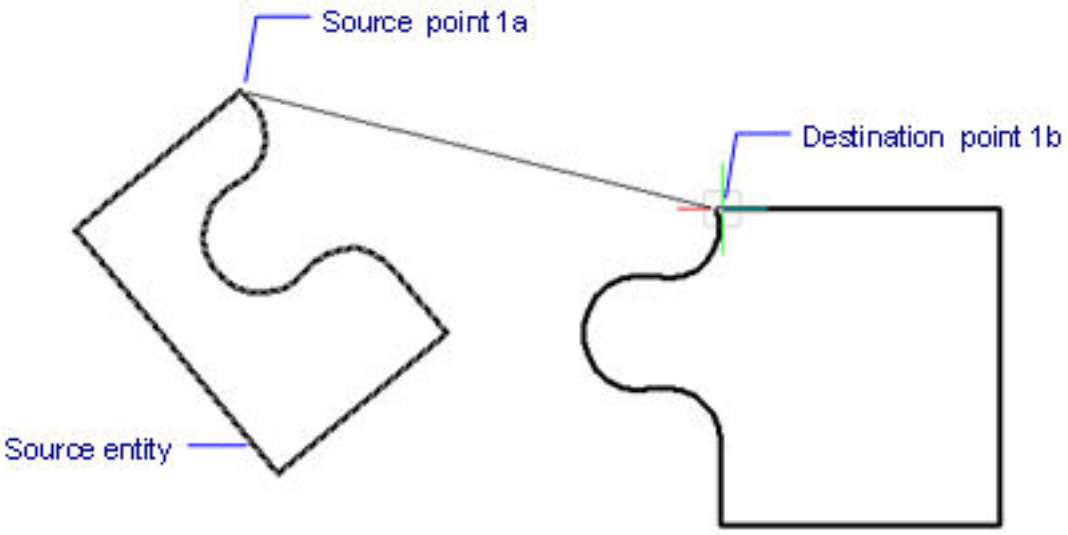


Figure 6.56

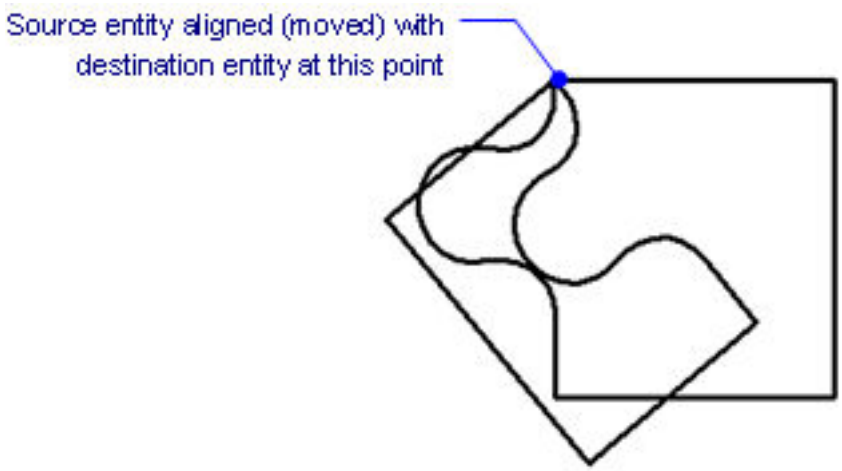


Figure 6.57

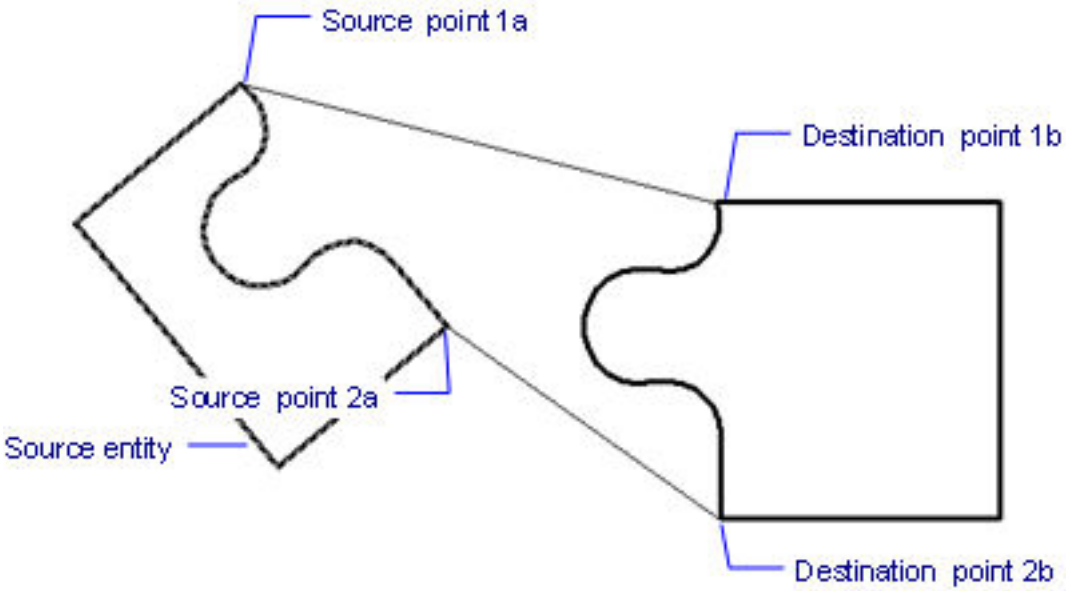
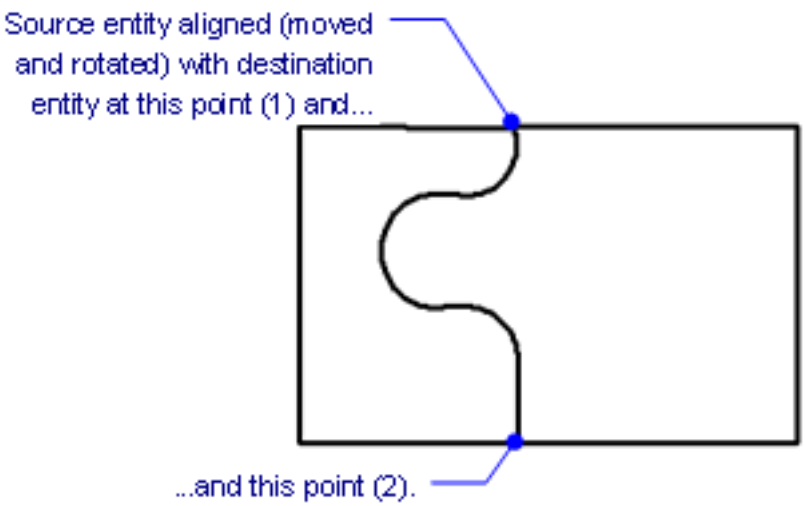


Figure 6.58



Procedures

- Aligning an entity in 2D (1.23.14)
- Aligning an entity in 3D (1.23.14)

Related Commands

Move (6.21.30) - moves entities in 3D. Rotate (6.26.34) - rotates entities in 2D. Rotate3D (6.26.35) - rotates entities in 3D. Scale (6.27.6) - scales entities.

Figure 6.59



6.4.7 AlignSpace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adjusts viewport angle, zoom factor, and pan position based on alignment points specified in model space and paper space.This command works only in paper space.

Accessing the Command

command bar: **alignspace** transparent: **'alignspace** : **alignspace Prompts you in the command bar:** First alignment point in Model space: **(Click inside the viewport you want to align, and then pick the first alignment point)** Second alignment point in Model space or <None>: **(Click on a second point in the same viewport, or enterN)** First alignment point in Paper space: **(Click a point in paper space; this point will be made coincident with the first alignment point)** Second alignment point in Paper space: **(Click another point in paper space; this point will be made coincident with the second alignment point)** Activate the viewport which will be used for alignment. **(Click in a viewport, and then press Enter)**

Command Options

Option	Description
First (second) alignment point in Model space	Follow these steps: 1. In a layout tab, click a paperspace viewport to make it current. 2. Pick a first alignment point that represents a position in model space. 3. Choose one: • Pick a second alignment point that specifies the view size and rotation in model space • Or, enter **N** to skip this step
None	Press Enter to skip the selection of the second alignment point in model space.The selected point will automatically coincide with the alignment point you pick in paper space.
First (second) alignment point in Paper space	Follow these steps: 1. Click a point in paper space; this point will be made coincident with the first alignment point in model space 2. Click another point in paper space; this point will be made coincident with the second alignment point of model space, if any
Activate the viewport which will be used for alignment	Choose a layout viewport by clicking in it, and then press Enter

Procedures

- To align model space and paper space points. (1.16.14)

Related Commands

MvSetup (6.21.39) - creates multiple paper space viewports; aligns, rotates and scales paper space viewports. MView (6.21.37) - creates one or more viewports in layout tabs. MSpace (6.21.34) - switches to model space. PSpace (6.24.54) - switches to paper space.

Figure 6.60



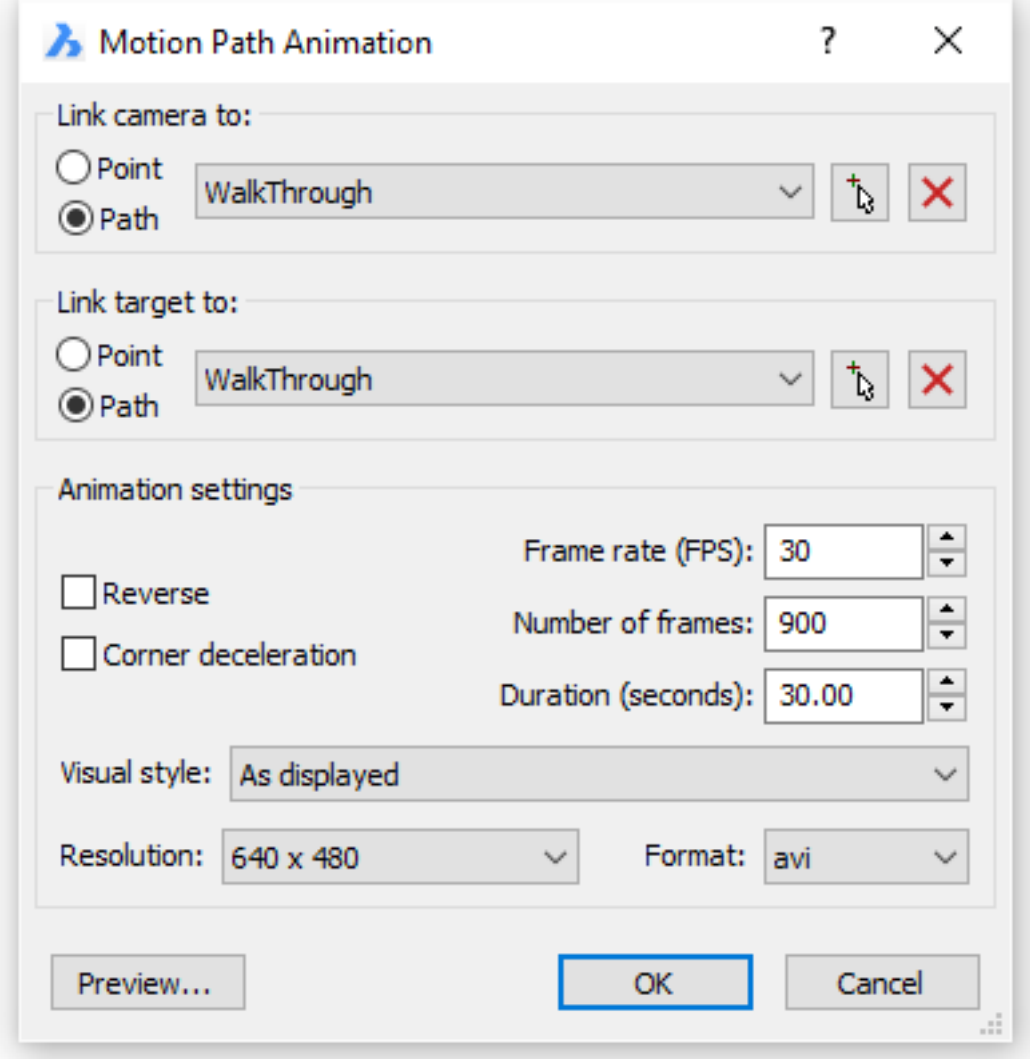
6.4.8 AniPath

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.

Accessing the Command


command bar: **anipath** menu bar: **View | Motion Path Animations** ribbon: **View | Motion Path Animations |** 
: **anipath** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Moti

Figure 6.61


Motion Path Animation
?
Link camera to:
Point
Path
WalkThrough
Link target to:
Point
Path
WalkThrough
Animation settings
Frame rate (FPS):
30
Reverse
Number of frames:
900
Corner deceleration
Duration (seconds):
30.00
Visual style:
As displayed
Resolution:
640 x 480
Format:
avi
Preview...
OK
Cancel

Command Options

Option	Description
Link camera to	Defines the location of the camera. Select an option: Point : Links a camera to a static point in the drawing. If **Link target to** is set to **Point** , this option is not available for the camera. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Do one of the following: • Click the list button, and choose a previously saved point. • Click the **Pick** ( ) button. You are prompted to specify a point in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou Type a name for the point, then click the **OK** button. The point is added to the points list. • Click the **Delete** button ( ) to remove the currently selected point from the points list. Path : Links a camera to a linear entity in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Do one of the following: • Click the list button, and choose a previously saved path. • Click the **Pick** ( ) button. You are prompted to pick a linear entity in the drawing. A path can be a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou Type a name for the path, then click the **OK** button. The path is added to the paths list. • Click the **Delete** button ( ) to remove the currently selected path from the paths list.

Link target to	Defines the target point of the camera. If the camera is linked to a point, the target must be linked to a path. If the camera is linked to a path, the target can can be linked to either a point or a path. Select an option: Point : Links the target to a static point in the drawing. If **Link camera to** is set to **Point** , this option is not available for the target. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Do one of the following: • Click the list button, and choose a previously saved point. • Click the **Pick** ( ) button. You are prompted to specify a point in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so Type a name for the point, then click the **OK** button. The point is added to the points list. • Click the **Delete** button ( ) to remove the currently selected point from the points list. Path : Links a target to a linear entity in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Do one of the following: • Click the list button, and choose a previously saved path. • Click the **Pick** ( ) button. You are prompted to pick a linear entity in the drawing. A path can be a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so Type a name for the path, then click the **OK** button. The path is added to the paths list. • Click the **Delete** button ( ) to remove the currently selected path from the paths list.

Animation Settings	Allows to control the output of the movie file. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Frame rate (FPS) : defines the number of frames per second. When you change the **Frame rate** , the **Number of frames** will be adjusted according to the **Duration** . Number of frames : specifies the total number of frames in the movie. When you change the **Number of frames** , the **Duration** will be adjusted accordingly. Duration (seconds) : specifies the duration in seconds of the movie. When you change the **Duration** , the **Number of Frames** will be adjusted accordingly. Reverse : Reverser the sequence of the frames. Corner deceleration : Slows down the camera at corners in the animation path. Visual style : Defines the visual style for the movie. Click the button, then select a visual style in the list. All visual styles (6.31.13) in the current drawing are available. Resolution : Sets the width x height resolution of the movie. Click the button to select a resolution. Available resolutions are: 160x120, 320x240, 640x480, 800x600, 1024x768, 1280x1024, 1600x1024, 1920x1080, 1920x1200. Format : Specifies the file format of the movie. Available formats are: **avi** , **mpg** and **wmv** .
Preview	Displays the movement of the camera.
OK	Generates the movie. Displays a **File** dialog: 1. Select a folder. 2. Type a name for the movie. 3. (option) Select a file type, thus overriding the previously selected **Format** setting. 4. Click the **Save** button to create the movie file.

Related Commands

Camera (6.9.2) - creates perspective views. -Render (6.26.23) - generates a photorealistic rendering using materials and lights through the command bar. Render (6.26.22) - generates a photorealistic rendering using materials and lights through a dialog box. -VisualStyles (6.31.14) - manages visual style definitions at the command line. VisualStyles (6.31.13) - creates and controls visual style definitions in the Drawing Explorer.

Figure 6.62

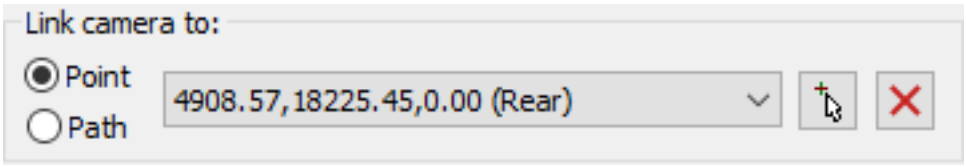


Figure 6.63

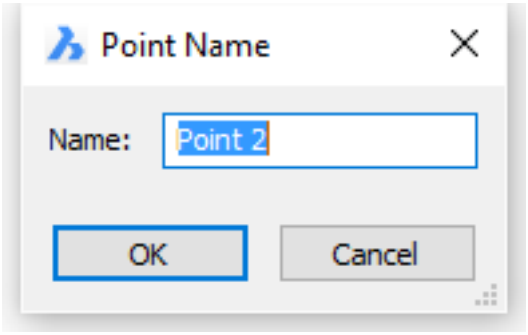


Figure 6.64



Figure 6.65

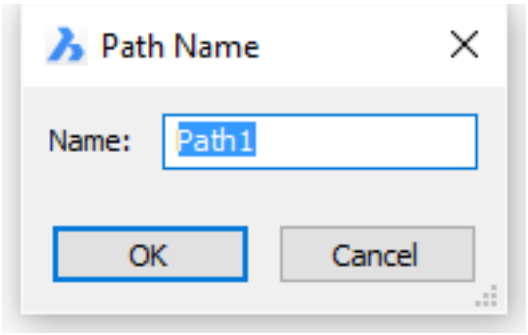


Figure 6.66

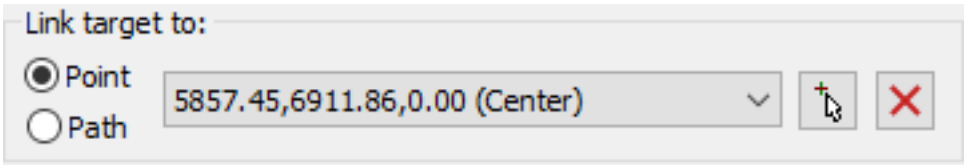


Figure 6.67

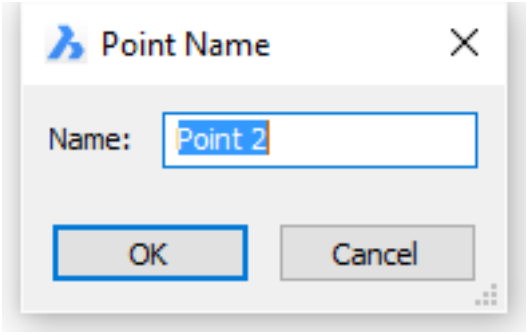


Figure 6.68



Figure 6.69

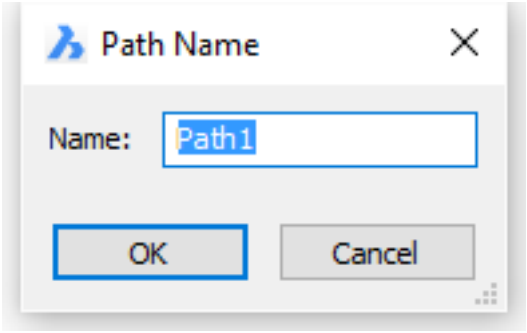


Figure 6.70

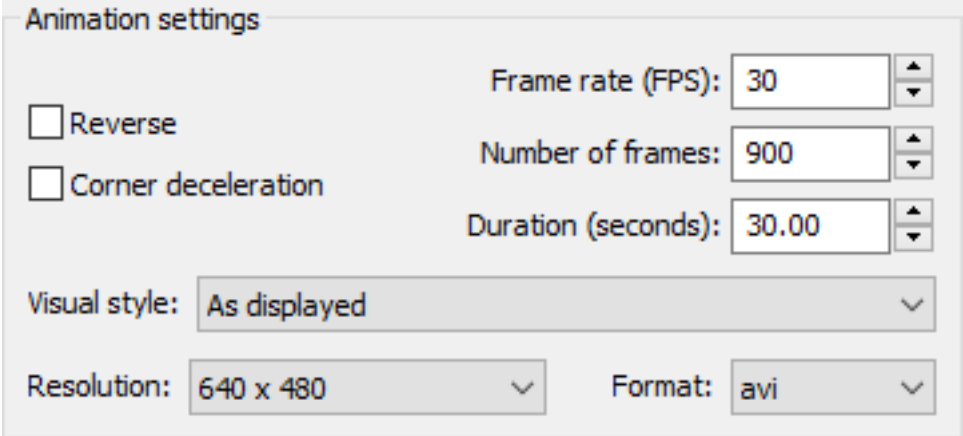


Figure 6.71



6.4.9 AnnoUpdate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Updates the annotative style of selected annotative entities to match the current style.

Accessing the Command

 command bar: **annoupdate** ribbon:**Home | Annotation Scaling | Update Annotations** : **annoupdate Prompts you in the command bar:** Select annotative entities: **(Select entities)** Entities in set: <x> Select annotative entities: **(Select more entities, or press Enter to finish the command)** <x> found <y> were updated

Command Options

Option	Description
Select annotative entities	• Select one or more annotative entities, such as text, dimensions, and hatch patterns • Enter **all** to select all of them; the program will ignore non-annotative entities

Related Commands

AnnoReset - resets all scale representations of the selected annotative entities to the location of their current scale representation.

Figure 6.72



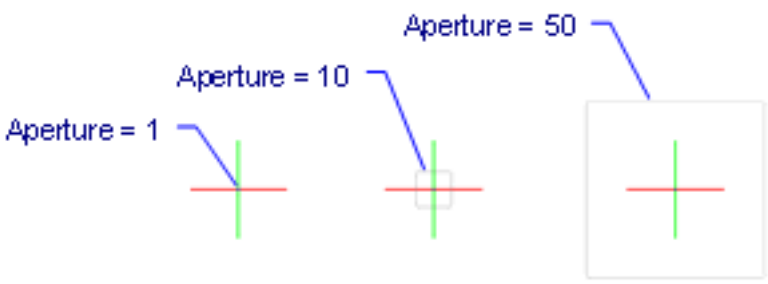
6.4.10 Aperture

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the size of selection area for snapping to entities.

Accessing the Command

command bar: **aperture** transparent: **'aperture** alias: **ap** menu bar: **Settings | Entity Snap Precision** toolbar: **Settings |** ribbon:**Settings | Entity Snap Precision** : **aperture Prompts you in the command bar:** Selection area for entity snap (1-50 pixels) <10>: **(Enter a value between 1 and 50, or press Enter to keep the default value.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image38.png}) The aperture cursor works with entity snaps only. The program searches the area inside the rectangular aperture cursor for geometric features, such as endpoints and mid points. A large aperture cursor makes it easier for you to hover over geometric features, but then the program may find too many matching entity snap points; a smaller one may be more difficult for you to position over geometry, but is more precise at snapping to geometric features. The default value of 10 is a good compromise. By default, the aperture cursor is turned on. Use the **ApBox** system variable to turn it off.

Figure 6.73



Command Options

Option	Description
Selection area for entity snap	Changes the size of the aperture cursor; range is between 1 and 50 pixels wide.

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar.

Figure 6.74



6.4.11 Apparent

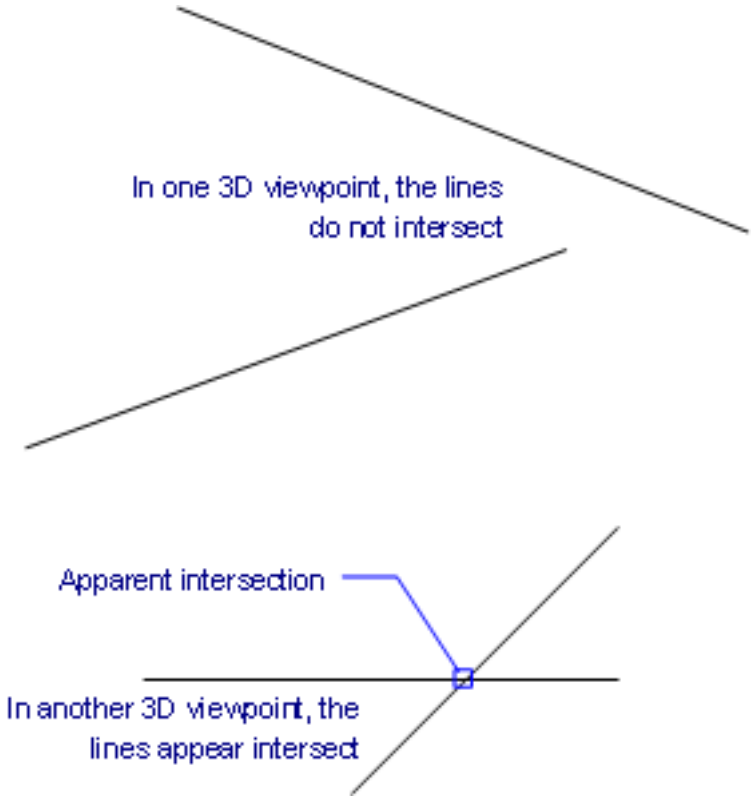
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles Apparent intersection entity snap; snaps to the intersections of entities, even when they only appear to intersect in 3D space.

Accessing the Command



command bar: **apparent** transparent: **apparent** aliases: **app,planviewint,2dintersection (6.2.1)**toolbar: **Entity Snaps |** ribbon: **Home | Entity Snaps | Snap to Apparent** : **apparent** Displays no prompt in the command bar. Each time you enter the command, it toggles the apparent entity snap mode, from on to off, to on again. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image698.png}) This entity snap snaps to intersecting entities, including those that appear to intersect in 3D space.

Figure 6.75



Command Option

Option	Description
Apparent	Entering this command toggles apparent intersection (2dInsersection) entity snap This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.76



6.4.12 Arc

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws arcs by a variety of methods.

Accessing the Command

command bar: **arc** alias: **a** menu bar: **Draw | Arc** toolbar: **Draw |** ribbon: **Draw | Draw | Arc** : **arc Prompts you in the command bar:** Enter start of arc or [Center]: **(Pick a point, enter coordinates, or enter an option)** Enter second point or [Angle/Center/Direction/End/Radius]: **(Pick a point, enter coordinates, or enter an option)** End point: **(Pick a point or enter coordinates)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou **Subsequent uses of this command change the initial prompt to:** Enter start of arc or [Center/Follow] <Follow>: **(Pick a point, enter coordinates, or enter an option)** This command draws arcs by many methods. The full list is in the Draw | Arc menu shown below: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Ar On each Arc icon a large green dot ( ) indicates the first point that you pick. A small green dot ( ) indicates the second point, and a small white dot ( ) indicates the final point.

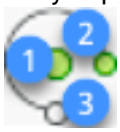


Figure 6.77

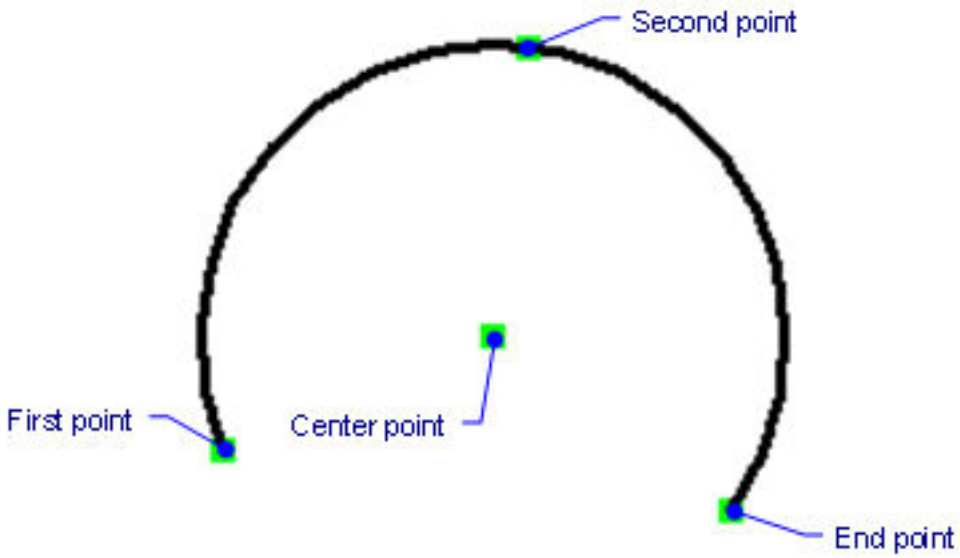
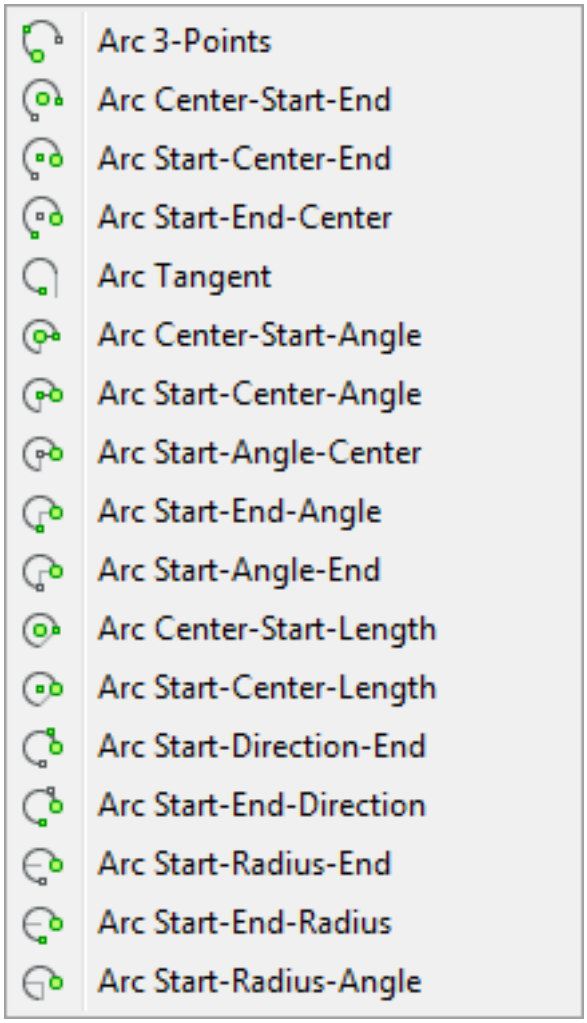


Figure 6.78



Command Options

Option	Description
ENTER to use last point	Continues the arc tangent from the last point picked.
Center	Specifies the center point of the arc.
Follow	Continues the arc tangent from the last point picked (same as pressing Enter).
Start of arc	Specifies a different start point for the new arc.
Angle (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/arcs/arc_start_point40.jpg})	Specifies the included angle, the angle between the arc's start point and: • Endpoint • Center point
Center  (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/arcs/arc_start_point42.jpg})	Specifies the center point of the arc; after this, you can specify the following to complete the arc: • Endpoint • Included angle • Length of the chord
Direction	Specifies the direction of the tangent to the arc's start point; following this, you specify the following: • End point
End point	Specifies the arc's end point; following this, you can specify the following: • Center point • Included angle • Direction of tangency • Radius
Radius	Specifies the arc's radius; following this, you can specify the following: • Endpoint • Included angle
Second point	Specifies a point along the arc's circumference; following this, you specify the following: • Endpoint

Procedures

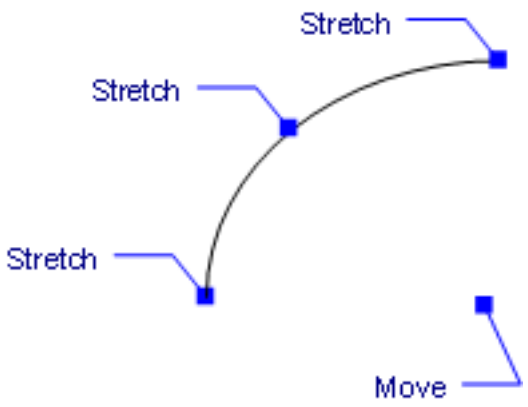
- General procedure to draw an arc (1.17.16)
- Editing an arc (1.17.16)

Grips Editing

Arcs can be edited directly through grips:

1. Select the arc. Notice that it has four grips.
2. Drag the grips to perform the following actions:

Figure 6.79



3. • Drag the endpoint and midpoint grips to stretch the arc.
   • Drag the center grip to move the arc.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image753.png})

Related Commands

Circle (6.9.20) - draws arcs and circles. Ellipse (6.13.5) - draws elliptical arcs. Join (6.18.1) - joins arcs. PEdit (6.24.20) - converts arcs to polylines. PLine (6.24.27) - draws connected polyline arcs. DimRadius (6.11.17) and DimDiameter (6.11.10) - dimensions arcs. Change (6.9.10) - changes the endpoints of arcs. Entity snaps that work with arcs: Center (6.9.3) - snaps to the center of arcs. Endpoint (6.13.7) - snaps to the endpoints of arcs. Quadrant (6.25.7) - snaps to the 90-degree points on the circumference of arcs. Tangent (6.29.9) - snaps to the tangency of arcs.

Figure 6.80

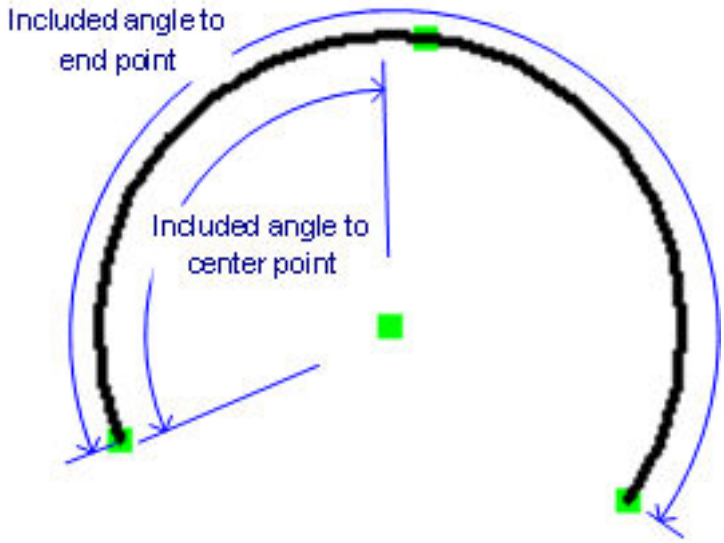


Figure 6.81

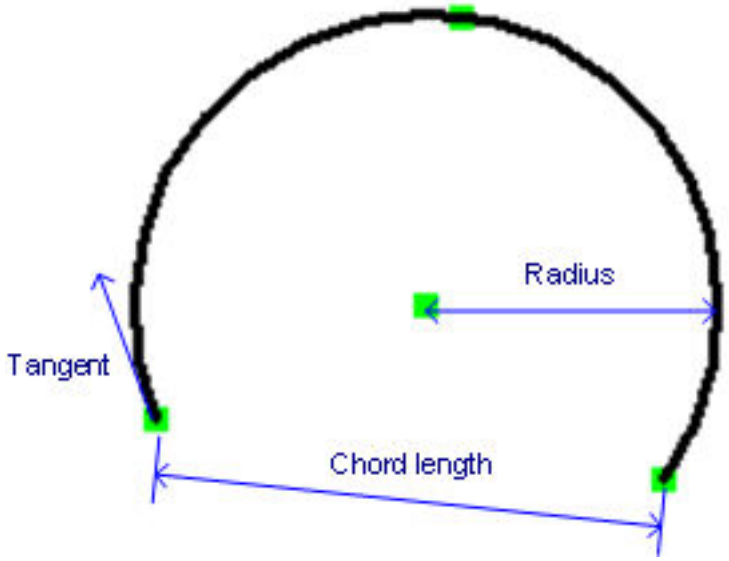


Figure 6.82



6.4.13 Area

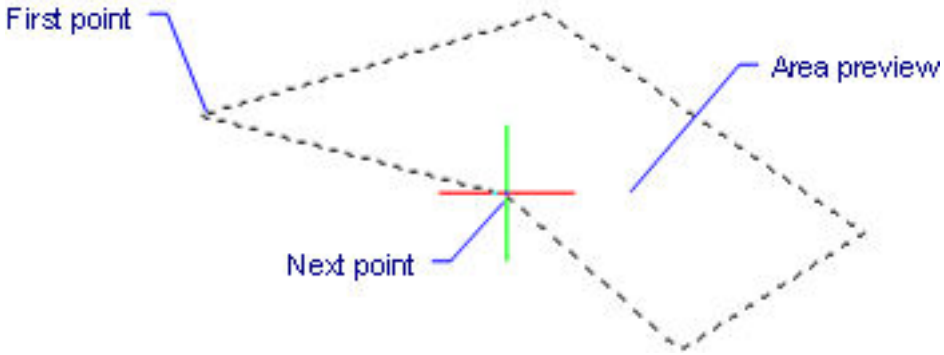
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Finds the area and perimeter of 2D entities.

Accessing the Command



 command bar: **area** alias: **aa** menu bar: **Tools | Inquiry | Area** toolbar: **Inquiry** | ribbon:**Tools | Inquiry | Area** : **area Prompts you in the command bar:** Select first point [Entity/Add/Subtract]: **(Pick a point, or enter an option)** <Next point>: **(Pick a point)** <Next point>: **(Pick another point)** <Next point>: **(Pick another point, or else press Enter to end the command)** Use entity snap modes to ensure your picks are accurate. As you pick points, BricsCAD previews the area with a dashed polygon. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so
When you press Enter to indicate you have ended picking points, BricsCAD reports the area and perimeter like this: **Area = 6.1836 sq. units, Perimeter = 11.2968** This command does not work with solids (2D), traces, rays, xlines, or hatches. To find the area of hatches, use the Properties command.

Figure 6.83



Command Options

Option	Description
Entity	Provides the area and/or perimeter of the selected entity. The information reported by this command varies according to the entity selected: • Line and sketch - length • Arc and elliptical arc - length • Circle and ellipse - area and perimeter • Closed polyline and closed spline - area and perimeter • Open polyline and spline - area and perimeter, as if they were closed • 3D objects - surface area
Add	Allows you to add up the areas of two or more entities. The prompt changes to: **Adding: Entity/Subtract/ <First point>** - **(pick a point, enter an option, or press Enter to exit Adding mode.)**
Subtract	Allows you to removes areas and/or entities from the total area and perimeter; this option can only be used after you have found the area of at least one entity. The prompt changes to: **Subtracting: Entity/Subtract/ <First point>** - **(pick a point, enter an option, or press Enter to exit Subtracting mode.)**
First point	Specifies the first point of the area to be measured. The prompt changes to: **<Next point>** - **(pick another point, or press Enter to end the area specifying.)**

Related Commands

Dist (6.10.31) - reports the 2D and 3D distances between two points. Id (6.17.1) - reports the x,y,z coordinates of picked points. List (6.20.35) - lists the area and other parameters of entities in the command bar. MassProp (6.21.4) - reports the properties of solid models and regions. Properties (6.24.49) - lists the area and other parameters of entities in a palette.

Figure 6.84



6.4.14 AppLoad

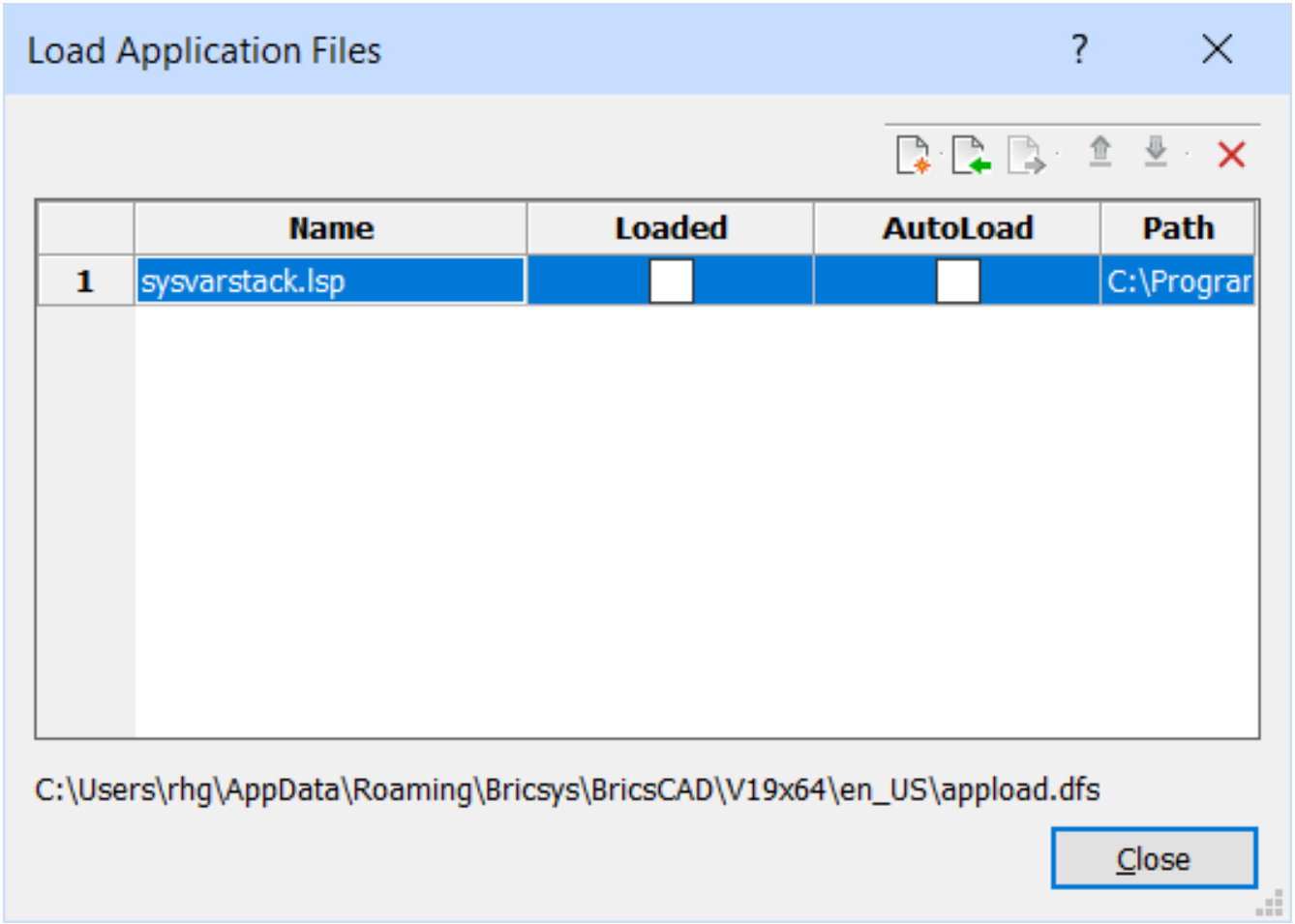
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads and unloads applications to run inside BricsCAD:

- LSP - LISP applications from BricsCAD and other CAD programs
- DES - DES LISP applications
- BRX - BricsCAD runtime extension applications
- TX - Teigha extension applications, from Open Design Alliance
- ARX - AutoCAD runtime extension applications from Autodesk
- DBX - DBX applications
- DRX - Drawing runtime extension applications
- DVD - Visual Basic applications
- MNL - LISP expressions for menu files

Accessing the Command

command bar: **appload** menu bar: **Tools | Load Applications** : **appload** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/appload.png})

Figure 6.85


Load Application Files
?
Name
Loaded
AutoLoad
Path
1
sysvarstack.lsp
C:\Prograr
C:\Users\rhg\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\appload.dfs
Close

Command Options

Option	Description
 Add Application File	Displays the Select Application Files dialog box: 1. (Optional) Click the **Files of type**droplist to select the specific file type to load. 2. Select one or more files to load; hold down the **Shift** or **Ctrl** keys to select more than one file at a time. 3. Click**Open**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/ap select-dlg.png})
 Load Selected Application	Loads the selected application(s) into BricsCAD.
 Unload Selected Application	Unloads the selected application(s) from BricsCAD.
 Move Application Upward	Moves the selected application(s) up in the list.
 Move Application Downward	Moves the selected application(s) down in the list.
 Remove Selected Application	Removes the selected application(s) from the list.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. Commands (6.9.37) - lists the names of internal and external commands. CuiLoad (6.9.56) and CuiUnload (6.9.57) - loads and unloads menu files. Customize (6.9.58) - customizes the user interface. MenuLoad (6.21.14) and MenuUnload (6.21.15) - loads and unloads menu files. Netload (6.22.3) - loads a .NET application. Security (6.27.15) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not. VbaLoad (6.32.2) and -VbaLoad (6.32.3)- loads VBA projects. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaMan (6.32.4) - manages VBA projects. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.86

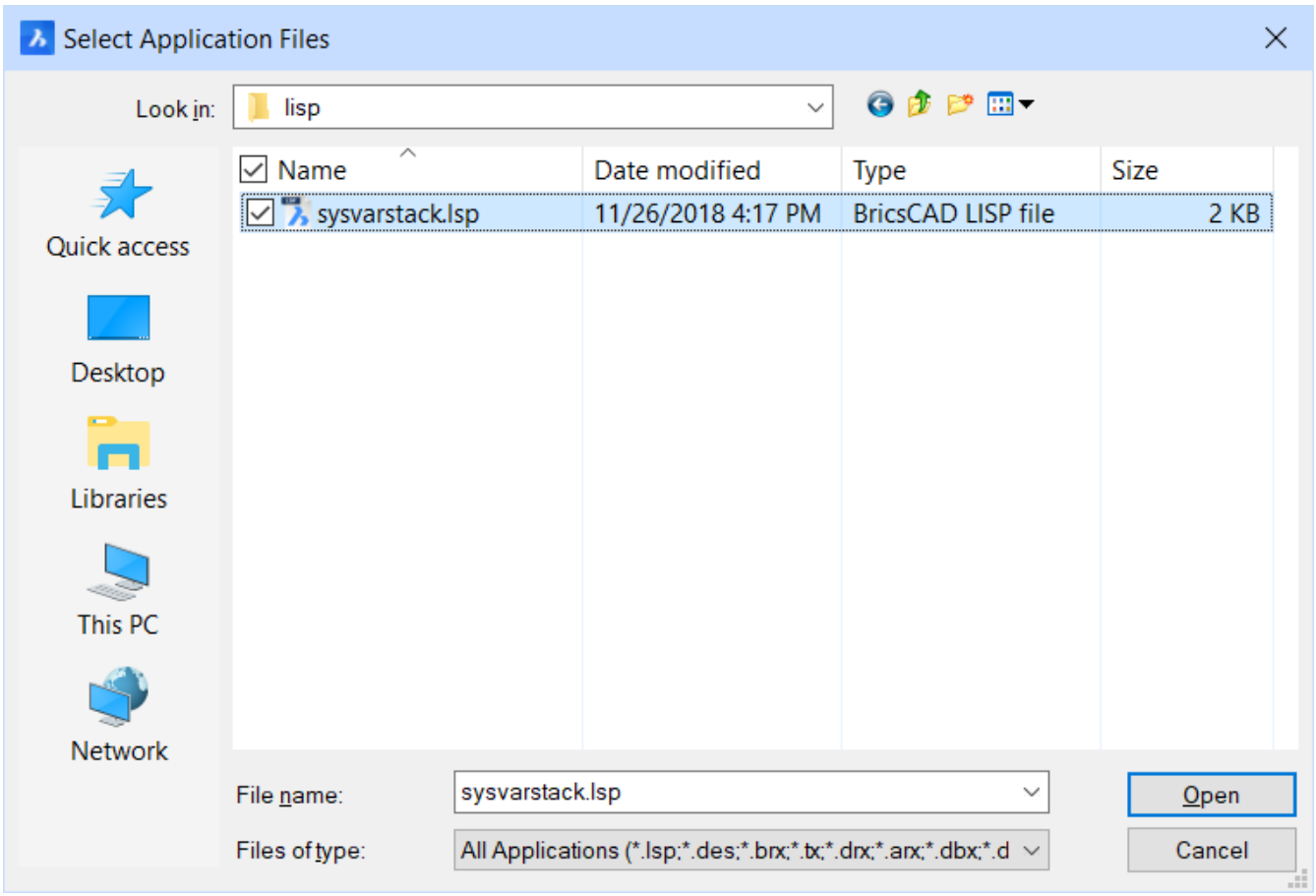

Select Application Files
Look in:
lisp
Name
Date modified
Type
Size
sysvarstack.lsp
11/26/2018 4:17 PM
BricsCAD LISP file
2 KB
Quick access
Desktop
Libraries
This PC
Network
File name:
sysvarstack.lsp
Open
Files of type:
All Applications (*.lsp;*.des;*.brx;*.tx;*.drx;*.arx;*.dbx;*.d
Cancel

Figure 6.87



6.4.15 -Array

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates a static polar or rectangular array of entities; displays prompts at the command bar.

Accessing the Command

command bar: **-array** : **-array Prompts you in the command bar:** Select entities to array: **(Select one or more entities.)** Entities in set: 1 Select entities to array: **(Press Enter to end entity selection.)** Type of array [Polar/Rectangular] <Rectangular>: **(Enter an option.)** This command creates linear, rectangular, and polar arrays.

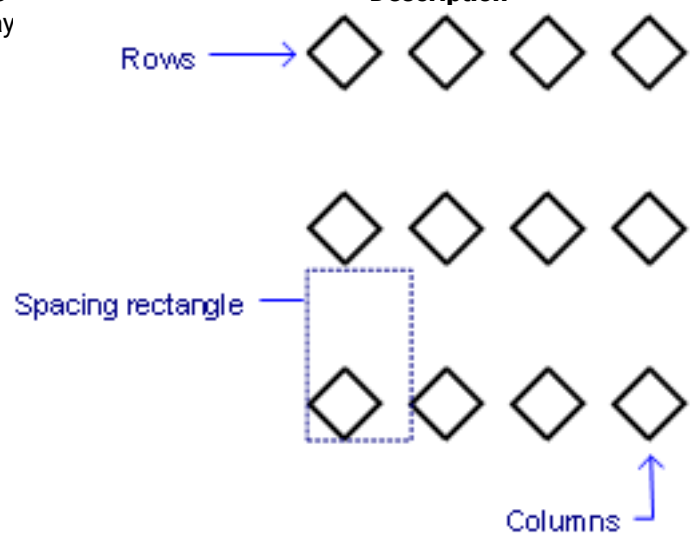
Command Options

Option	Description
Select entities to array	Specifies the entities to be arrayed. Use any entity selection method; press Enter when you are done.
Polar	Creates polar (circular) arrays. Prompts you: **Center of polar array or [Base]: (Pick a point, or type B.) ENTER to specify angle between items/<Number of items to array>: (Enter a number, or press Enter.) Angle to fill (+ for ccw, - for cw)<360>: (Type a positive or negative value or press Enter to fill 360°.) Rotate entities around the array? [Yes-Rotate entities(Yes)/No-Do not rotate(No)]<Yes>: (Type N or Y.)**
Rectangular	Creates linear, rectangular, or square arrays. Prompts you: **Number of rows in the array <1>: (Enter the number of rows.) Number of columns <1>: (Enter the number of columns.) Vertical distance between rows, or spacing rectangle: (Specify the distance between rows, or pick two points.) Horizontal distance between columns: (Specify the distance between columns.)**

Rectangular arrays have rows and columns: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/

Figure 6.88

Rectangular Array Options	Description
Number of rows in the array	ıs; for a linear array, enter 1. ls, enter a negative number.



Number of columns	Specifies the number of columns. To draw the array to the left, enter a negative number.
Vertical distance between rows	Specifies the distance between rows of entities.
Spacing rectangle	Specifies two points that indicate the spacing between rows and columns.
Horizontal distance between columns (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image41.png})	Specifies the distances between columns of entities.

Polar Array Options	**Description**
Base	Relocates the base point of the selected entities.
Center of polar array	Specifies the center point; enter coordinates or pick a point.
ENTER to specify angle between items	Specifies the angle between each copy.
Number of items to array	Specifies the number of items in the array; enter a number larger than 1
Angle to array	Determines the extent and direction of the array. Enter 360 for a full circle of copies; enter a smaller number to draw a partial polar array. Enter a positive angle to draw the array counter clockwise; enter a negative number to draw the array clockwise.
Rotate entities around the array	Determines whether copies are rotated when they are arrayed: • No does not rotate entities; all copies look the same as the original. • Yes rotates copies. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Related Commands

3dArray (6.3.2) - creates 3D arrays. Array (6.4.16) - associatively distributes entity copies in an evenly spaced rectangular, polar or path array. ArrayClassic (6.4.17) - creates a static polar or rectangular array of entities through a dialog box. ArrayEdit (6.4.20) - allows editing associative array entities and their source entities. ArrayPath (6.4.21) - associatively distributes entity copies evenly along a path into multiple rows, and levels. ArrayPolar (6.4.23) - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. ArrayRect (6.4.25) - associatively distributes entity copies into any number of rows, columns, and levels. MInsert (6.21.17) - inserts arrays of blocks.

Figure 6.89

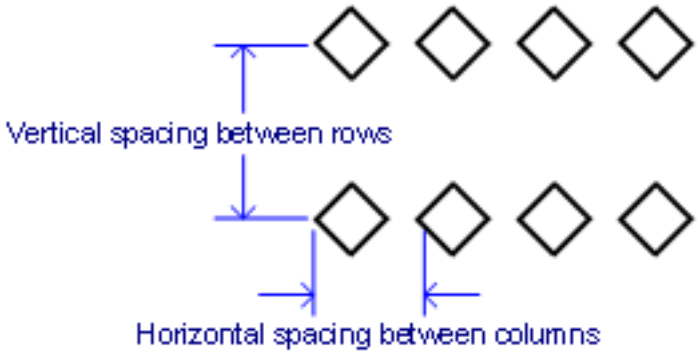


Figure 6.90

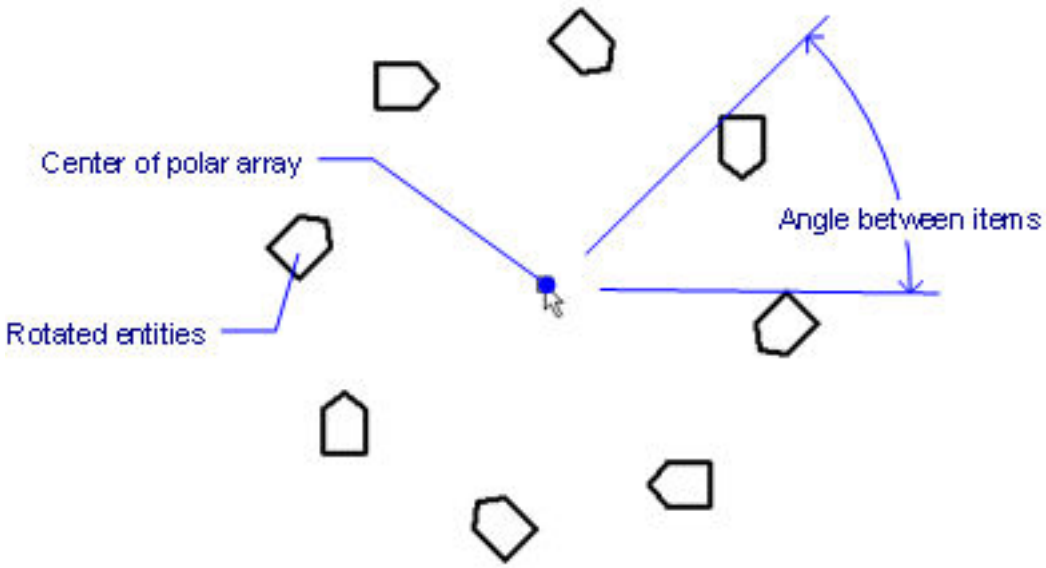


Figure 6.91

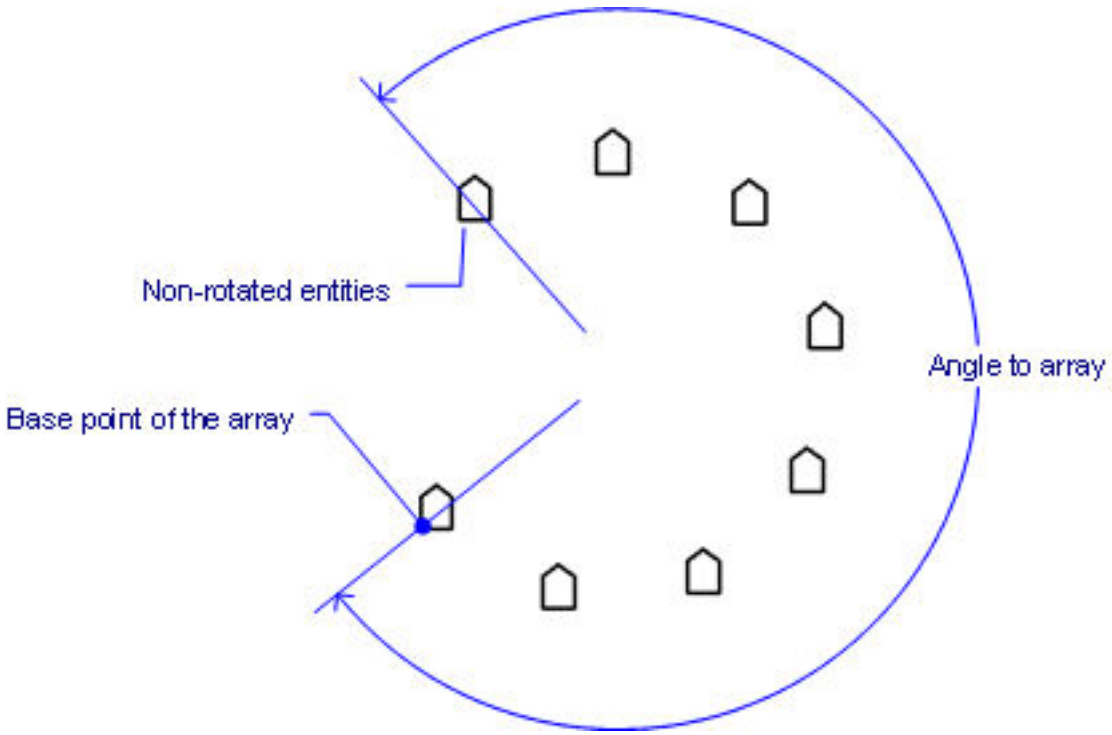


Figure 6.92



6.4.16 Array

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates 2D and 3D arrays of entities in rectangular or polar patterns, or along a path.

Accessing the Command

 command bar: **array** alias: **ar** menu bar: **Modify | Array | 2D Array** toolbar: **Modify |** **ribbon:** : **array** **Prompts you in the command bar:** Select entities: **(Select an entity)** Entities in set: 1 Select entities: **(Select more entities or press Enter to stop selecting entities)** Enter array type [Rectangular (6.4.25)/PAth (6.4.21)/POlar (6.4.23)] <Rectangular>: **(Choose an array type or press Enter to accept the default)** The default array type is saved by the ArrayType variable.The ArrayAssociativity variable controls whether static or associative arrays are created.

Command Options

Option	Description
Rectangular	Distributes entity copies into any number of rows, columns, and levels (z direction). See the ArrayRect (6.4.25) command.
Path	Distributes entity copies evenly along a path into multiple rows and levels.See the ArrayPath (6.4.21) command.
Polar	Distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows and levels.See the ArrayPolar (6.4.23) command.

Related Commands

3dArray (6.3.2) - creates 3D arrays. -Array (6.4.15) - creates a polar or rectangular array of entities; displays prompts at the command bar. ArrayClassic (6.4.17) - creates a static polar or rectangular array of entities through a dialog box. ArrayEdit (6.4.20) - allows editing associative array entities and their source entities. ArrayPath (6.4.21) - associatively distributes entity copies evenly along a path into multiple rows, and levels. ArrayPolar (6.4.23) - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. ArrayRect (6.4.25) - associatively distributes entity copies into any number of rows, columns, and levels. MInsert (6.21.17) - inserts arrays of blocks.

Figure 6.93



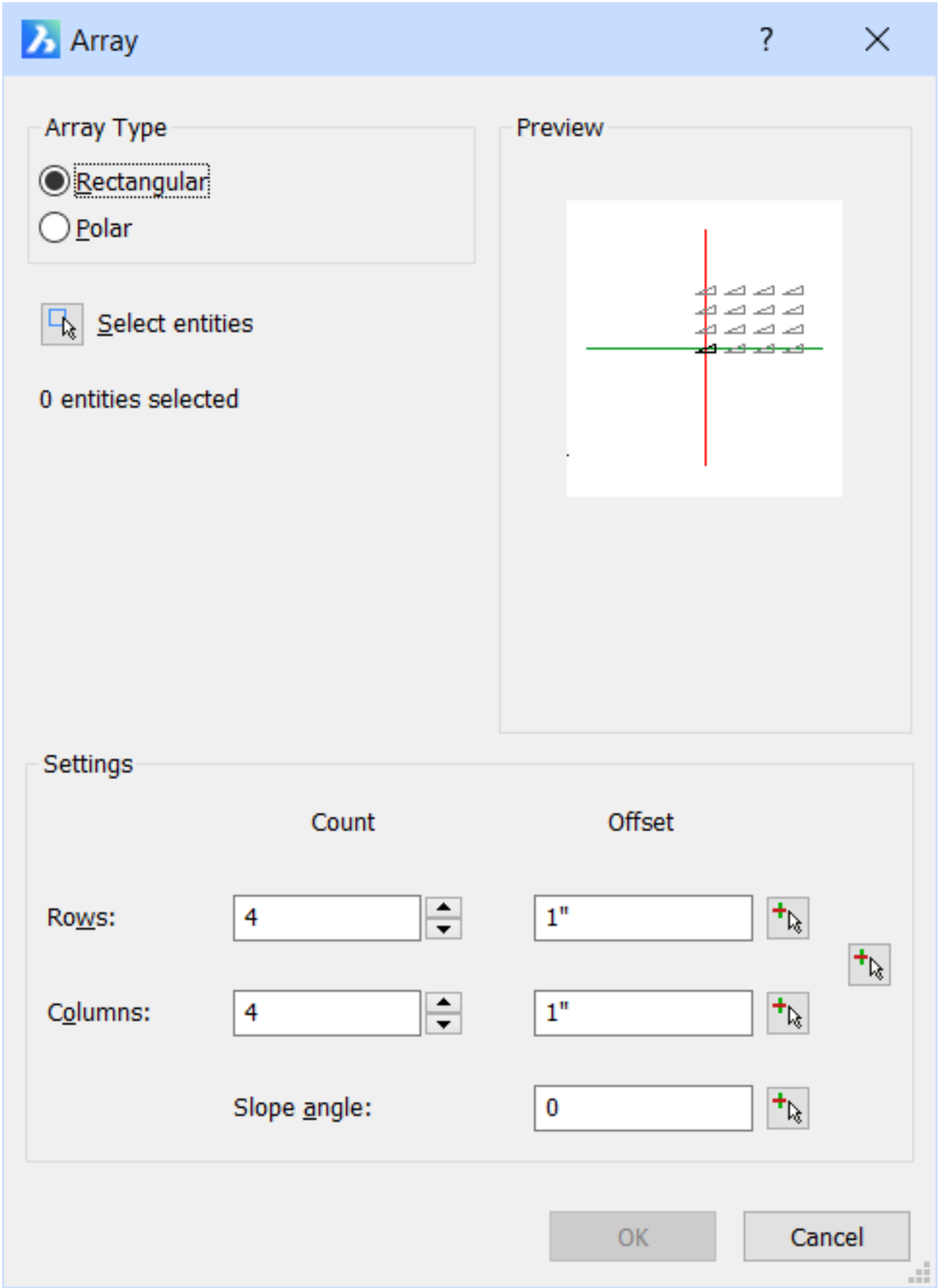
6.4.17 ArrayClassic

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a static 2D rectangular or polar array of entities through a dialog box.

Accessing the Command

command bar: **arrayclassic** menu: **Modify | Array | Array Dialog Box** : **arrayclassic** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/arrayclassic-dlg.png}) (Dialog box for creating rectangular arrays.)

Figure 6.94


Array
?
Array Type
Rectangular
Polar
Select entities
0 entities selected
Preview
Settings
Count
Offset
Rows:
4
1"
Columns:
4
1"
Slope angle:
0
OK
Cancel

Command Options

Rectangular Array Option	Description
 Select Entities	To select entities to be arrays: 1. Click the **Select Entities** button () 2. The Array dialog box temporarily closes, and youprompted to to select entities: **Select entities to array** -Select one or more entities 3. Press Enter to return to the dialog box
Rows Count	Specify the number of rows: • Enter a value for the number of rows • Click the spin arrows to increase or decrease the distance Notice that the preview window updates with the number of rows.
 Rows Offset	Specify the offset distance between subsequent rows.A positive offset distance adds rows from left to right, a negative one right to left. Do one of the following: • Type a distance in the **Rows Offset** field. • Click the **Pick Points** button () • The Array dialog box temporarily closes and prompts you: **Vertical distance between rows** -Specify two points in the drawing
Columns Count	Specify the number of columns: • Enter a value for the number of rows • Click the spin arrows to increase or decrease the distance Notice that the preview window updates with the number of columns.
 Columns Offset	Specify the offset distance between columns.A positive offset distance adds columns from bottom to top, while a negative distance places columns from top to bottom. Do one of the following: • Type a distance in the **Columns Offset** field. • Click the spin arrows to increase or decrease the distance • Click the **Pick Points** button (). The **Array** dialog box temporarily closes, and then prompts you: **Horizontal distance between columns** - specify two points in the drawing

 Spacing Rectangle	Specifies the spacing between rows and columns with a rectangle that you pick in the drawing: • Length of rectangle = distance between columns • Width of rectangle = distance between rows The distance is measured from the start of one entity to the start of the next. 1. Click the **Pick Points** button () 2. The Array dialog box temporarily closes and prompts you: **Specify spacing rectangle** - specify two points in the drawing.
 Slope Angle	Specifies the angle of the array from the x axis (0 degrees).Do one of the following: • Type an angle in the **Slope angle** field. • Click the **Pick Points** button ().The **Array** dialog box temporarily closes and prompts you: **Select angle of array** - pick two points in the drawing Notice that the preview window updates with the number of columns.

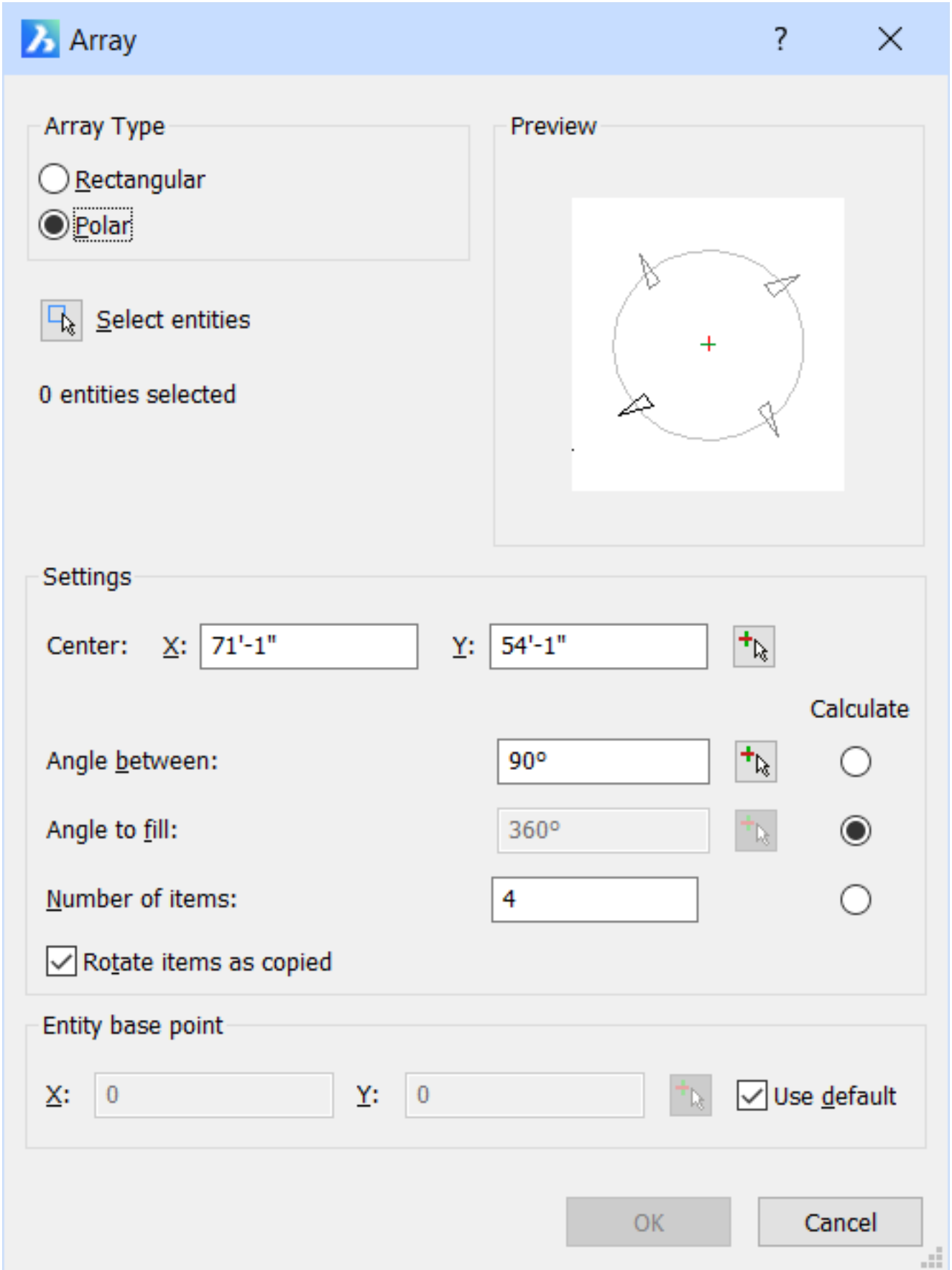
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/array-polar-dlg.png})

Polar Array Options	**Description**
 Select Entities	To select entities to be arrays: 1. Click the **Select Entities** button () 2. The Array dialog box temporarily closes, and youprompted to to select entities: **Select entities to array** -Select one or more entities 3. Press Enter to return to the dialog box
Center	Specifies the center point of the array.Do one of the following: • Enter coordinates in the X and Y fields. • Click the **Pick Point** button ().The Array dialog box temporarily closes and you are prompted:**Base/Center of polar array** -specify a point in the drawing.

Angle between	Specifies the angle between each copy of the entity in the array.Do one of the following: • Click the **Calculate** radio button to calculate the angle from the **Angle to fill** and **Number of items**. • Type an angle in the **Angle** field. • Click the **Pick Points** button ().The Array dialog box temporarily closes, and you are prompted:**Specify angle between** - pick a point in the drawing. As you do, a line displays dynamically between your pick point and the previously-selected center point.
Angle to fill	Determines the extent and direction of the array. • Enter 360 for a full circle of copies • Enter a smaller number to draw a partial polar array • Enter a positive angle to draw the array counter clockwise • Enter a negative number to draw the array clockwise Do one of the following: • Click the **Calculate** radio button to calculate the angle from **Angle between** and **Number of items** . • Type an angle in the **Angle** field. • Click the **Pick Points** button ().The Array dialog box temporarily closes and prompts you:**Specify angle between** - pick a point in the drawing. As you do, a line displays dynamically between your pick point and the previously-selected center point.
Number of items	Specifies the number of items in the array.Do one of the following: • Click the **Calculate** radio button to calculate the angle from the **Angle between** and **Angle to fill** . • Enter a number larger than 1. Notice that the preview window updates with the number of items.
Rotate items as copied	Determines whether copies are rotated when they are arrayed: • **No** - does not rotate entities; all copies have the same orientation as the original • **Yes** - rotates copies

Entity base point	Relocates the base point of the selected entities.Do one of the following: • Check the **Use Default** option (= lower left corner of the selection set bounding box). • Click the **Pick Point** button ().The Array dialog box temporarily closes and prompts you: **Base/Center of polar array** - specify a point in the drawing.

Figure 6.95


Array
?
Array Type
Rectangular
Polar
Select entities
0 entities selected
Preview
Settings
Center: X: 71'-1" Y: 54'-1"
Calculate
Angle between: 90°
Angle to fill: 360°
Number of items: 4
Rotate items as copied
Entity base point
X: 0 Y: 0
Use default
OK
Cancel

Related Commands

3dArray (6.3.2) - creates 3D arrays. -Array (6.4.15) - creates a polar or rectangular array of entities; displays prompts at the command bar. Array (6.4.16) - associatively distributes entity copies in an evenly spaced rectangular, polar or path array. ArrayEdit (6.4.20) - allows editing associative array entities and their source entities. ArrayPath (6.4.21) - associatively distributes entity copies evenly along a path into multiple rows, and levels. ArrayPolar (6.4.23) - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. ArrayRect (6.4.25) - associatively distributes entity copies into any number of rows, columns, and levels. MInsert (6.21.17) - inserts arrays of blocks.

Figure 6.96



6.4.18 -ArrayClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Exits the array editing begun by the ARRAYEDIT (6.4.20) command, after prompting you at the command line to save or discard the changes you made to the source entities in the array; applies to associative arrays only.

Accessing the Command

 command bar: **-arrayclose** Quad: **Modify**|  : **-arrayclose Prompts you in the command bar:** Save changes to array? [Yes/No] <Yes>: **(Type Y or N)**

Command Options

Option	Icon	Description
Yes		Saves the changes made to the source entities of the array, and then exits the array editing state.
No		Exits the array editing state and discards all changes, thus returning the array to its original state.

Related Commands

ArrayEdit (6.4.20) - edits the entity(ies) that make up associative array. ArrayClose (6.4.19) - exits the array editing mode, and displays a dialog box to save or discard the changes made to the array source entities.

Figure 6.97



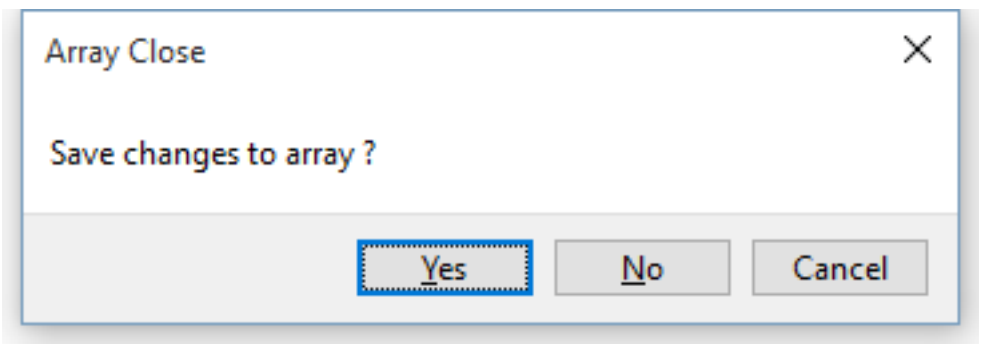
6.4.19 ArrayClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Exits the associative array editing state, displaying a dialog box to save or discard the changes made to the array's source entities. **Note:** This command is available only after an associative array is checked out with ARRAYEDIT (6.4.20) command.

Accessing the Command

command bar: **arrayclose** : **arrayclose** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU

Figure 6.98



Command Options

Option	Description
Yes	Saves the changes made to the source entities of the array and exits the array editing state.
No	Exits the array editing state and discards all changes, thus returning to the original state of the array.
Cancel	Closes the dialog box and stays in the array editing state.

Related Commands

ArrayEdit (6.4.20) - allows editing associative array entities and their source entities. -ArrayClose (6.4.18) - Exits the array editing state and prompts you in the command bar to save or discard the changes made to the array source entities.

Figure 6.99



6.4.20 ArrayEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Edits the entities that make up associative arrays.

Accessing the Command

 command bar: **arrayedit** menu bar: **Modify | Array | Edit Array** toolbar: **Arrays |**  : **arrayedit Prompts you in the command bar:** Select associative array: **(Select an array entity) The following prompts depend on the type of the selected array entity -- rectangular, path or polar.** Rectangular Array Options [Source/REPlace/Base point (6.4.25)/Rows (6.4.25)/COLumns (6.4.25)/Levels (6.4.25)/RESet/eXit]<eXit>: **(Select an option)** Path Array Options [Source/REPlace/Method (6.4.21)/Base point (6.4.21)/Items (6.4.21)/Rows (6.4.21)/Levels (6.4.21)/Align items (6.4.21)/Z direction (6.4.21)/RESet/eXit]<eXit>: **(Select an option)** Polar Array Options [Source/REPlace/Base point (6.4.23)/Items (6.4.23)/Angle between (6.4.23)/Fill angle (6.4.23)/Rows (6.4.23)/Levels (6.4.23)/ROTate (6.4.23)/RESet/eXit]<eXit>: **(Select an option)**

Command Options

Option	Description
Source	Edits the source entities of the array.Prompts you: 1. **Select item in array** - **(select one of the items in the selected array entity.)** 2. Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so 3. Click the **OK** button on the **Array Editing State** dialog box. 4. Notice that only the selected item in the array is now displayed so that you can modify, add, and delete entities. 5. When done, type ArrayClose (6.4.19) at the command prompt; a message box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so 6. Click the **Yes** button. Notice that all the items of the array are now displayed. The change(s) you made to the one item are applied to all items in the array.

Replace	Replaces some or all items in the array.Prompts you: **Select replacement objects** - **(select one or more entities) Entities in set: 1 Select replacement objects**- **(select more entities, or press Enter to end entity selection) Enter base point of replacement objects or [Key point] <centroid>**- **(do one of the following actions)**: • Press Enter to use the centroid (center point) of the selection • Specify a point. • **Select key point on replacement object as base point** - when you move the cursor over the selected entities, all keypoints highlight one by one.Click to select a keypoint. **Select an item in the array to replace or [Source objects]** - **(do one of the following):** • Select one or more items in the array • **Source objects** - replaces all items
Reset	Restores erased entities, and removes any item overrides.Hold the **Ctrl**key to select more than one item in the associative array, then tap the **Delete** key to erase the selection.
Exit	Accept the changes and conclude the command.

Related Commands

3dArray (6.3.2) - creates 3D arrays. -Array (6.4.15) - creates a polar or rectangular array of entities; displays prompts at the command bar. Array (6.4.16) - associatively distributes entity copies in an evenly spaced rectangular, polar or path array. ArrayClassic (6.4.17) - creates a static polar or rectangular array of entities through a dialog box. -ArrayClose (6.4.18) - exits the array editing state and prompts you in the command bar to save or discard the changes made to the array source entities. ArrayClose (6.4.19) - exits the array editing state and displays a dialog box to save or discard the changes made to the array source entities. ArrayPath (6.4.21) - associatively distributes entity copies evenly along a path into multiple rows, and levels. ArrayPolar (6.4.23) - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. ArrayRect (6.4.25) - associatively distributes entity copies into any number of rows, columns, and levels. MInsert (6.21.17) - inserts arrays of blocks.

Figure 6.100

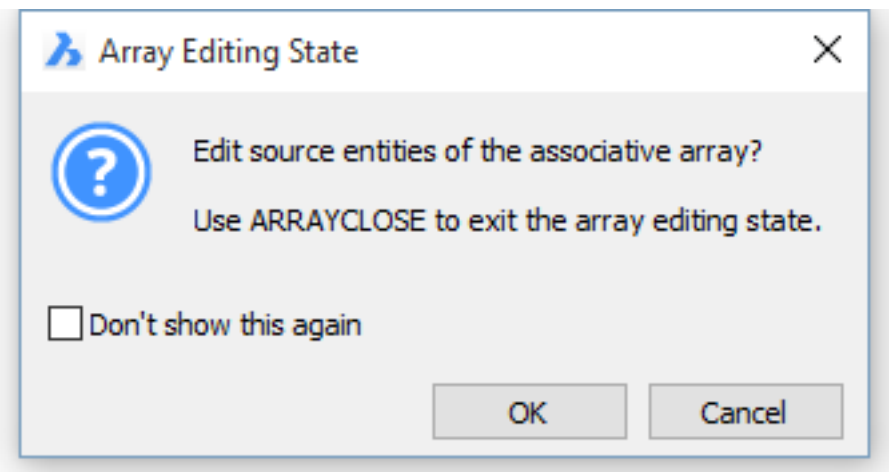


Figure 6.101

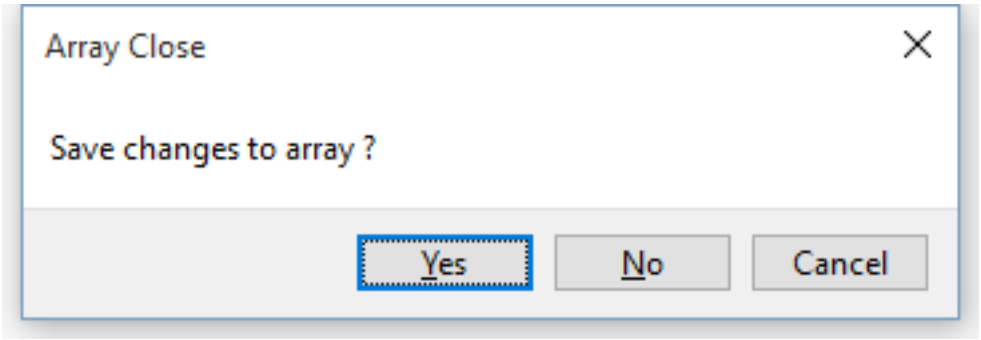


Figure 6.102



6.4.21 ArrayPath

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Associatively distributes entity copies evenly along a path into multiple rows, and levels.

Accessing the Command

command bar: **arraypath** menu bar: **Modify | Array | Path Array** toolbar: **Modify | Arrays |** ribbon:**Sketch | Arrange |**(Mechanical and SheetMetal workspaces) ribbon:**Solid | Edit |**(Mechanical and SheetMetal workspaces) ribbon:**Model | Modify 3D |**(BIM workspace) ribbon:**Change | Arrange |**(BIM and 3D Modeling workspaces) : **arraypath Prompts you in the command bar:** Select entities to array: **(Select an entity)** Entities in set: 1 Select entities to array: **(Select more entities, or else press Enter to end entity selection)** Type = Path, Associative = Yes Select path curve: **(Select an entity -- line, polyline, arc, circle, ellipse, elliptical arc, spline, helix, or 3D polyline)** [ASsociative/Method/Base point/Tangent direction/Items/Rows/Levels/Align items/Z direction/eXit]<eXit>: **(Enter an option, or else press Enter to end the command)**

Command Options

Option	Description
Associative	Determines whether an array entity is created or a series of copies of the selected entities. The default setting is saved by the ARRAYASSOCIATIVITY (1.13.11) system variable. Prompts you: **Create associative array? [Yes/No]<Yes> - (enter an option):** • **Yes** : An Array entity is created. • **No** : A series of copies of the selected entities is created. Parameters for associative arrays can be specified with parametric expressionsset in the Array properties, found in bottom part of the Mechanical Browser (3.2.2), Properties panel (6.24.49), or though the ARRAYEDIT (6.4.20) command. The **End offset** parameter is available only when the Method = Divide. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V20/Commands/A/Ar Entities making up associative array are modified with the ARRAYEDIT (6.4.20) command.
Method	Specifies how to distribute entities along the path.Prompts you: **Enter path method or [Divide/Measure]<Measure>** - **(choose an option)**: • **Divide**- distributes a specified number of items evenly-spaced along the path • **Measure**- places a sufficient number of items along the path, separated by the interval specifie
Base point	Sets the base point of the array. Items are placed relative to the base point.Prompts you: **Enter base point** - **(specify a point in the drawing)**

Tangent direction	Determines how the items are aligned relative to the starting direction of the path.Prompts you: **Select first point of tangent direction vector or [Normal]** - **(enter an option)**: • **2 points** - specify two points that define the tangency of the first item relative to the starting direction of the path. Depending on the Align Items setting, the other items are placed parallel to the first item or at the specified tangent direction relative to the direction of the path; see figures below, where the blue arrowhead indicates the tangency direction: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • **Normal** - orients the Z direction of the first item with the starting direction of the path; see figure below: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so
Items	Specifies the number of items or the distance between items, depending on the Method setting. • **Divide method prompts you:(Enter number of items along path <xxx>) -** specifies the number of items in the array. • **Method Measure method prompts you: (Enter the distance between items along path <xxx>)** - Specify a distance by enteringa value or picking two points. (xxx == maximum number items possible) **Enter number of items or [Fill entire path] <xxx>** - **(enteran option):** • **Enter a number of Items**- places items along the entire path at the specified distance. This option allows you to specify a smaller number if you only want to place items along a portion of the path. • **Fill entire path**- fills the entire path with items at the given spacing; the number of items changes should the length of the path be modified, such as by grips editing; see below.

Rows	Specifies the number of rows in the array, the distance between them and the incremental elevation.Prompts you: 1. **Enter the number of rows <1>** - **(specify the number of rows)** 2. **Enter the distance between rows or [Total] <1.0>** - **(enter an option):** • **Distance** - specify the distance between subsequent rowsby enteringa value or picking two points • **Total** - specify the distance between the first and the last row. 3. **Enter the incrementing elevation between rows <0.0>** - **(defines the increasing or decreasing elevation for each subsequent row.)**
Levels	Create 3D arrays by specifying the number and spacing of levels. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/A Prompts you: 1. **Enter the number of levels <1>** - **(specifies the number of levels)** 2. **Enter the distance between levels or [Total] <xxx>** - **(enter an option)** • **Distance** - specifies the distance between subsequent levels • **Total** - specifies the distance between the bottom and the top level
Align items	Specifies whether to align each item tangent to the path's direction. Alignment is relative to the first item's orientation.Prompts you: **Align arrayed items to path? <Yes>/No>** - **(enter an option):** • **Yes**- each item is placed tangent to the path direction; see top figure below • **No**- each item keeps the orientation of the first item; see bottom figure below (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so
Z direction	Determines whether to maintain the items' original Z direction or to bank the items naturallyalong a 3D path. Prompts you: **Maintain Z direction for all items in the array? [Yes/No]<Yes>** - **(enter an option:)** • **Yes** - Z-direction (vertical) is maintained; see figure below at left • **No** - items banked along the helix; see figure below at right (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so Direction.png})
Exit	Accept the current settings and conclude the command.

Grips Editing

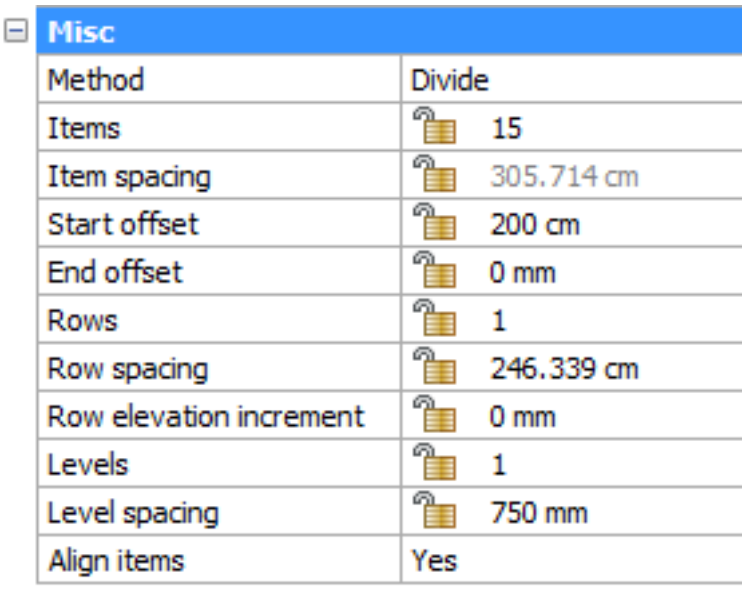
When grips editing the path, the array entity is recreated along the modified path.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Array/staticbricsyscom-1.png}) Editing the path length.	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/A 2.png}) Editing the path shape.

Related Commands

3dArray (6.3.2) - creates 3D arrays. -Array (6.4.15) - creates a polar or rectangular array of entities; displays prompts at the command bar. Array (6.4.16) - associatively distributes entity copies in an evenly spaced rectangular, polar or path array. ArrayClassic (6.4.17) - creates a static polar or rectangular array of entities through a dialog box. ArrayEdit (6.4.20) - allows editing associative array entities and their source entities. ArrayPolar (6.4.23) - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. ArrayRect (6.4.25) - associatively distributes entity copies into any number of rows, columns, and levels. MInsert (6.21.17) - inserts arrays of blocks. -BmParameters (6.8.19) - allows to list and edit the parameters of inserted components and associative array.

Figure 6.103



Misc	
Method	Divide
Items	15
Item spacing	305.714 cm
Start offset	200 cm
End offset	0 mm
Rows	1
Row spacing	246.339 cm
Row elevation increment	0 mm
Levels	1
Level spacing	750 mm
Align items	Yes

Figure 6.104

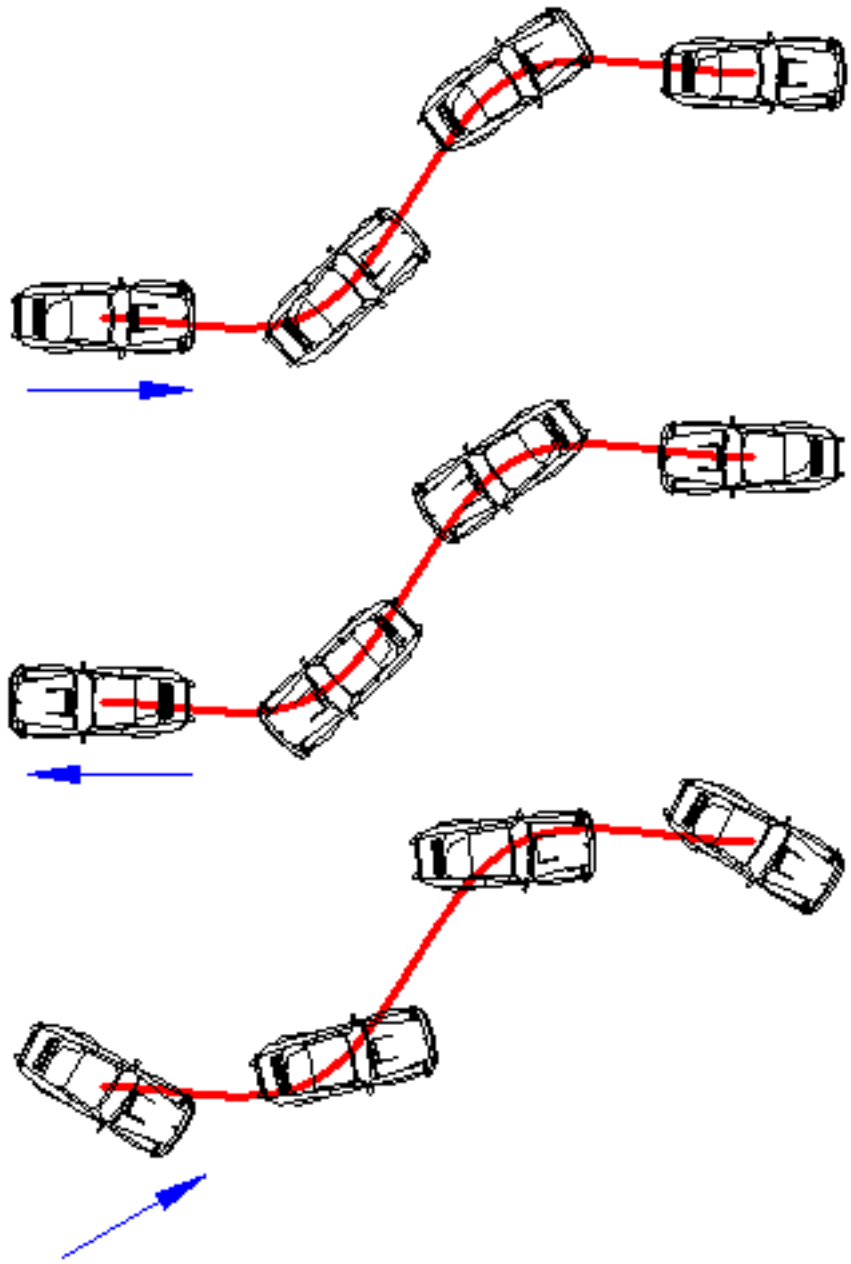


Figure 6.105

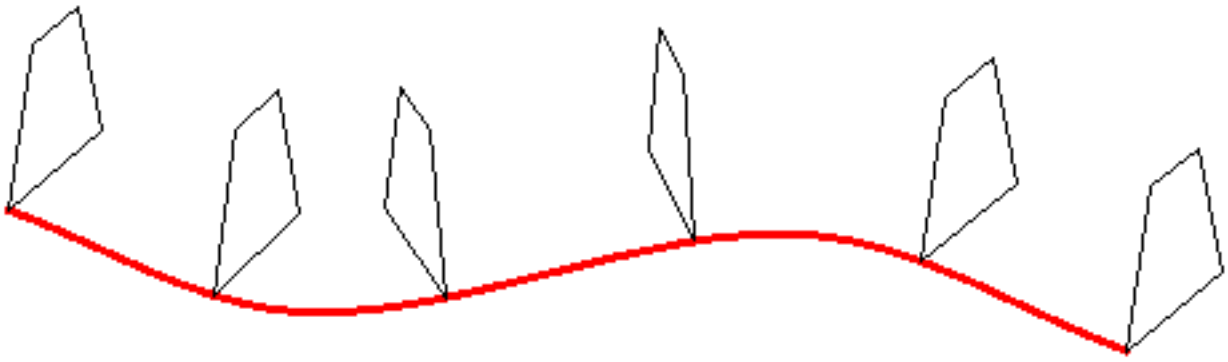


Figure 6.106

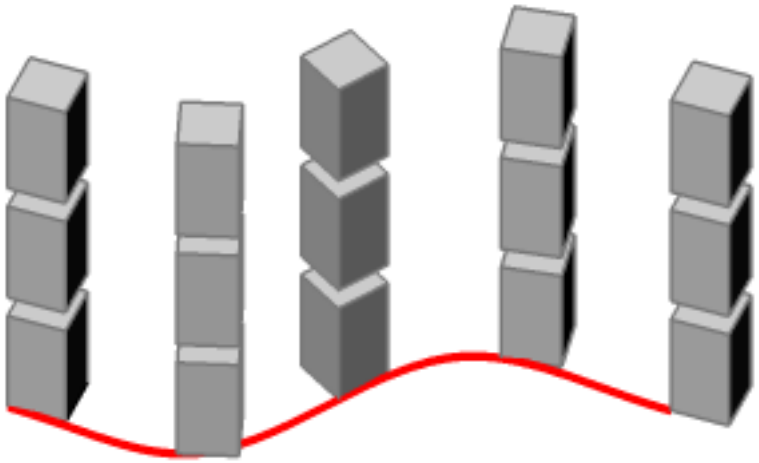


Figure 6.107

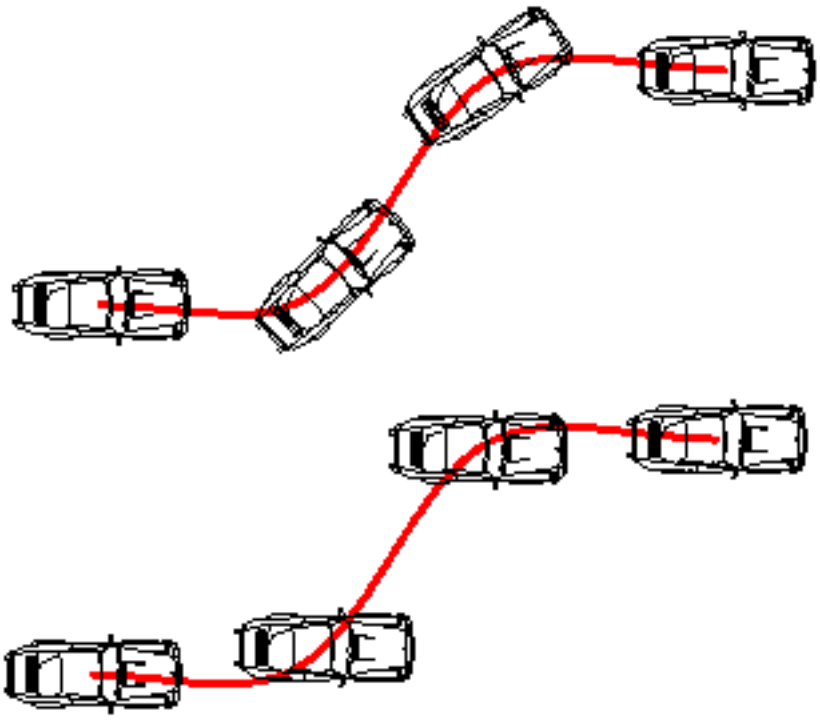


Figure 6.108

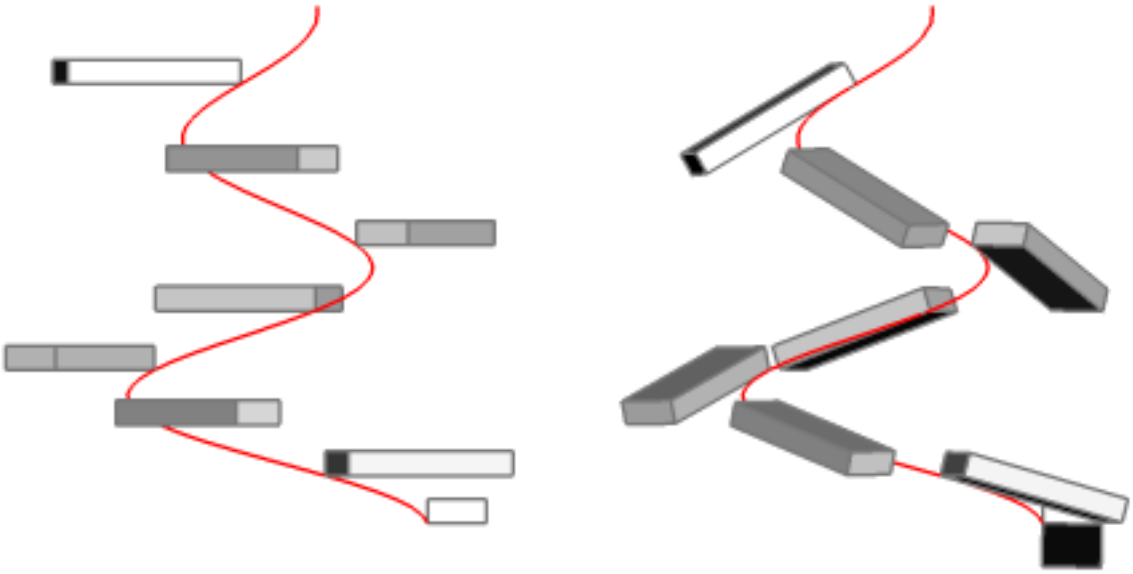


Figure 6.109

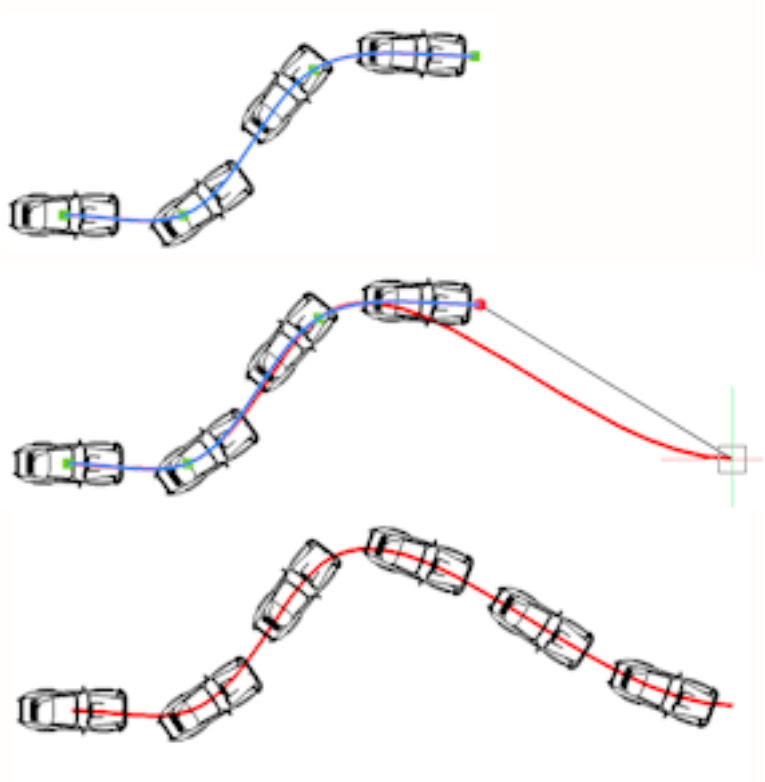


Figure 6.110

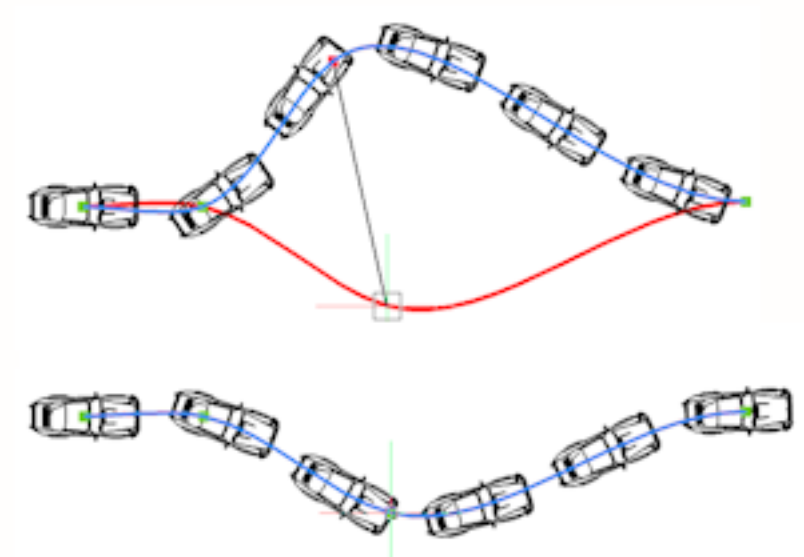


Figure 6.111



6.4.22 ArrayEditExt

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Provides additional editing ofassociative arrays:

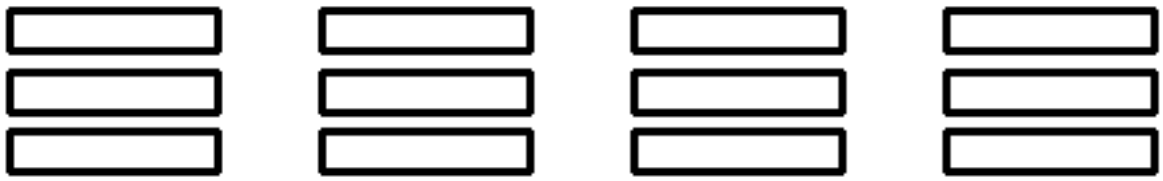
- For all elements of the array, interactively adjusts the spacing between them, and changes the number of elements
- For an individual element in the array, moves (offsets), scales, rotates, or deletes it.

Note: Arrays edited by this command must have been made associatively by the Array (6.4.16) command; it does not work with arrays created by the ArrayClassic (6.4.17)or-Array (6.4.15)commands.

Accessing the Command

command bar: **arrayeditext** : **arrayeditext Prompts you in the command bar:** Select option [SPacing/REsizing/Offset item/Scale item/Rotate item/Delete item]:**(Enter an option)** Select item in array:**(Select a single entity from an associative array)** Original associative array for the examples illustrated below: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyseg-1.png})

Figure 6.112



Command Options

Option	Description
SPacing	Adjusts the spacing of an entire associative array interactively; prompts you: 1. **Select item in array** - **(select one element of the array; the entire array will be re-spaced)** 2. **Enter radius and angle between items** - **(enter an option):** • Move the cursor to change the spacing and radius • Enter a distance value at the prompt (a value for the angle is ignored) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/arr eg-2.png})
REsizing	Resizes the entire array by adding an removing entities interactively; prompts you: 1. **Select item in array** - **(selectone element of the array)** 2. **Enter new number of items** - **(enter an option):** • Move the cursor to change the size • Enter a number for how many items the array should hold (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/arr eg-3.png})
Offset item	Moves one entity of an associative array to a different location; prompts you: 1. **Select item in array** - **(select the entity you wish to move)** 2. **Enter base point or center** - **(pick a point from which to move the entity, or enter x,y coordinates)** 3. **Enter displacement vector** - **(pick a point to which to move the entity„ or enter x,y coordinates)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/arr eg-4.png})

Scale item	Resizesan entity of an associative array; prompts you: 1. **Select item in array** - **(select the entity you wish to resize)** 2. **Enter base point or center** - **(specify the point from which the scaling takes place)**; the entity will move towards or away from the host array, depending on the location of the base point **Enter scale** - **(move the cursor, or enter a scale factor; numbers larger than 1 make the entity larger, numbers smaller than 1 make it smaller)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/arr eg-5.png})
Rotate item	Rotates an entity of an associative array; prompts you: 1. **Select item in array** - **(select the entity you wish to rotate)** 2. **Enter base point or center** - **(specify the point about which the rotation takes place; the entity will rotate around the host array, depending on the location of the base point)** 3. **Enter rotation angle**- **(move the cursor, or enter an angle; positive numbers rotate counter clockwise, negative numbers rotate clockwise)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/arr eg-6.png})
Delete item	Deletes an entity of an associative array; prompts you: **Select item in array** - **(erases the entity, without warning; use the U command to return it, if you erased it by accident)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/arr eg-7.png})

Related Commands

Array (6.4.16) - associatively distributes entity copies in an evenly spaced rectangular, polar or path array. Array-Classic (6.4.17) - creates a static polar or rectangular array of entities through a dialog box. ArrayClose (6.4.19) - exits the array editing state and displays a dialog box to save or discard the changes made to the array source entities.

Figure 6.113

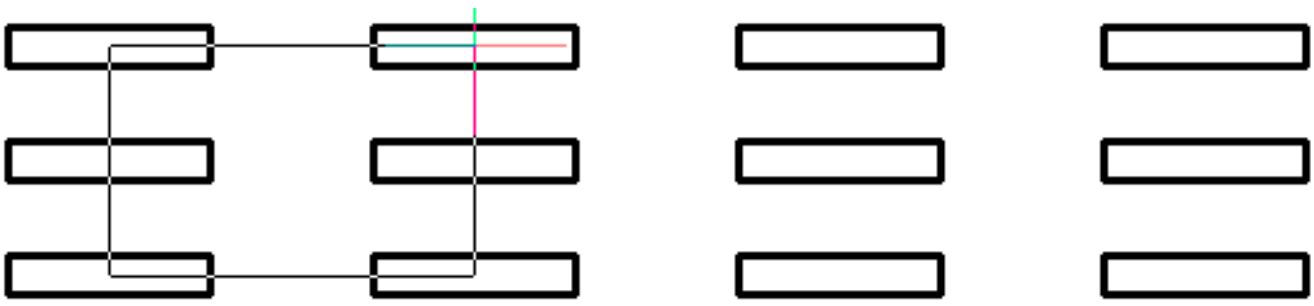


Figure 6.114

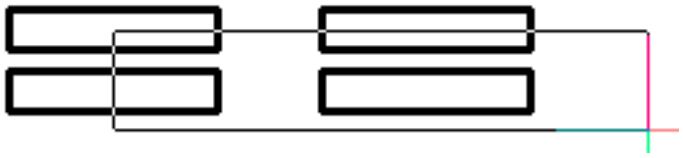


Figure 6.115

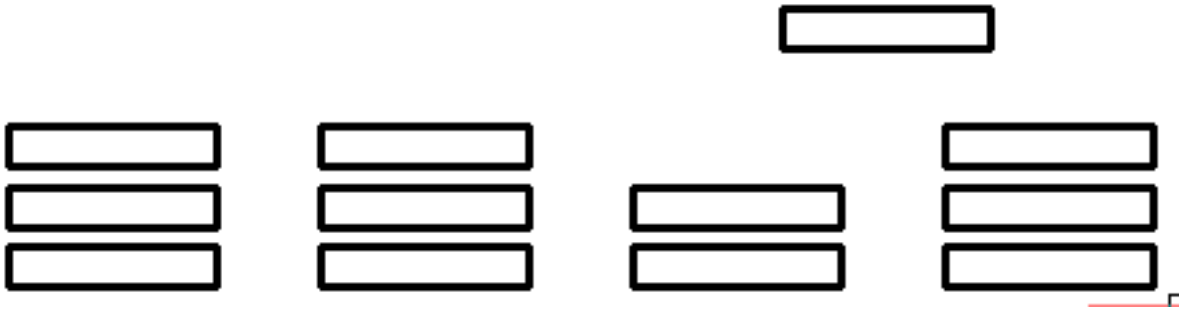


Figure 6.116

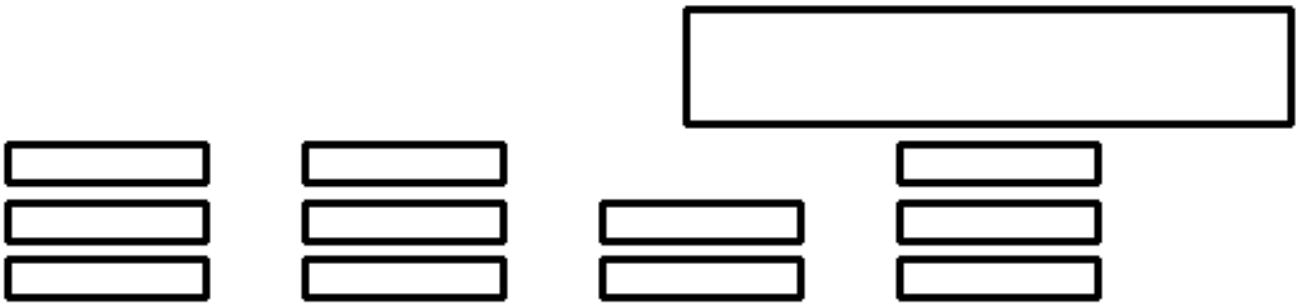


Figure 6.117

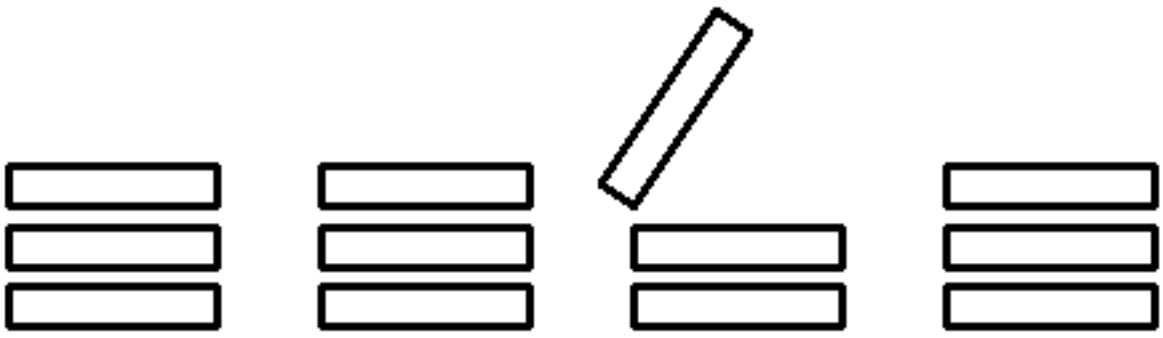


Figure 6.118

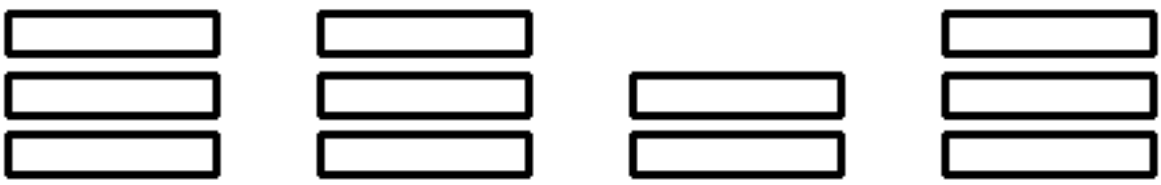


Figure 6.119



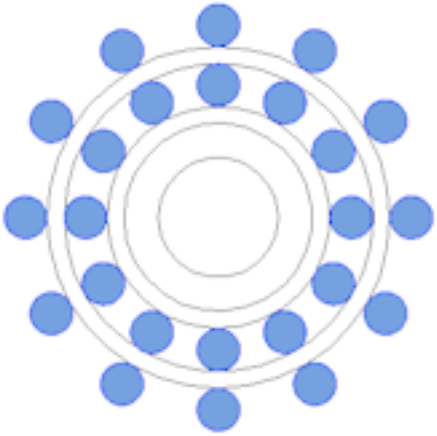
6.4.23 ArrayPolar

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.

Accessing the Command

command bar: **arraypolar** menu bar: **Modify | Array | Polar Array** toolbar: **Modify | Arrays |** ribbon:**Sketch | Arrange |** (Mechanical and SheetMetal workspaces) ribbon: **Solid | Edit |** (Mechanical and SheetMetal workspaces) ribbon:**Model | Modify 3D |** (BIM workspace) ribbon:**Change | Arrange |** (BIM and 3D Modeling workspaces) : **arraypolar Prompts you in the command bar:** Select entities to array: **(Select an entity)** Entities in set: 1 Select entities to array: **(Select more entities, or else press Enter to end entity selection)** Type = Polar, Associative = Yes Enter center point of array or [Base point/Axis of rotation]: **(Specify a point, or else choose an option)** [ASsociative/Base point/Items/Angle between/Fill angle/ROWs/Levels/ROTate/eXit]:<eXit>: **(Select an option, or else press Enter to end the command)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sourc

Figure 6.120



Command Options

Option	Description
Center point	Specifies the center point of the polar array. The axis of rotation is parallel to the Z-axis of the current coordinate system.
Axis of rotation	Specifies the axis of rotation by prompting you to pick two points.
Associative	Determines whether an array entity is created or a series of copies of the selected entities.The properties of array entities can be modified using the Arrayedit (6.4.20) command.The default setting is saved by the Arrayassociativity (1.13.11)system variable.Prompts you: **Create associative array? [Yes/No]<Yes>** - **(enter an option:)** • **Yes** - creates an associative array • **No**- creates a non-associative array, which is like a series of copies made in a round pattern The associative arrays parameters can be specified using a parametric expressions.Expressions can be set in the array properties in bottom part of the Mechanical Browser (3.2.2), Properties panel (6.24.49) or via the ArrayEdit (6.4.20)command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P
Base point	Sets the base point of the array, so that items are placed relative to the base point.Prompts you: **Enter base point** - **(specify a point in the drawing)**
Items	Specifies the number of items in the array.Prompts you: **Enter number of items in array <xx> -(type a value or press Enter to accept the current number.)**
Angle between	Specifies the angle between two subsequent items.Prompts you: **Enter angle between items <xx.xx>** - **(type a value, or else press Enter to accept the current angle)**
Fill Angle	Specifies the angle between the first and the last item.Prompts you: **Enter the angle to fill (+=ccw, -=cw) <xx.xx> -(type a value, or else press Enter to accept the current angle)**
Rows	Specifies the number of rows and the spacing between subsequent rows.Prompts you: 1. **Enter the number of rows <xx>** - (**type a value or press Enter to accept the current number)** 2. **Enter the distance between rows or [Total] <xx.x>** - (**enter an option:)** • **Distance** - specifies the distance between subsequent rows • **Total** : specifies the distance* between the first and the last row 3. **Enter the incrementing elevation between rows <0.0>** - **(defines the increasing or decreasing elevation for each subsequent row.)**

Levels	Creates 3D arrays by specifying the number and spacing of levels. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/A Prompts you: 1. **Enter the number of levels <1>** - (**specify the number of levels.**) 2. **Enter the distance between levels or [Total] <xxx>** - **(enter an option:)** • **Distance**- specify the distance* between subsequent levels bytyping a value or picking two points. • **Total**- specify the distance* between the bottom and the top level.
Rotate	Determines whether items in the array are rotated.Prompts you: **Rotate arrayed items? <Yes>/No** - **(enter an option:)** • **Yes** - rotates items around the circular path • **No**- preserves the orientation of the source entities
Exit	Accept the current settings and conclude the command.

Related Commands

3dArray (6.3.2) - creates 3D arrays. -Array (6.4.15) - creates a polar or rectangular array of entities; displays prompts at the command bar. Array (6.4.16) - associatively distributes entity copies in an evenly spaced rectangular, polar or path array. ArrayClassic (6.4.17) - creates a static polar or rectangular array of entities through a dialog box. ArrayEdit (6.4.20) - allows editing associative array entities and their source entities. ArrayPath (6.4.21) - associatively distributes entity copies evenly along a path into multiple rows, and levels. ArrayRect (6.4.25) - associatively distributes entity copies into any number of rows, columns, and levels. MInsert (6.21.17) - inserts arrays of blocks. -BmParameters (6.8.19) - allows to list and edit the parameters of inserted components and associative array.

Figure 6.121

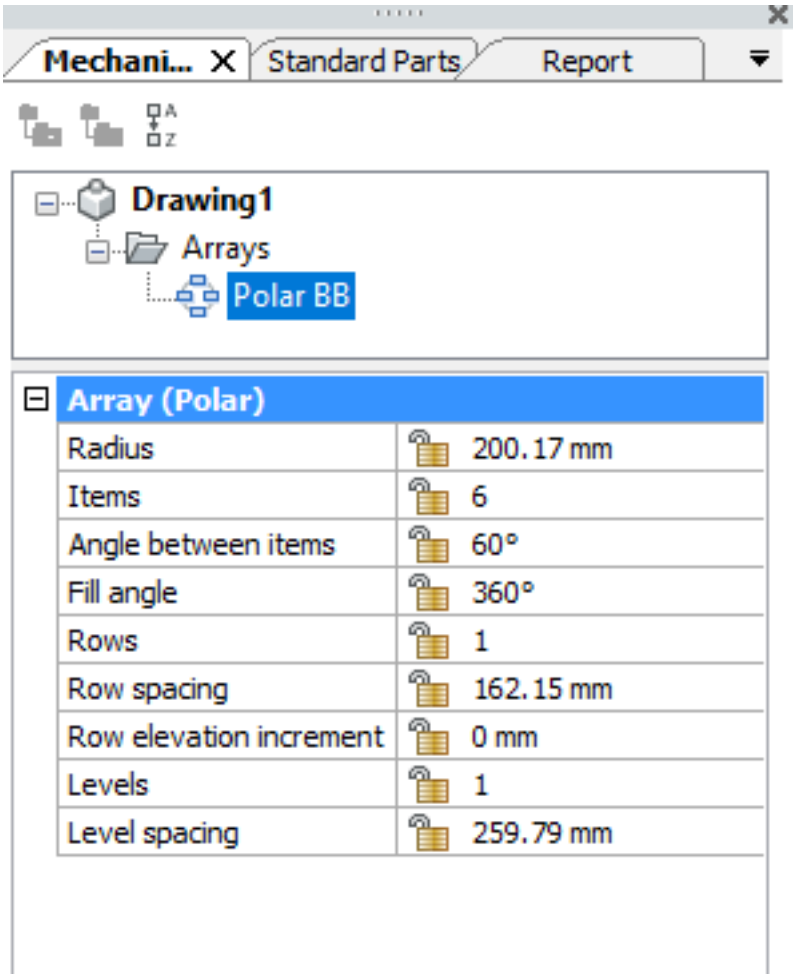


Figure 6.122

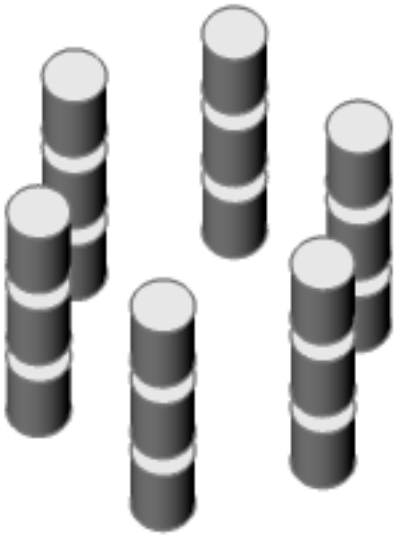


Figure 6.123



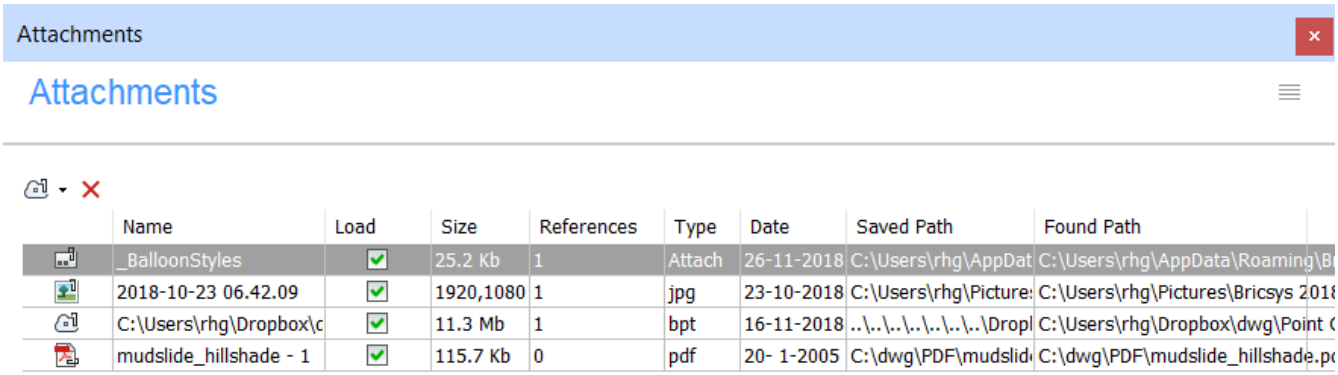
6.4.24 AttachmentsPanelOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the Attachments panel for managing external references (DWG), raster images, PDF files, and point clouds that are attached to the current drawing.

Accessing the Command

command bar:**attachmentspanelopen** : **attachmentspanelopen** Displays the Attachments panel with a list of files attached to the current drawing, if any. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/panel.png})

Figure 6.124



Command Options

Options	Description
Attach DWG	Displays the Open File to Attach dialog box like the -XRef (6.34.9) command
Attach Image	Displays the Select Image File dialog box like the ImageAttach (6.4.26) command
Attach PDF	Displays the Select PDF Underlay File dialog box like the PDFAttach (6.24.13) command
Attach Point Cloud	Displays the Preprocessed Point Cloud Data File dialog box like the PointCloudAttach (6.24.34) command
Detach	Detaches the selected attachment from the drawing
Load	Toggles the display of the attachment in the drawing; turn off to hide the attachment

Procedures

To open or close a panel (1.12.7)

Related Commands

AttachmentsPanelClose (6.4.26)- closes the Attachments panel. ImageAttach (6.34.9) - attaches raster images to the current drawing. PDFAttach (6.24.13) - attaches PDF files to the current drawing. PointCloudAttach (6.24.34) - attaches BPT processed point cloud files to the current drawing. - (6.34.9)XRef (6.4.26) - attaches drawing files to the current drawing.

Figure 6.125



6.4.25 ArrayRect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Associatively distributes copies of entities into any number of rows, columns, and levels.

Accessing the Command


command bar: **arrayrect** menu bar: **Modify | Array | Rectangular Array** toolbar: **Arrays** ribbon:

- **Sketch | Arrange**(Mechanical and SheetMetal workspaces)
- **Solid | Edit**(Mechanical and SheetMetal workspaces)
- **Model | Modify 3D**(BIM workspace)
- **Change | Arrange**(BIM and 3D Modeling workspaces)

: **arrayrect Prompts you in the command bar:** Select entities to array: **(Select an entity)** Entities in set: 1 Select entities to array: **(Select more entities, or else press Enter to end entity selection)** Type = Rectangular, Associative = Yes [ASsociative/Base point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>: **(Enter an option, or else press Enter to end the command)**

Command Options

Option	Description
Associative	Determines whether an array entity is created or a series of copies of the selected entities.The default setting is saved by the ArrayAssociativity (1.13.11)system variable.Prompts you: **Create associative array? [Yes/No]<Yes>** - **(enter an option):** • **Yes**- creates an associative array, which can be edited later with the ArrayEdit (6.4.20)command • **No** - creates a non-associative array, which consists of copies of the entities laid out in a pattern The associative arrays parameters can be specified using a parametric expressions.Expressions can be set in the array properties in bottom part of the Mechanical Browser (3.2.2), Properties panel (6.24.49) or via the ArrayEdit (6.4.20)command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/F
Base point	Sets the base point of the array. Prompts you: **Enter base point** - **(specify a point in the drawing.)** Items are placed relative to the base point.
Count	Specifies the number of rows and columns.Prompts you: 1. **Enter the number of columns <xx>** - **(type a value, or else press Enter to accept the current number.)** 2. **Enter the number of rows <xx>** - (**type a value, or else press Enter to accept the current number)**

Spacing	Specifies the spacing between columns and/or rows.Prompts you: **Enter the distance between columns or [Unit cell] <xx.x>** - **(type a value, or else press Enter to accept the current spacing)** Unit cell option prompts you: 1. **Select first point of unit cell** - **(Specify a point).**A rectangle is displayed dynamically. (If dynamic input is on, then the width and height of the spacing rectangle are displayed.) 2. **Opposite corner** - **(specify a point, or else enter the desired column and row spacings in the dynamic entry fields.Press the Tab key to switch back and forth between the entry fields, and then press Enter to confirm the values.)**
Columns	Specifies the number of columns and the spacing between subsequent columns.Prompts you: 1. **Enter the number of columns <xx>** - **(type a value, or else press Enter to accept the current number.)** 2. **Enter the distance between columns or [Total] <xx.x>** - **(enter an option:)** • **Distance** - specifies the distance between subsequent columns • **Total** - specifies the distance between the first and the last column
Rows	Specifies the number of rows and the spacing between subsequent rows.Prompts you: 1. **Enter the number of rows <xx>** - **(type a value, or else press Enter to accept the current number.)** 2. **Enter the distance between rows or [Total] <xx.x>** - **(enter an option)**: • **Distance**- specify the distance between subsequent rows • **Total** - specifies the distance between the first and the last row 3. **Enter the incrementing elevation between rows <0.0>** - **(specifies the increasing or decreasing elevation for each subsequent row.)**

Levels	Creates 3D arrays by specifying the number and spacing of levels. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/A Prompts you: 1. **Enter the number of levels <1>** - **(specifies the number of levels)** 2. **Enter the distance between levels or [Total] <xxx>** - **(enter an option):** • **Distance**- specifies the distance between subsequent levels bytyping a value or picking two points. • **Total**- specifies the distance between the bottom and the top level
Exit	Accept the current settings and conclude the command.

Related Commands

3dArray (6.3.2) - creates 3D arrays. -Array (6.4.15) - creates a polar or rectangular array of entities; displays prompts at the command bar. Array (6.4.16) - associatively distributes entity copies in an evenly spaced rectangular, polar or path array. ArrayClassic (6.4.17) - creates a static polar or rectangular array of entities through a dialog box. ArrayEdit (6.4.20) - allows editing associative array entities and their source entities. ArrayPath (6.4.21) - associatively distributes entity copies evenly along a path into multiple rows, and levels. ArrayPolar (6.4.23) - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. MInsert (6.21.17) - inserts arrays of blocks. -BmParameters (6.8.19) - allows to list and edit the parameters of inserted components and associative array.

Figure 6.126

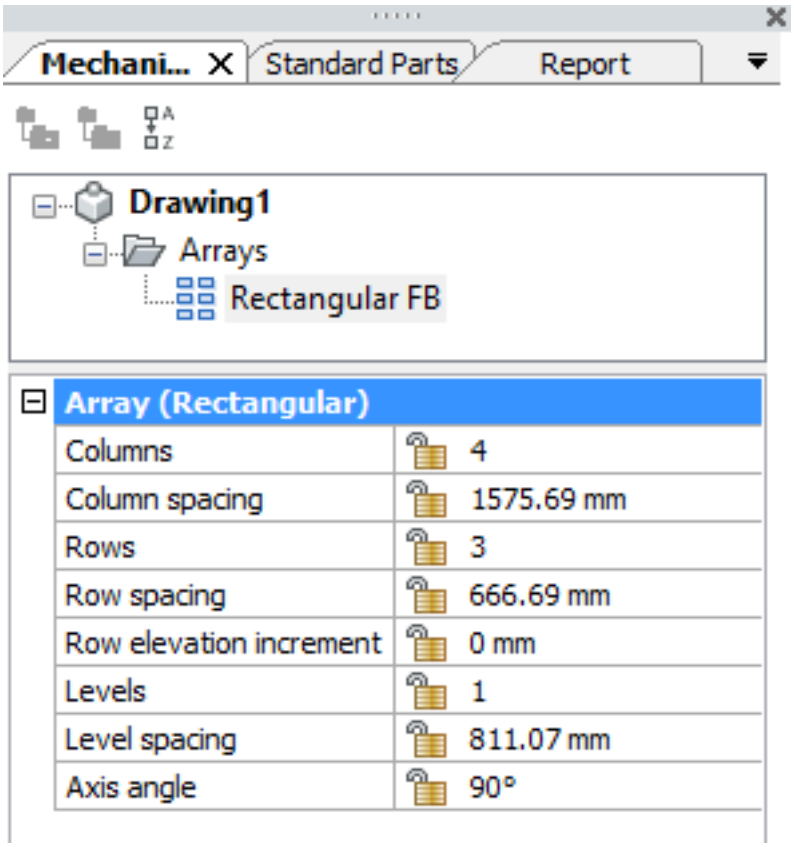


Figure 6.127

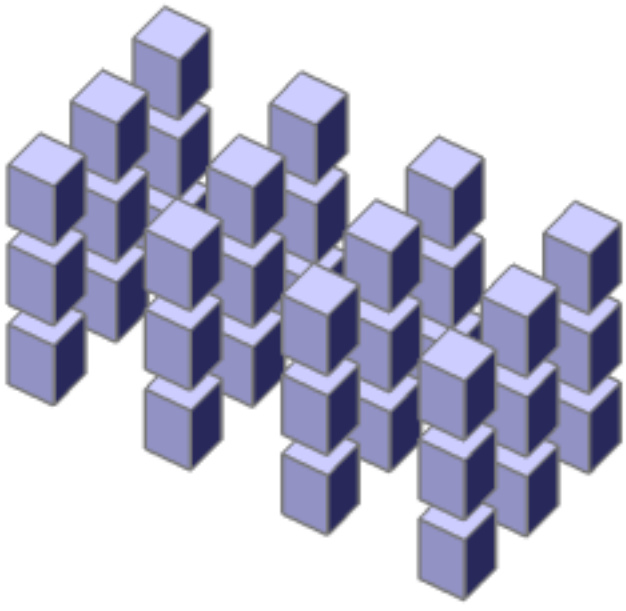


Figure 6.128



6.4.26 AttachmentsPanelClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes the Attachments panel.

Accessing the Command

command bar: **attachmentspanelclose** : **AttachmentsPanelClose** BricsCAD closes the Attachments panel. Reopen it with the AttachmentsPanelOpen command.

Related Commands

AttachmentsPanelOpen (6.4.24) - opens the Attachments panel to control DWG, PDF, image, and point cloud attachments.

Figure 6.129



6.4.27 AttDef

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Defines how a block's attributes will be created through a dialog box.

Accessing the Command

 command bar: **attdef** aliases: **at**, **ddattdef** menu bar: **Tools | Attributes | Define Attributes** toolbar: **Attributes** |  ribbon:**Insert | Blocks | Define Attributes** : **attdef** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc dlg.png}) Enter options, and then click **OK**.To create and use attributes, follow these steps:

1. Use this command to define attributes.
2. Use the Block (6.6.11) command to combine attributes and symbols into a block.
3. Use the Insert (6.17.13) command to insert the blocks, answering the prompts with data with attributes.
4. When it is necessary to edit the values of the attributes, use the AttEdit (6.4.36) command.
5. To extract attribute values to external data files, use the AttExt (6.4.28)or DataExtraction (6.10.1) commands.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image49.jpg}) You can use the AttDisp (6.4.35) command to toggle the display of attributes in drawings on and off.

Figure 6.130

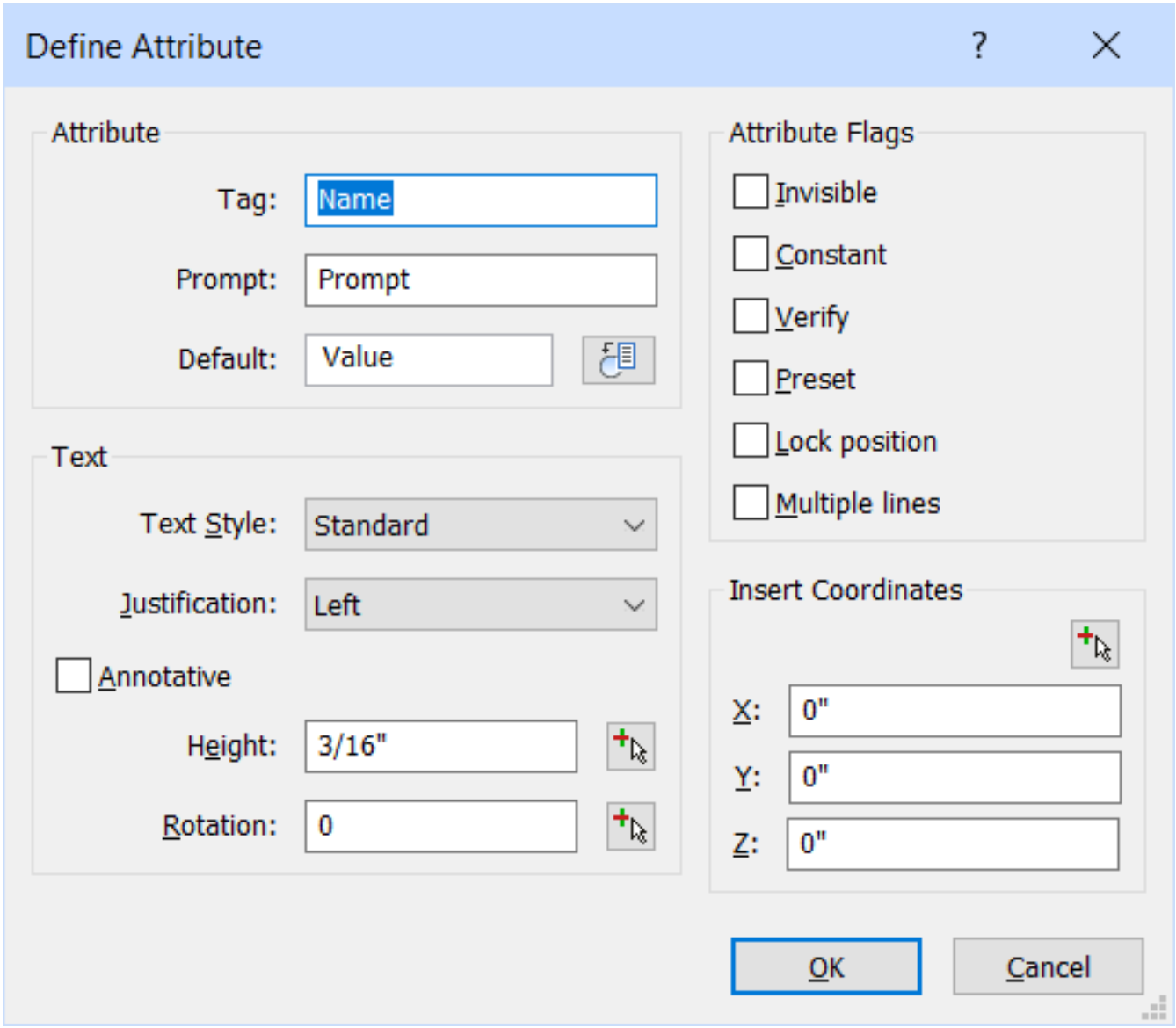
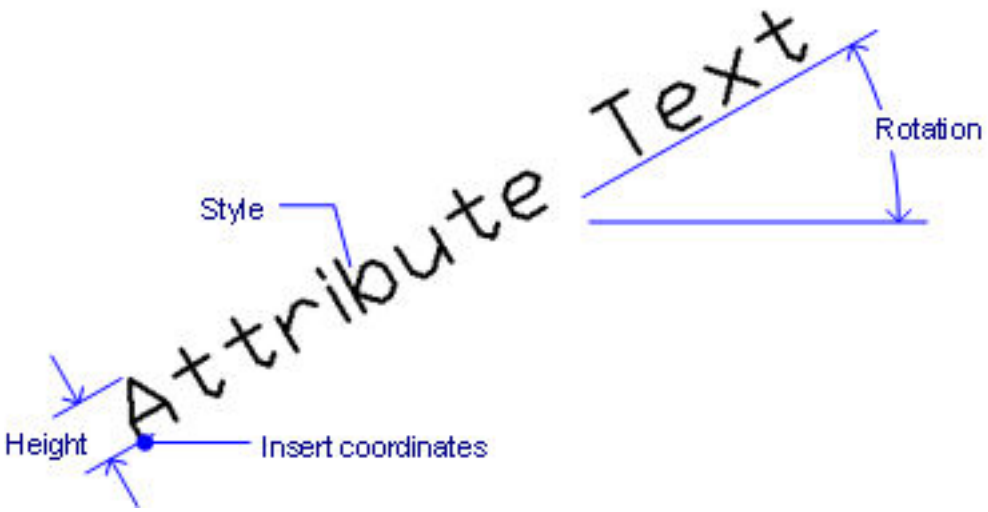


Figure 6.131



Command Options

Attribute Options	Description
Tag	Specifies the name of the attribute; this is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers, and punctuation.
Prompt	Specifies the user prompt; this is displayed at the command prompt when the attribute is later inserted in the drawing. You can leave this field blank; BricsCAD uses the tag as the prompt during attribute insertion.
Default	Specifies the default value; this is displayed in angle brackets, like <360>. As with other defaults, the user need only press Enter to accept this value. Click the **Insert Field** button () to assign a field value to the attribute (see the FIELD (6.14.3) command).

Text Options	Description
Text Style	Specifies the text style for the attribute text; choose from one of the styles defined in the drawing. (You can create additional text styles with the **Style (6.27.53)** command.)
Justification	Specifies the justification of the attribute text, such as Center or Fit.
 Height	Specifies the height of the attribute text, unless overridden by the text style. Enter a value, or click the button to pick two points in the drawing: 1. Dismisses the dialog box temporarily. 2. Prompts you at the command prompt**Select text height** - enter a value, or pick two points in the drawing
 Rotation	Specifies the rotation angle of the rotation text. Enter a value, or click the button to pick two points in the drawing: 1. Dismisses the dialog box temporarily. 2. Prompts you at the command prompt**Select text angle** - enter an angle, or pick two points in the drawing.
Annotative	Sets the Annotative property of the attribute text.

Attribute Flags Options	Description
Invisible	Hides the attributes from view; they are not displayed and not printed. Hidden attributes can, however, be displayed with the **AttDisp (6.4.35)**command.
Constant	Specifies a default value that the user cannot change.
Verify	Forces the user to enter the value a second time; this helps to ensure the value is entered correctly.
Preset	Inserts attributes without prompting the user; the attributes can be changed later with the **AttEdit (6.4.36)** command.

Lock Position	**On** - locks the position of the attribute within the block reference. **Off** - unlocked attributes can be moved relative to the rest of the block using grip editing.
Multiple lines	**On** - allows the use of multiline text **Off** - limits text to a single line When the **Multiple lines** flag is checked, the button (...) displays next to the dimmed **Default** text field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/D When you click the button, the **Define Attribute** dialog box temporarily closes to let you define a multiline text entity. Prompts you: **Specify location of multiline attribute** - specify a point, and then the MTEXT (6.21.35) command launches.Type the text for the attribute, then click the **OK** button on the Text Formatting toolbar to return to the Attribute Definition dialog box.

Insert Coordinates Options	**Description**
Insert Coordinates	Dismisses the dialog box temporarily, and then prompts you at the command prompt: **Select insertion point** - enter coordinates, or pick a point in the drawing.
X	Specifies the x coordinate of the attribute's insertion point.
Y	Specifies the y coordinate.
Z	Specifies the z coordinate.

Procedures

- Defining attributes (1.22.11)

Related Commands

-AttDef (6.4.34) - defines attributes through the command bar. AttDisp (6.4.35) - toggles the display of attribute text. AttEdit (6.4.36) - edits attribute values and properties. -AttExt (6.4.29) - extracts attributes from drawings to data files through the command bar. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. DataExtraction (6.10.1) - exports entity properties, block attributes and drawing information to a an external file. DdAtte (6.10.13) - edits the values of attributes through a dialog box. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Field (6.14.3) - inserts a field in a multiline text, table cell or attribute. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. InsertAligned (6.17.15) - inserts blocks repeatedly, and inserts mirrored blocks. Properties (6.24.49) - edits attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place.

Figure 6.132

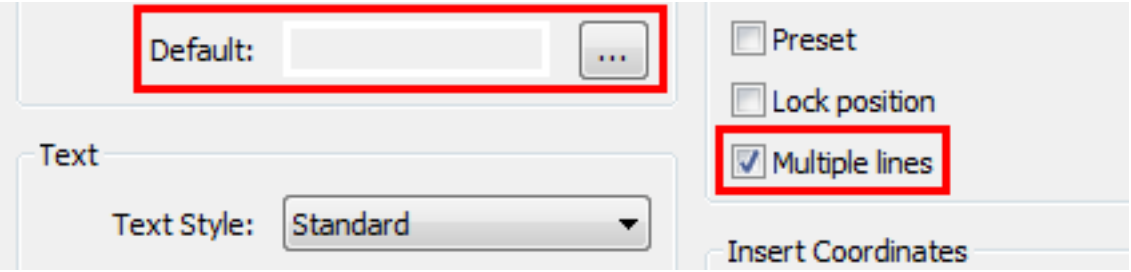


Figure 6.133



6.4.28 AttExt

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Copies data from attributes to a text file via a dialog box.Deprecated: use instead theDataExtraction (6.10.1)command.

Accessing the Command

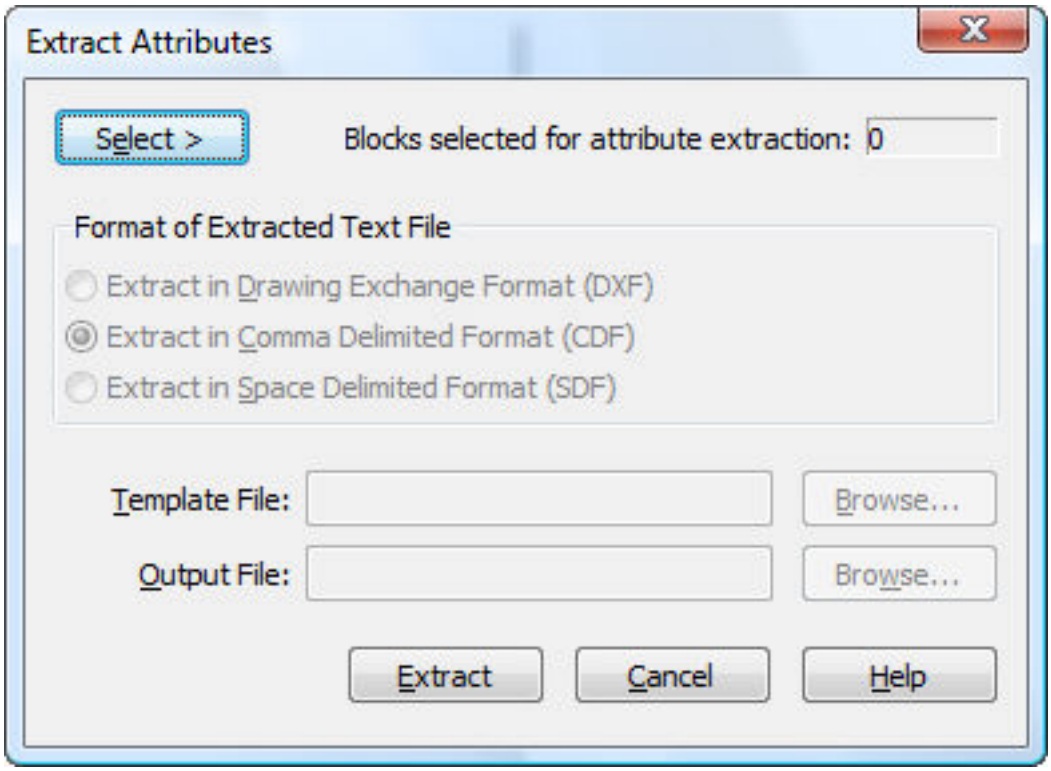
 command bar: **attext** aliases: **ax, ddattext** menu:**Tools | Attributes | Extract Attributes** toolbar:**Tools | Attributes** |  ribbon:**Insert | Blocks | Extract Attributes** : **attext** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image35.jpg}) Enter options, and then click **Extract** . If you select blocks without attributes and non-block entities, BricsCAD ignores them. The template file determines which attributes are extracted and the format of related data. For example, a **template.txt** file contains the following fields: BL:NAME C008000 BL:NUMBER N003000 PRODUCT C016000 BL:X N016004 BL:Y N016004 This means that the AttExt command will extract the following values:

- BL:NAME C008000 -Name of the block formatted to 8 characters.
- BL:NUMBER N003000 -Count (number) of the block, formatted to a 3-digit number.
- PRODUCT C016000 -Values of the attribute with the ”Product” tag, formatted to 16 characters.
- BL:X N016004 and BL:Y N016004 -The x and y coordinates of the block’s insertion point, formatted to 16 digits, including 4 decimal places.

The table below lists all available data extraction parameters. The table uses the follow codes: C = indicates alphanumeric field N = indicates numeric field fff = indicates width of field, in characters (max = 999) ddd = indicates number of decimal places (max = 999) 0 = indicates zero.

Data	**Format**
Attribute tag (numeric)	Nfffddd
Attribute tag (alphanumeric)	Cfff000
Block name	BL:NAME Cfff000
Block counter	BL:NUMBER Nfff000
Block handle (identifier)	BL:HANDLE Cfff000
Block’s layer name	BL:LAYER Cfff000
Block nesting level	BL:LEVEL Nfff000
Block insertion, x coordinate	BL:X Nfffddd
Block insertion, y coordinate	BL:Y Nfffddd
Block insertion, z coordinate	BL:Z Nfffddd
Block’s rotation angle	BL:ORIENT Nfffddd
Block’s x scale factor	BL:XSCALE Nfffddd
Block’s y scale factor	BL:YSCALE Nfffddd
Block’s z scale factor	BL:ZSCALE Nfffddd
Block’s x extrusion direction	BL:XEXTRUDE Nfffddd
Block’s y extrusion direction	BL:YEXTRUDE Nfffddd
Block’s z extrusion direction	BL:ZEXTRUDE Nfffddd

Figure 6.134



Command Options

Option	Description
Select	Dismisses the dialog box temporarily so that you can select blocks with attributes: **Select entities** - **(choose one or more blocks with attributes.) Select entities** - **(press Enter to return to the dialog box.)**
Extract in Drawing Exchange Format (DXF)	Exports the attribute values in DXF format as .dxx files.This format is meant for use with other computer programs that can read .dxf files.
Extract in Comma Delimited Format (CDF)	Exports the attributes with commas separating values.This format is meant for use with spreadsheets and databases.
Extract in Space Delimited Format (SDF)	Exports the attributes in fixed-width format.This format is meant for use in word processors.
Template File	Specifies the name of the .txt template file.The file must exist before this command is used.
Browse	1. Displays the Attribute Extract Template File dialog box. Choose a file. 2. Click **Open**.
Output File	Specifies the name of the output file,
Browse	1. Displays the Attribute Extract Template File dialog box. Specify a file name. 2. Click **Save**.
Extract	Exports the attribute values to the output file.

Procedures

- Extracting attribute information (1.22.15)

Related Commands

Figure 6.135



6.4.29 -AttExt

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Copies data from attributes to a text file via the command line.

Accessing the Command

command bar: **-attext** alias: **-at** The template .**txt** file must exist before this command is used. See the AttExt (6.4.28) command for the format of this file. : **-attext You prompted:** Select Entities or extract attributesas [Entities/Dxf/Cdf/Sdf] <CDF>: **(Enter an option or press Enter.)** BricsCAD displays the Attribute Extract Template File dialog box.Choose a template .**txt** file, and then click **Open**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V template-dlg.png}) BricsCAD displays the Attribute Extract Output File dialog box. Specify a file name, and then click **Save**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/attext-output-dlg.png}) The attribute values and other data are extracted, and saved to the output file. BricsCAD then reports the number extracted: **15 records extracted.** If you select blocks without attributes and non-block entities, BricsCAD ignores them. If you ignore this option by entering DXF, SDF, or CDF directly, then BricsCAD selects all attributes in the drawing.

Figure 6.136

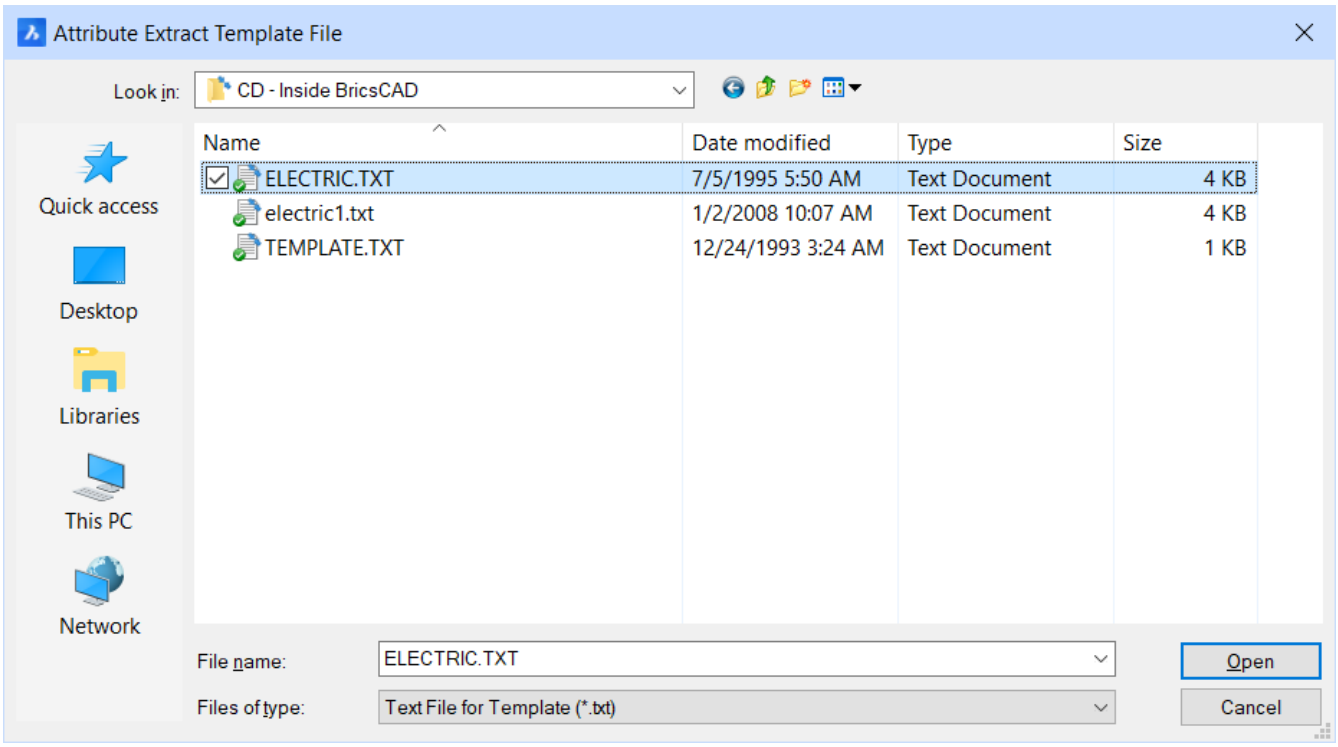
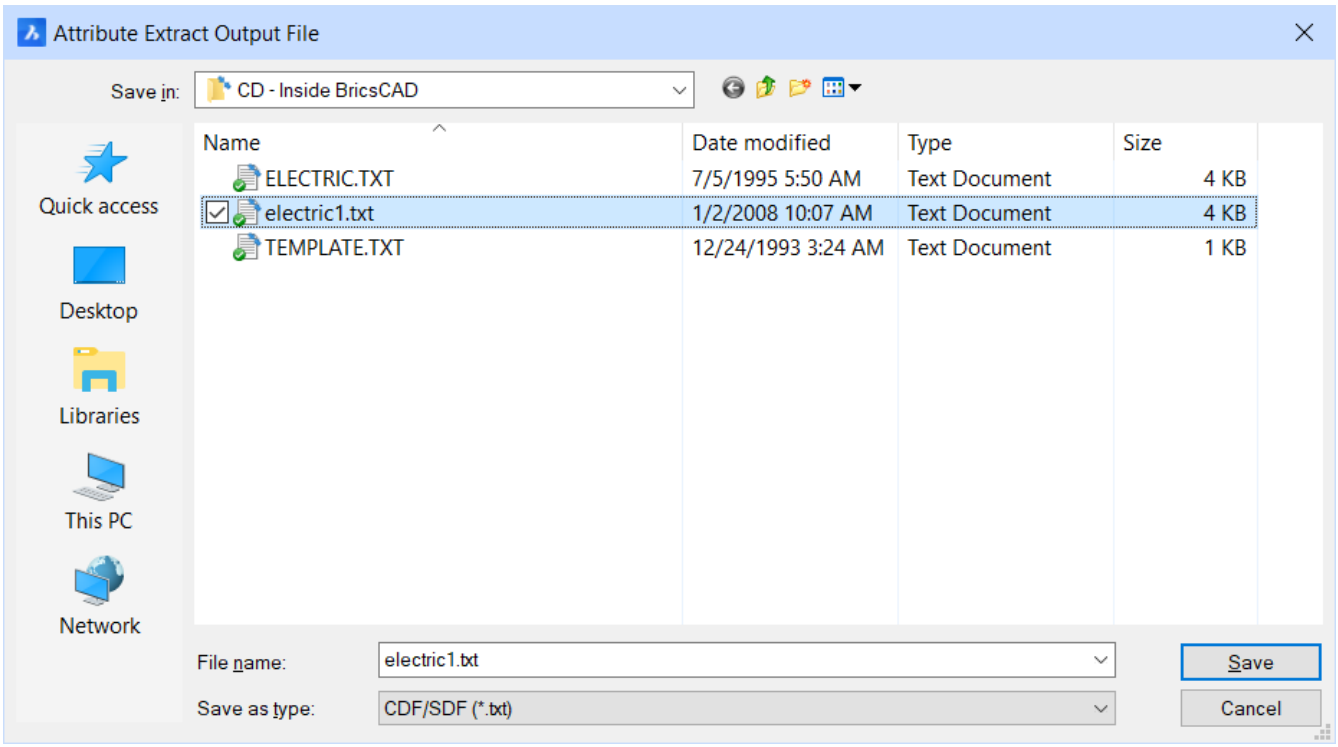


Figure 6.137



Command Options

Option	Description
Select Entities	Prompts you to select entities. Enter E for the following prompts: **Select entities** - **(choose one or more blocks with attributes)**. **Select entities** - **(press Enter to continue the command)**. If you ignore this option by entering DXF, SDF, or CDF directly, then BricsCAD selects all attributes in the drawing.
Extract attributes as: DXF/CDF/SDF	Specifies the data format: • **DXF** - exports the attribute values in a partial DXF format; this format is meant for use with other computer programs that can read .dxx files (DXf eXtraction). • **CDF** - exports the attributes with commas separating values; this format is meant for use with spreadsheets and databases. • **SDF** - exports the attributes in fixed-width format; this format is meant for use in word processors.

Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttDisp (6.4.35) - toggles the display of attribute text. AttEdit (6.4.36) - edits attribute values and properties. AttExt (6.4.28) - (deprecated) extracts attributes from drawings to data files through a dialog box. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. DataExtraction (6.10.1) - exports entity properties, block attributes and drawing information to a an external file. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. Properties (6.24.49) - edits attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place.

Figure 6.138



6.4.30 AttRedef

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Redefines a block and updates associated attributes.

Accessing the Command

command bar: **attredef** menu bar:**Tools | Attributes | Redefine Attribute Block** : **attredef Prompts you in the command bar:** Enter name of block you wish to redefine: **(Type the name of an existing block)** Select entities for block: **(Select entities and attributes to be included in the block definition)** Select entities for block: **(Select more entities, or else press Enter to stop selecting entities)** Specify new base point: **(Specify the origin point of the block)** The following rules apply to existing block references:

- New attributes use default values.
- Values of old attributes that are included in the new block definition are preserved.
- Old attributes that are not included in the new block definition are deleted.
- Format or property changes made with the ATTEDIT (6.4.36) or EATTEDIT (6.13.1) commands are removed.
- Extended data associated with the block are deleted.

Command Options

Option	Description
Enter name of block you wish to redefine	Specify the name of a block
Select entities for block	Select entities (and optionally the attributes) that should become part of the block
Specify new base point	Pick a point or enter coordinates

Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttDisp (6.4.35) - toggles the display of attribute text. AttEdit (6.4.36) - edits attribute values and properties. -AttExt (6.4.29) - extracts attributes from drawings to data files through the command bar. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. DataExtraction (6.10.1) - exports entity properties, block attributes and drawing information to a an external file. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. InsertAligned (6.17.15) - inserts blocks repeatedly, and inserts mirrored blocks. Properties (6.24.49) - edits attribute values and properties.

Figure 6.139



6.4.31 AttSync

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Synchronizes attribute definitions in all block references of a specified block definition.

Accessing the Command

command bar: **attsync** menu bar:**Tools | Attributes | Synchronize Attributes** : **attsync Prompts you in the command bar:** Enter an option [Select/Name/?]<Select>: **(Press Enter or choose an option)** Select a block: **(Select a block in the drawing)** ATTSYNC block? [Yes/No]<Yes>: **(Press Enter to synchronize all instances of the selected block)**

Command Options

Option	Description
Select	Selects the block whose attributes should be used as the template for synchronizing other blocks. Prompts you: **Select a block** - **(select a block in the drawing; skips the next prompt).**
Name	Selects the block definition to be synchronized.Prompts you: **Enter name of block to sync or ? to list attributed blocks** - **(type the name of a block containing attributes, or enter ? to display a list block names that contain attributes in the current drawing).**
Select a block	This prompt appears when you start with the Name option; it mimics the Select option.
ATTSYNC block?	Synchronizes attributes in the specified blocks: **Yes** -synchronizes attributes in the block **No** - does not synchronize the block

Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. A (6.4.35)ttDisp (6.4.35)- toggles the display of attribute text. AttEdit (6.4.36) - edits attribute values and properties. -AttExt (6.4.29) - extracts attributes from drawings to data files through the command bar. AttRedef (6.4.30) - redefines a block and updates associated attributes. BattMan (6.6.2) - manages the attributes of a block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. DataExtraction (6.10.1) - exports entity properties, block attributes and drawing information to a an external file. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. InsertAligned (6.17.15) - inserts blocks repeatedly, and inserts mirrored blocks. Properties (6.24.49) - edits attribute values and properties.

Figure 6.140



6.4.32 Audit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Repairs drawings that are open.

Accessing the Command

 command bar: **audit** transparent: **'audit** menu bar: **File | Drawing Utilities | Audit** ribbon:**Home | File | Audit** : **comandName Prompts you in the command bar:** Fix any errors detected? [Yes/No] <No>: **(Type Y or N)** 55 objects audited Total errors found during audit 0, fixed 0

Command Options

Option	Description
Yes	Fixes errors, if BricsCAD finds any.
No	Searches for errors in the drawing database, but does not repair them.

Procedures

- To analyze the integrity of the current drawing (1.14.9)
- To repair a drawing (**??**)

Related Commands

ProxyInfo (6.24.51) - displays a information about proxy entities, contained in the current drawing, in a dialog box. Recover (6.26.3) - recovers damaged drawings. RecoverAll (6.26.4) - recovers a drawing with all nested XRef drawings, each drawing is saved with .BAK copy and .ADT report.

Figure 6.141



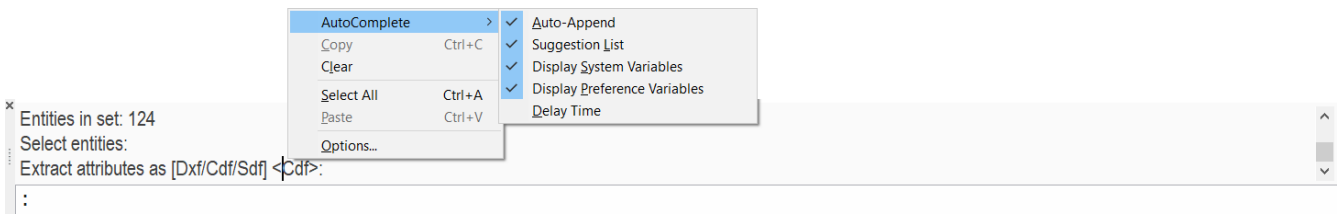
6.4.33 AutoComplete

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Specifies how the auto-complete function works in the command bar.

Accessing the Command

command bar: **autocomplete** transparent:**'autocomplete** context menu: **Right-click the command bar** : **autocomplete** Prompts you in the command bar: Enter an automatic completion option [Append/List/System variables/Delay/ON/OFf] <ON>: **(Choose an option.)** The auto-complete options are saved in the AutoCompleteMode system variable. Alternatively, you can right-click the command bar, and then choose options from the AutoComplete item in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/autocomplete-context-menu.png})

Figure 6.142



Command Options

Option	Description
Append	Prompts you: **Automatically append suggestions as you type? [Yes/No] <Yes> -(enter Y or N)**: • **Yes**- automatically completes the command as you type • **No**- does not complete the command
List	Prompts you: **Display a list of suggestions? [Yes/No] <Yes>** - **(enter Y or N)**: • **Yes**- display a list of command name and other suggestions • **No**- do not display suggestions
System variables	Prompts you: **Include system variables in suggestions? [Yes/No] <Yes>** - **(enter Y or N)**: • **Yes- includes system variables anduserpreferences in the list of suggestions** • **No** - does not include system variables; leaving out variables helps declutter the list
Delay	Specifies the delay time before the list of suggestions displays.Prompts you: **Specify the delay time in seconds <0.50>** - **(type the desired delay time, and then press Enter).**
ON	Switches on auto-complete mode.
OFF	Switches off auto-complete mode.

Procedures

- Using autocomplete in the command bar (1.12.2)

Related Commands

Commands - lists the names of commands used by the program Settings - controls the values of variables through a dialog box SetVar - controls and lists variables at the command line

Figure 6.143



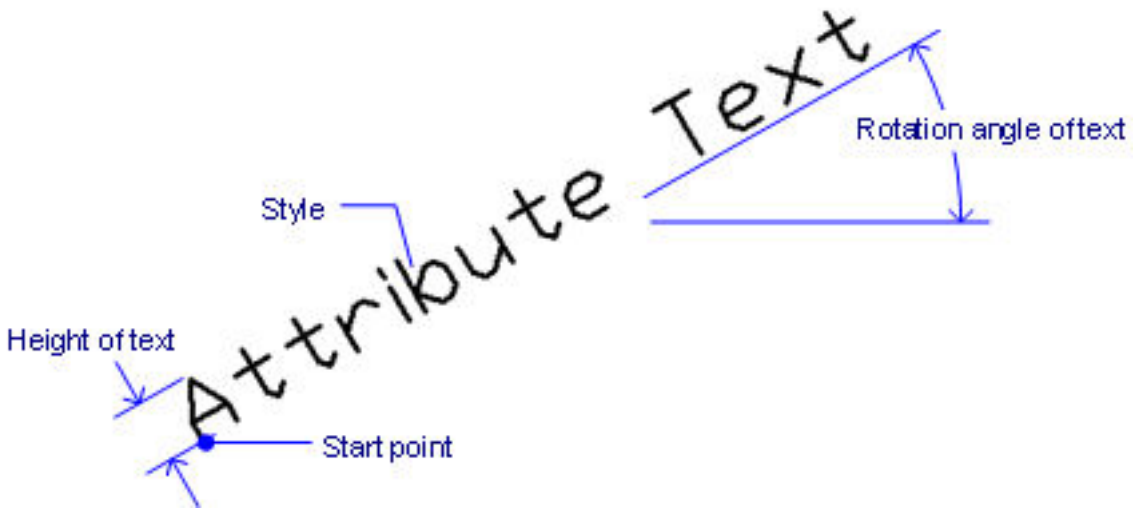
6.4.34 -AttDef

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Defines attribute data (used by blocks) at the command prompt. This command is meant for use by macros, scripts, and LISP routines. To specify attributes with a dialog box, use the **AttDef (6.4.27)**command.

Accessing the Command

command bar: **-attdef** : **-attdef Prompts you in the command bar:** Toggle attribute modes : Constant=Off Invisible=Off Preset=Off Verify=Off Lock position=Off Multiple lines=Off <done>:**(Enter an option, or else press Enter to continue)** Name of attribute: **(Enter a name of up to 255 characters)** Prompt: **(Enter a prompt of up to 255 characters)** Default text: **(Enter a default value of up to 255 characters)** Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: **(Pick a point, or enter an option)** Height of text <0.2000>**: (Enter a value, or else press Enter to accept the default height)** Rotation angle of text <0>: **(Enter a value, or press else Enter to accept the default)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image50.jpg})

Figure 6.144



Command Options

Toggle Attribute Modes Options	Description
Constant	Type **C** , then press Enter to toggle the constant mode: • **Off** - users can change the value of the attribute • **On** - users cannot change the value.
Invisible=Off	Type **I** , then press Enter to toggle the invisibility mode: • **Off** - displays the attribute in the drawing • **On** - hides the attribute Invisible attributes are not displayed nor printed, but can be displayed with the AttDisp (6.4.35) command.
Preset=Off	Type **P**, then press Enter to toggle the prest mode: • **Off** - prompts the user to enter values • **On** - inserts attributes automatically, without prompting the user The value of attributes can be changed later with the AttEdit (6.4.36) command.
Verify=Off	Type **V**, then press Enter to toggle the verify mode: • **Off** - user enters the attribute value once • **On**- forces the user to enter the value a second tim to ensure the value is entered correctly
Lock position=Off	Type **L**, then press Enter to toggle the lock-position mode: • **Off** - users can move attributes using grip editing • **On** - locks the position of the attribute within the block reference
Multiple lines=Off	Type **M**, then press Enter to toggle the multiple-lines mode: • **Off** - forces the use only of single-line text • **On** - allows the use uses multiline text

Additional Options	Description
Name of attribute	Specifies the tag of the attribute; this is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers, and punctuation.
Prompt	Specifies the user prompt; this is displayed at the command prompt when the attribute is later inserted in the drawing. You can leave this field blank; BricsCAD uses the tag as the prompt during attribute insertion.
Default text	Specifies the default value, which is displayed in angle brackets, such as <360>. As with other defaults, you need need only to press Enter to accept this value.If the **Multiple lines** mode is on, you are prompted: **Next line or <done>** - **(type the next line of the Mtext, or else press Enter to stop.)**

Text	Specifies the style and justification of the text: • **Style** - specifies the name of the text style; enter the name of a style present in the drawing • **Align**- aligns the text between two points; the text height is adjusted to maintain its aspect ratio • **Fit**- aligns the text between two points; the text height is maintained • **Center**- centers the text's baseline on the pick point • **Middle**- centers the text on the pick point • **Right**- right justifies the text • **Justify**- accesses additional justification options. • **Start point** - left-justifies the text
Height of text	Specifies the height of the attribute text, unless overridden by the text style or justification. Enter a value, or pick two points to indicate the height.
Rotation angle of text	Specifies the rotation angle of the rotation text. Enter a value, or pick two points to indicate the angle relative to the positive x axis.

Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. AttDisp (6.4.35) - toggles the display of attribute text. AttEdit (6.4.36) - edits attribute values and properties. -AttExt (6.4.29) - extracts attributes from drawings to data files through the command bar. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. DataExtraction (6.10.1) - exports entity properties, block attributes and drawing information to a an external file. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. InsertAligned (6.17.15) - inserts blocks repeatedly, and inserts mirrored blocks. Properties (6.24.49) - edits attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place.

Figure 6.145



6.4.35 AttDisp

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the display of attribute text in the drawing, between turn all off, turn all on, or turn on only non-visible ones.

Accessing the Command

command bar: **attdisp** transparent: **'attdisp** alias: **ad** menu bar: **Tools | Attributes | Attribute Display** toolbar: **Attributes |** ribbon:**Insert | Blocks | Attribute Display** : **attdisp Prompts you in the command bar:** Attribute display mode [ON/OFf/Normal] <Normal>: **(Enter an option)** When attributes are off or invisible, they do not display in the drawing and are not plotted.

Command Options

Option	Description
Attribute display mode	Changes the display of attribute values: • **ON** - displays all attributes, including those set to invisible mode by the AttDef (6.4.27) command • **OFF** - hides all attributes • **Normal** - displays attributes, except those set to invisible mode

Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. DataExtraction (6.10.1) - exports entity properties, block attributes and drawing information to a an external file. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. Properties (6.24.49) - edits attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place.

Figure 6.146



6.4.36 AttEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits the values and properties of attributes.This command is meant for editing attributes globally with macros, scripts, and LISP routines.For editing individual attributes, it is easier to use the dialog box of the EAttEdit (6.13.1) command.

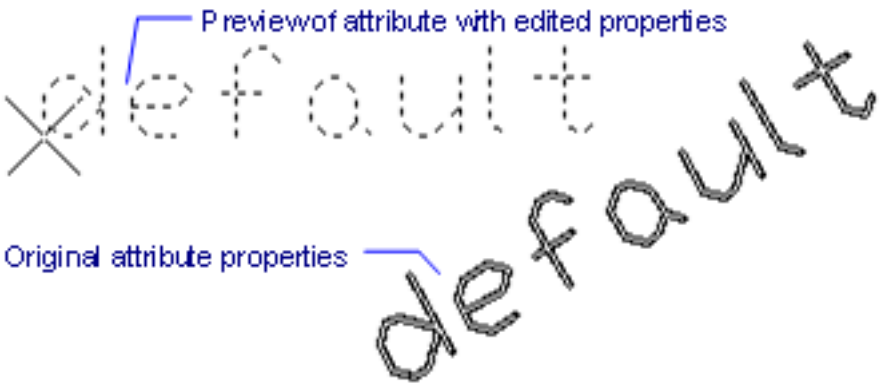
Accessing the Command

 command bar: **attedit** alias: **-ate** menu bar: **Tools | Attributes | Edit Attributes** toolbar: **Attributes** | ribbon:**Insert | Blocks | Edit Attributes** : **attedit Prompts you in the command bar:** Edit attributes one at a time? [Yes/No] <Yes>: **(Enter Y or N)**

- Y (yes) - edits attributes one at a time; allows changes to values and properties
- N (no) - edits attributes globally; allows changes to values only

As you change the attribute text, the program previews them: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU **To edit the values and all properties of attribute text one attribute at a time:** Edit attributes one at a time? <Y> **(Press Enter to edit them one at a time)** Edit attributes of which blocks? <*>: **(Press Enter for all, or else specify the names of specific block)** Edit attributes with these names <*>: **(Press Enter for all, or else specify the individual attribute tag names)** Edit attributes with this text <*>: **(Press Enter for all, or else specify specific attribute values)** Select attributes: **(Select one or more attributes)** Entities in set: 1 Select attributes: **(Press Enter to end entity selection)** 1 attributes were selected. [Position/Angle/Text/Style/Color/Height/Layer/Next/pRev/Quit]<Next>: **(Enter an option, or else press Enter to exit the command) To edit the values of all attributes:** Edit attributes one at a time? <Y>: **(Type N) Global edit of attribute values.** Edit only attributes visible on screen? <Y> **(Type Y or N)** Edit attributes of which blocks? <*>: **(Press Enter for all, or else specify the specific names of blocks)** Edit attributes with these names <*>: **(Press Enter for all, or else specify specific attribute names)** Edit attributes with this text <*>: **(Press Enter for all, or else specify the value)** Select attributes: **(Select one or more attributes)** Entities in set: 1 Select attributes: **(Press Enter to end entity selection)** 1 attributes were selected. Text to change: **(Enter the name of the attribute value you wish to edit)** New text: **(Enter the new value)**

Figure 6.147



Command Options

Option	Description
Edit attributes of which blocks	Specifies the names of blocks; you can use wildcards to specify similarly named blocks: • * refers to all characters in the block's name • ? refers to individual characters
Edit attributes with these names	Specifies the names of attributes. You can use wildcards to specify similarly named tags.
Edit attributes with this text	Specifies the values of attributes. You can use wildcards to specify similarly named values.
Select attributes	Chooses the attributes to edit.
Position	Moves the attribute's insertion point: **Enter text insertion point** - **(pick a new point, or enter coordinates.)**
Angle	Changes the rotation angle of the attribute text. Prompts you: **New rotation angle <0>** - **(enter a new angle, or pick two points to specify the angle.)**
Text	Changes the value of the attribute. Prompts you: **[Change/Replace]<Replace>** - **(type C or R)**: • **Change** - searches and replaces text. • **Replace** - new attribute text replaces the text of the currently selected attribute.
Style	Changes the text style of the attribute; the style name must exist in the drawing. (Use the Style (6.27.53) command to create or edit text styles.) **New text style or ENTER for no change <Standard>** - **(enter a different style name, or else press Enter to keep the style.)**
Color	Changes the color of the attribute text: **New color <BYLAYER>** - **(enter a different color name or number.)**
Height	Change the height of the attribute text: **New text height <0.2>** - **(enter a different text height, pick two points in the drawing, or else press Enter to keep the height.)**
Layer	Changes the layer of the attribute; the layer name must exist in the drawing. (Use the Layer command to create layers.) **New layer or ENTER for no change <0>** - **(enter a different layer name, or else press Enter to keep the layer.)**
Next	Goes to the next attribute; if there are no more attributes to edit, exits the command.
pRev	Goes to the previous attribute, if any.
Edit only attributes visible on screen	Determines which group of attributes to edit: • **Y** (yes) - edits only attributes visible in the current viewport. • **N** (no) - edits all attributes in the drawing.
Text to change	Specifies the attribute value to change.
New text	Specifies the replacement value.
Quit	Exits the command.

Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttDisp (6.4.35) - toggles the display of attribute text. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. DataExtraction (6.10.1) - exports entity properties, block attributes and drawing information to a an external file. DdAtte (6.10.13) - edits the values of attributes through a dialog box. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. Properties (6.24.49) - edits attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place.

6.5 Ai

6.5.1 Ai_CircTan

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws a circle tangent to three entities.

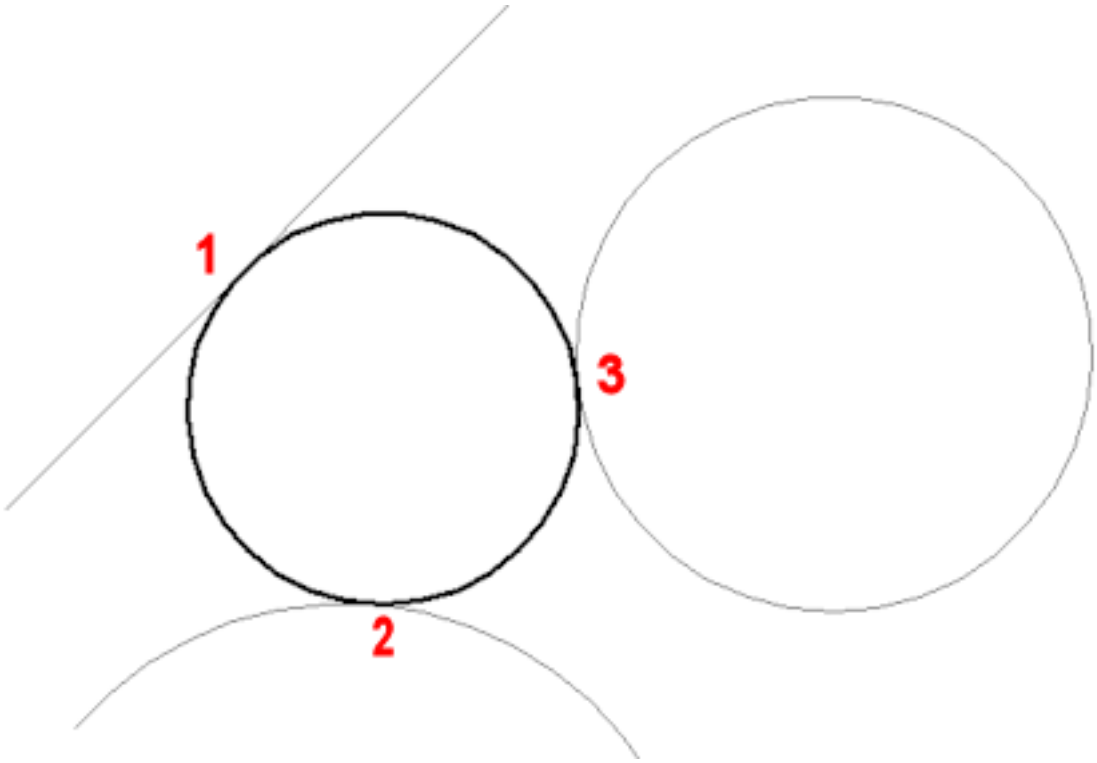
Figure 6.148



Accessing the Command

command bar: **ai_circtan** transparent: **'ai_circtan** : **ai_circtan Prompts you in the command bar:** Enter Tangent spec: **(Select a line, circle, arc, polyline segment, xline or ray)** Enter second Tangent spec: **(Select a line, circle, arc, polyline segment, xline or ray)** Enter third Tangent spec: **(Select a line, circle, arc, polyline segment, xline or ray)** (\vref {fig:C:/help/enUS/V18/CmdRef/source/AiCircTan.png})

Figure 6.149



Tutorial: Click here (1.9.1) to watch

Command Options

Option	Description
Enter Tangent spec	Select one of the following entities: • Line • Polyline segment • Polyarc • Circle • Arc • Xine • Ray When the entity is ineligible, the program responds, ”No tangent found for selected point.”
Enter second Tangent spec	Select a second entity
Enter third Tangent spec	Select a third entity

Related Commands

Circle (6.9.20) - draws circles by several methods.

Figure 6.150



6.5.2 AiDimFlipArrow

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Flips individual arrows of a dimension.

Accessing the Command



command bar: **aidimfliparrow** menu: **Dimension | Flip dimension arrowhead** toolbar: **Dimensions |** : **aidimfliparrow Prompts you in the command bar:** Select entities: **(Click the extension or dimension line nearest to the dimension arrow you want to flip)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source The arrow flips to the other side of the extension line. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/C

Figure 6.151

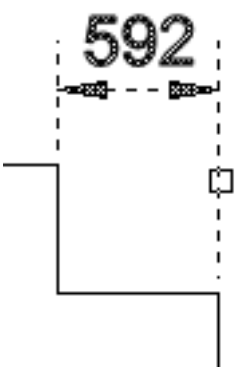
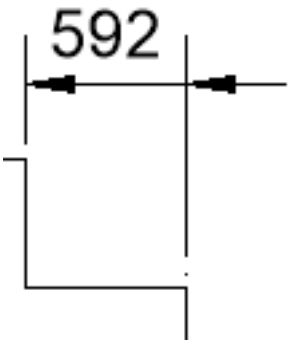


Figure 6.152



Command Options

Option	Description
Select entities:	Click the extension line of the dimension arrow you want to flip. Even though the prompt is plural (entities), this command flips only one arrowhead at a time. It flips the one closest to the pick point along the dimension or extension line.

Related Commands

DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17) - places radius dimensions. DimAngular (6.11.4) - places angular dimensions. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines.

Figure 6.153



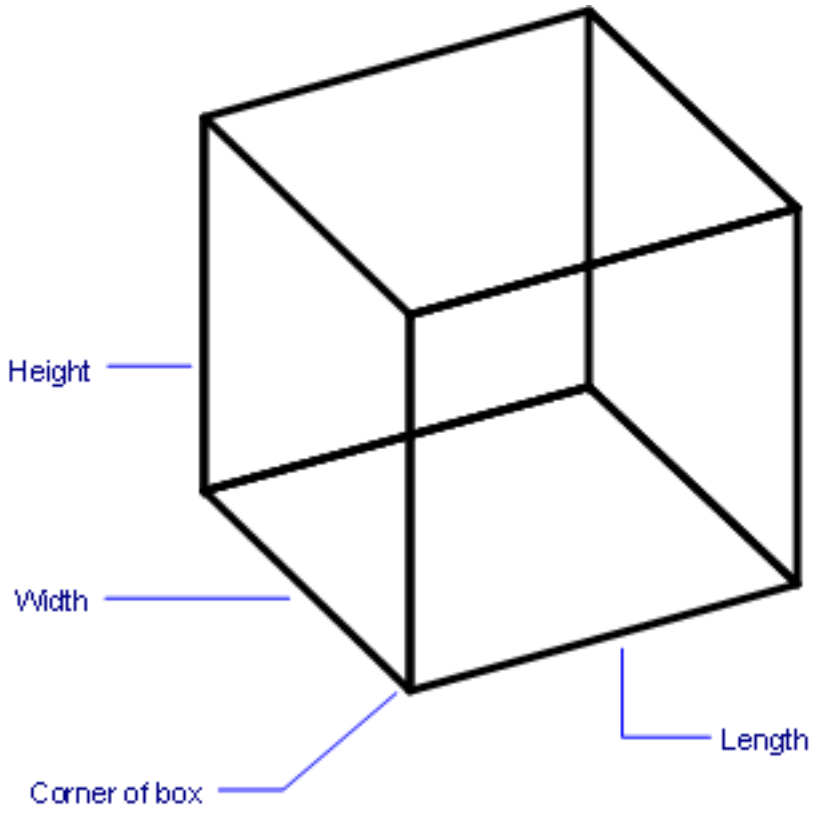
6.5.3 Ai_Box

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws 3D boxes from polyface meshes.

Accessing the Command

 command bar: **ai_box** menu bar: **Model | Meshes | Box** toolbar: **Meshes |** ribbon:**Model | Meshes | Box** : **ai_box Prompts you in the command bar:** Corner of box <0,0,0>: **(Pick a point, or enter coordinates)** Length of side of box: **(Pick a point, or enter a length)** Set width of box or [Cube]: **(Pick a point, enter a width, or type C)** Height of box: **(Enter a height)** Rotation angle for box <0>: **(Enter a rotation angle, or press Enter for none)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image8.png})

Figure 6.154



Command Options

Option	Description
Corner of box	Specifies the lower-left corner of the box.
Length of side of box	Specifies the length of one side, along the positive x-direction.
Width of box	Specifies the width of one side, along the positive y-direction.
Height of box	Specifies the height of one side, along the positive z-direction.
Rotation angle for box	Specifies the counterclockwise rotation angle about the corner (initial pick point).
Cube	Draws a box with all sides of equal length.

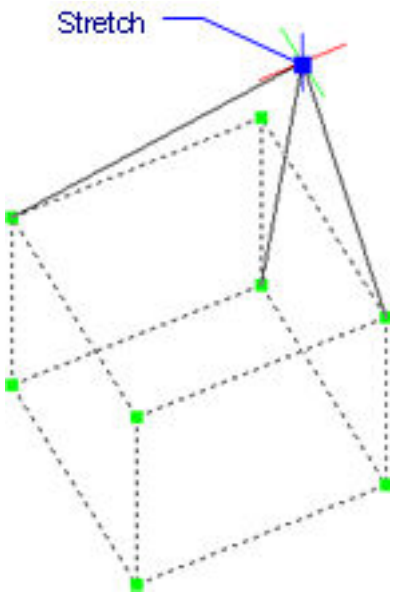
Grips Editing

Mesh-formed boxes can be edited directly through grips:

1. Select the box. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image483.jpg})

Figure 6.155



Related Commands

Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.156



6.5.4 Ai_Cone

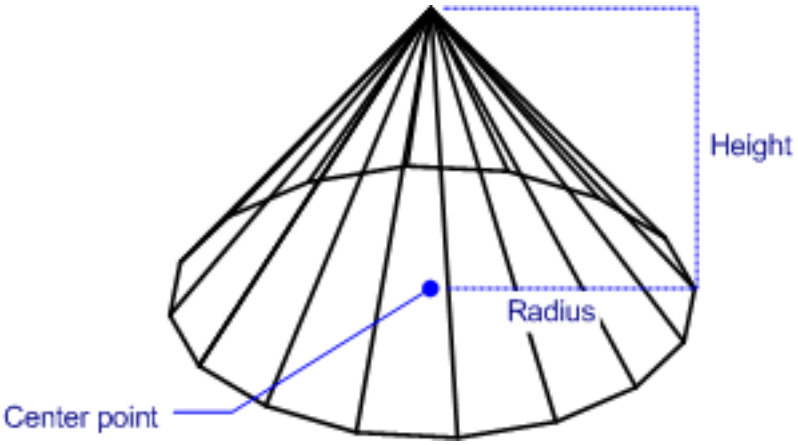
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws 3D cones as polyface meshes, with round or elliptical bases

Accessing the Command



command bar: **ai_cone** menu bar: **Model | Meshes | Cone** toolbar: **Meshes |** ribbon:**Model | Meshes | Cylinder | Cone** : **ai_cone Prompts you in the command bar:** Elliptical/Divisions/<Center of base of cone> <0,0,0>: **(Pick a point, enter x,y,z coordinates, or else enter an option.)** Diameter/<Radius of base>: **(Pick a point, enter coordinates, or type D)** Apex/<Height of cone>: **(Pick a point, enter coordinates, or type A)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image12.png})

Figure 6.157



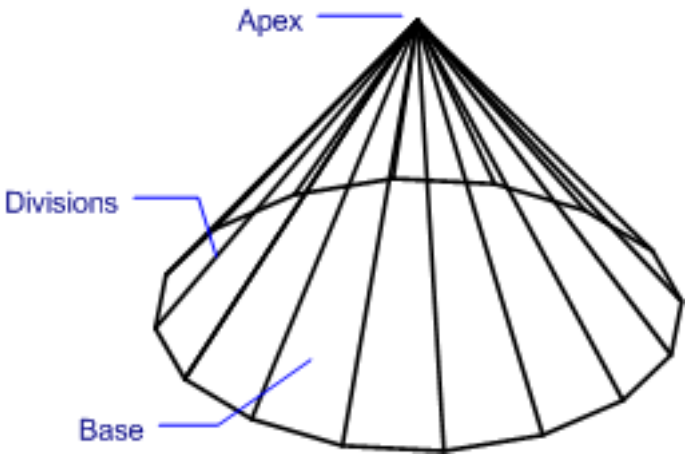
Command Options

Option	Description
Center of base of cone	Specifies the center point of the cone's base: • Pick a point • Enter x,y,z coordinates
Radius of base	Specifies the radius of the cone's base.
Height of cone	Specifies the height of the cone (in the z direction); enter a negative number to draw the cone upside down.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image11.png})

Option	Description
Apex	Specifies x,y,z coordinates for the top of the cone as an alternative to specifying the height. Prompts you: **Apex of cone** - **(specify the top of the cone by pointing or entering a height)**
Diameter	Specifies the diameter of the cone's base; prompts you: **Diameter of base** - **(specify the diameter by picking a point or entering a value)**
Divisions	Changes the number of isolines used to simulate the cone's curved surface. Prompts you: **Number of divisions** - **(change the number of isolines from the default of 1)**
Elliptical (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Prompts4.jpg})	Specifies that the cone should have an elliptical base. Prompts you: 1. **Center specifies the center of the ellipse. First end of ellipse axis** - **(specify the endpoint of one ellipse axis.)** 2. **Second end of ellipse axis** - **(specify the other endpoint of the first axis)** 3. **Other axis** - **(specify the length of the other (second) axis)**

Figure 6.158



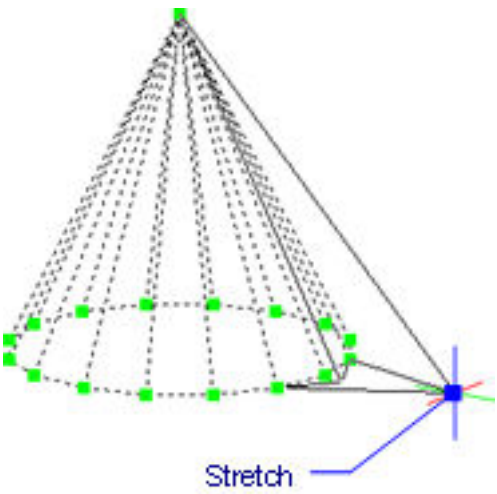
Grips Editing

Mesh-formed cones can be edited directly through grips:

1. Select the cone. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image484.jpg})

Figure 6.159



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.160

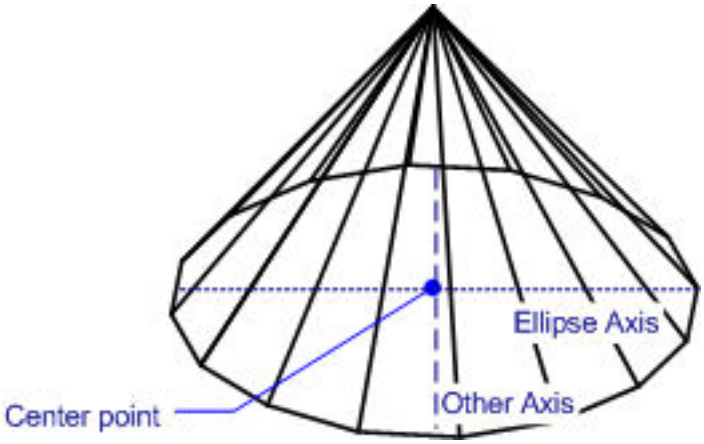


Figure 6.161



6.5.5 Ai_Cylinder

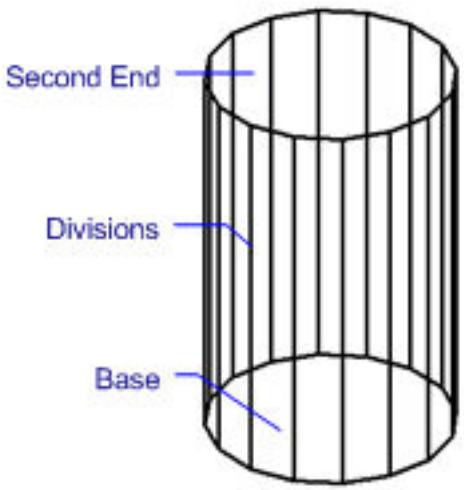
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws 3D cylinders as 3D polyface meshes, with round or elliptical bases.

Accessing the Command



command bar: **ai_cylinder** menu bar: **Model | Meshes | Cylinder** toolbar: **Meshes |** ribbon:**Model | Meshes | Cylinder** : **ai_cylinder Prompts you in the command bar:** Elliptical/Divisions/<Center of base of cylinder> <0,0,0>: **(Pick a point, enter x,y,z coordinates, or enter an option.)** Diameter/<Radius of cylinder>: **(Specify radius, or enter D.)** Center of second end/<Height of cylinder>: **(Specify height, or enter C.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image5.jpg})

Figure 6.162



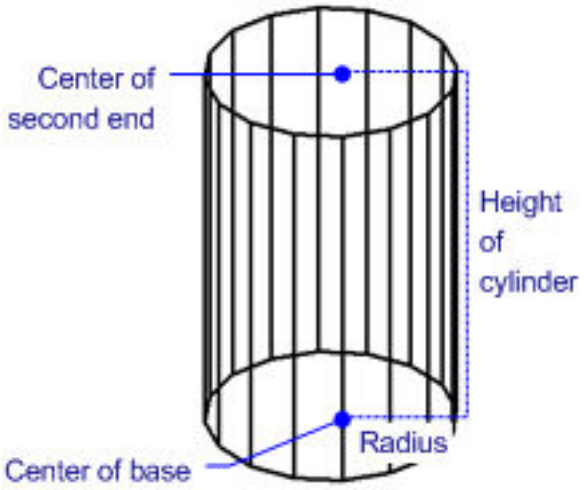
Command Options

Option	Description
Center of base of cylinder	Specifies the center point of the cylinder's base; the base is drawn on the current x,y-plane: • Pick a point • Enter x,y,z coordinates
Radius of cylinder	Specifies the radius of the cylinder.
Height of cylinder	Specifies the height of the cylinder in the z direction; enter a negative number to draw the cylinder downwards.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image6.jpg})

Option	Description
Center of second end	Specifies coordinates for the other of the cylinder, an alternative to specifying the height. Prompts you: **Center of second end** - **(specify the other end of the cylinder by pointing or entering a height.)**
Diameter	Specifies the diameter of the cylinder's base. Prompts you: **Diameter of cylinder** - **(specify the diameter by pointing or entering a value.)**
Divisions	Changes the number of isolines used to simulate the cylinder's curved surface. Prompts you: **Number of divisions** - **(change the number of isolines from the default of 16)**
Elliptical (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image7.jpg})	Specifies that the cylinder should have an elliptical cross section. Prompts you: 1. **Center** - **(specifies the center of the ellipse)** 2. **First end of ellipse axis** - **(specify the endpoint of one ellipse axis)** 3. **Second end of ellipse axis** - **(specify the other end point of the first axis)** 4. **Other axis** - **(specify the length of the second axis)**

Figure 6.163



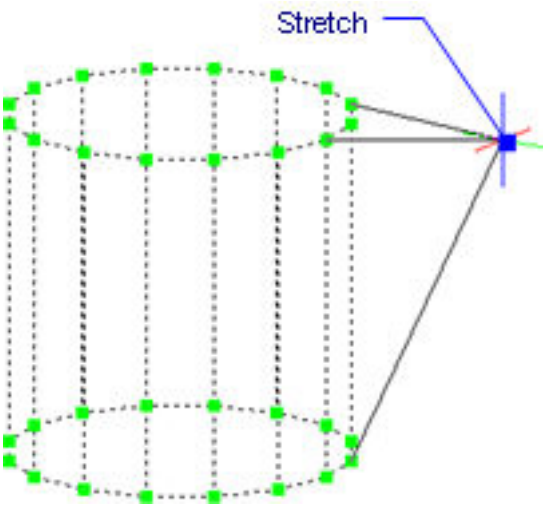
Grips Editing

Mesh-formed cylinders can be edited directly through grips:

1. Select the cylinder. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image485.jpg})

Figure 6.164



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.165

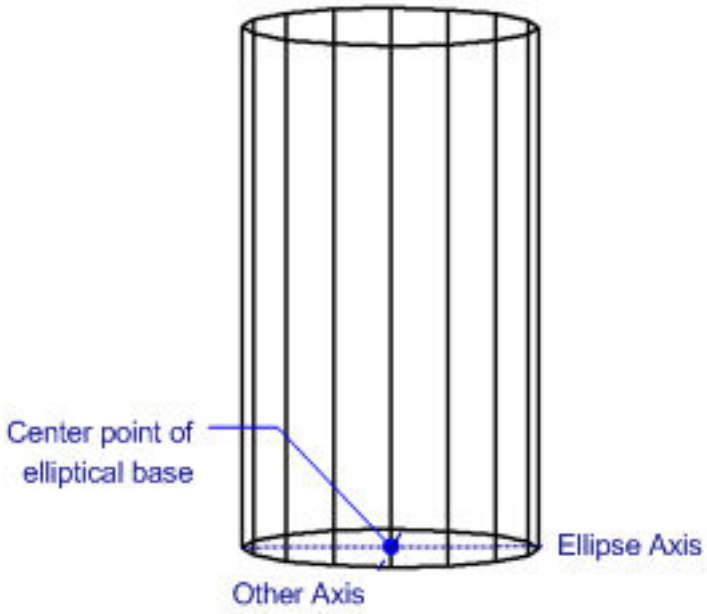
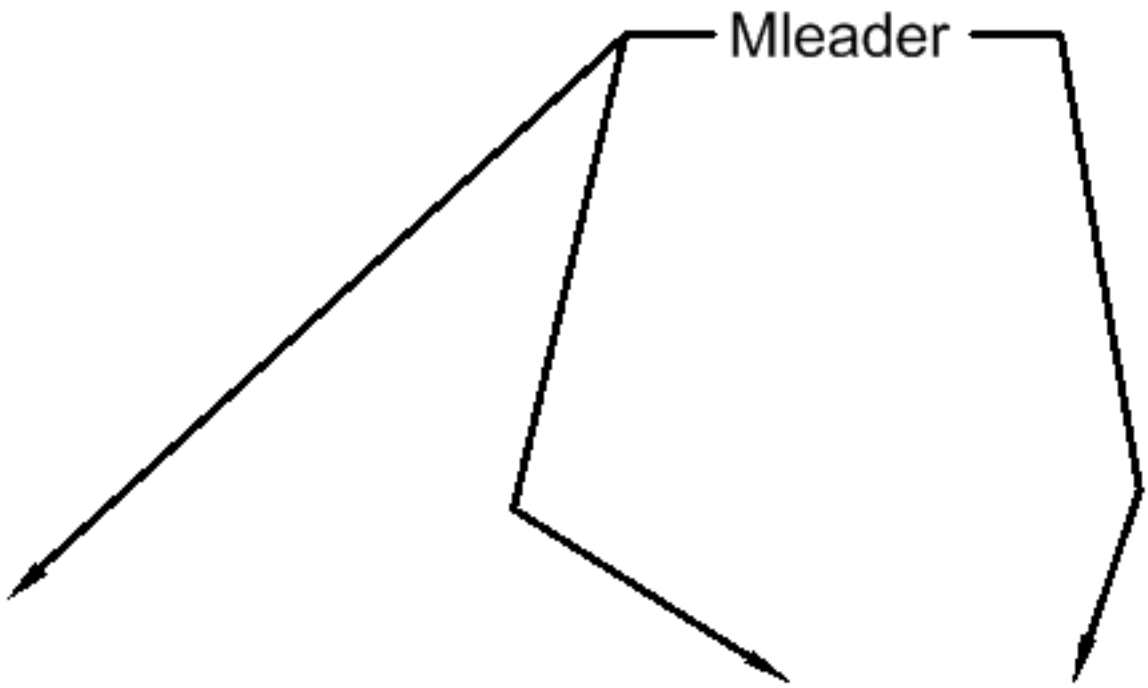


Figure 6.166



Figure 6.167



6.5.6 AiMLeaderEditAdd

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adds and removes leader lines from a multileader entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/A/aimleadereditadd-eg.png})

Accessing the Command

command bar: **aimleadereditadd** : **aimleadereditadd Prompts you in the command bar:** Select a multileader:**(Choose a multi-line leader)** Specify leader arrowhead location or [Remove leaders]:**(Pick a point to locate the vertex the new leader)** Specify next point:**(Pick a point to locate the arrowhead)** Specify leader arrowhead location or [Remove leaders]:**(Enter an option, or else press Enter to end the command)**

Command Options

(Pick a point to locate the vertex the new leader)

Option	Description
Select a multileader	Selects the multiline to which you wish to add leaders, or remove them

Specify leader arrowhead location	Pick a point to locate the vertex the new leader; prompts you: 1. **Specify next point** - **(Pick a point to locate the arrowhead)** 2. **Specify leader arrowhead location or [Remove leaders]**- **(enter an option)** • Pick a another point to locate another new leader • Press **R** to remove leaders • Press **Enter** to end the command
Remove leaders	Selects the leaders to remove; prompts you: 1. **Select leaders to remove or [Add leaders]** - **(select one or more leaders to remove)** 2. **Select leaders to remove or [Add leaders]** - **(enter an option)** • Choose one or more leaders to remove • Press**A**to add leaders • Press **Enter** to end the command The leaders are not removed until you end the command.

Related Commands

MLeader (6.21.20) - creates multileader entities using the current multileader style. MLeaderEditExt (6.21.24) - adds and removes leader lines, adds and removes vertices from a multileader entity. MLeaderStyle (6.21.25) - Creates and manages multileader styles in the Drawing Explorer dialog box.

Figure 6.168



6.5.7 Ai_Dish

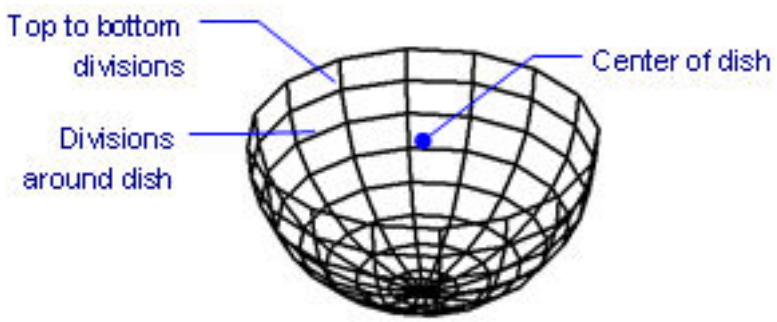
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws the bottom half of spheres as 3D polyface meshes.

Accessing the Command



command bar: **ai_dish** menu bar: **Model| Meshes | Dish** toolbar: **Meshes |** ribbon:**Model | Meshes | Cylinder | Dish** : **ai_dish Prompts you in the command bar:** Center of dish: **(Pick a point or enter coordinates)** Set radius of dish or [Diameter]: **(Pick a point, enter x,y,z coordinates, or enter a value)** Number of divisions around dish <16>: **(Enter a number, or press Enter to keep the default value)** Number of top-to-bottom divisions <8>: **(Enter a number, or press Enter to keep the default value)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cmd

Figure 6.169



Command Options

Option	Description
Center of dish	Specifies the center point of the dish's open end; the end is drawn on the current x,y-plane.
Radius of dish	Specifies the radius of the dish.
Number of divisions around dish	Changes the number of horizontal isolines used to simulate the dish's curved surface.
Number of top-to-bottom divisions	Changes the number of vertical isolines from the default of 8.
Diameter (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/Diameter_of_dish.jpg})	Specifies the diameter of the dish. Prompts you: **Diameter of dish: (specify the diameter by pointing or entering a value.)**

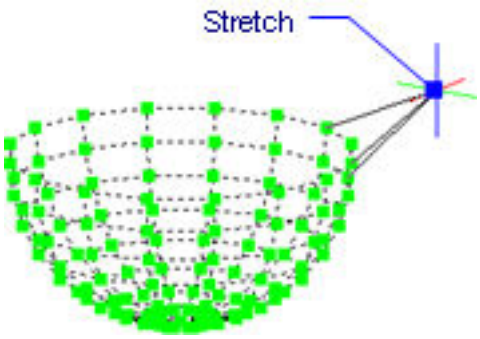
Grips Editing

Mesh-formed dishes can be edited directly through grips:

1. Select the dish. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image486.jpg})

Figure 6.170



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.171

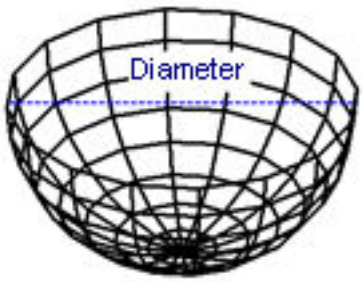


Figure 6.172



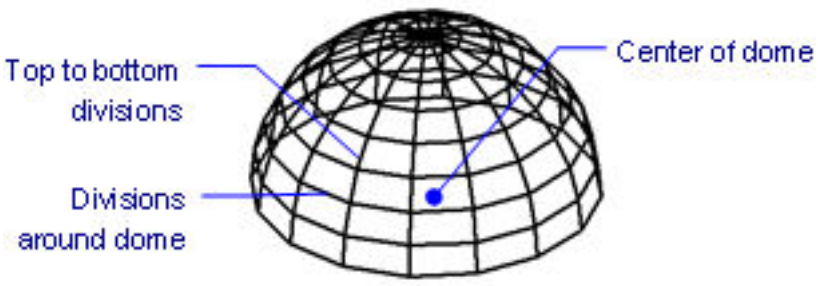
6.5.8 Ai_Dome

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws the upper half of a sphere as a 3D mesh surface.

Accessing the Command

 command bar: **ai_dome** menu bar: **Model | Meshes | Dome** toolbar: **Meshes |** ribbon:**Model | Meshes | Cylinder | Dome** : **ai_dome Prompts you in the command bar:** Center of dome: **(Pick a point or enter coordinates)** Set radius of dome or [Diameter]: **(Pick a point, enter coordinates, or type a value)** Number of divisions around dome <16>: **(Enter a number, or press Enter to keep the default value)** Number of top-to-bottom divisions <8>: **(Enter a number, or press Enter to keep the default value)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18

Figure 6.173



Command Options

Option	Description
Center of dome	Specifies the center point of the dome's open end; the end is drawn on the current x,y-plane.
Radius of dome	Specifies the radius of the dome.
Number of divisions around dome	Changes the number of horizontal isolines used to simulate the dome's curved surface.
Number of top-to-bottom divisions	Changes the number of vertical isolines from the default of 8.
Diameter (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/Diameter of dome.jpg})	Specifies the diameter of the dome. Prompts you: **Diameter of dome: (specify the diameter by pointing or entering a value.)**

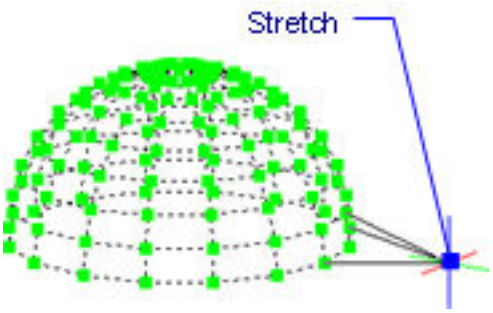
Grips Editing

Mesh-formed domes can be edited directly through grips:

1. Select the dome. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image487.jpg})

Figure 6.174



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.175

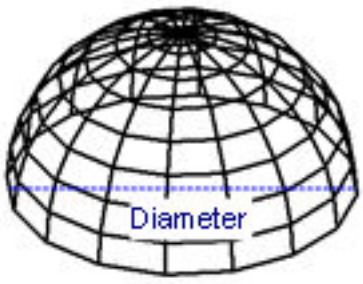


Figure 6.176



6.5.9 Ai_EdgeSurf

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) This command is obsolete. It still exists for backward compatibility only. Please use Edgesurf (6.13.2) instead.

Figure 6.177



6.5.10 Ai_Pyramid

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws a variety of pyramids and tetrahedrons as 3D polyface meshes. **Note:** To draw a pyramid from a 3D solid, use the Pyramid (6.24.58) command.

Accessing the Command



command bar: **ai_pyramid** menu bar: **Model | Meshes | Pyramid** ribbon:**Model | Meshes | Box | Pyramid** **toolbar: Meshes |** : **ai_pyramid Prompts you in the command bar:** First point for base of pyramid: **(Pick a point, or enter coordinates)** Second point: **(Pick another point, or enter coordinates)** Third point: **(Pick a third point, or enter coordinates)** Set last point for base or [Tetrahedron]: **(Pick the last point, enter coordinates, or type T)** Set apex of pyramid or [Ridge/Top]: **(Pick a point, or enter coordinates, or enter an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image16.jpg}) This command draws:

- Pyramids with four-sided bases and four sides; the base does not need to be an equilateral.
- Tetrahedrons with triangular base and three sides; the base can be scalene.
- Pyramids with pointed, ridged, or flat tops.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image15.jpg}) To draw a rotated pyramid, specify the angle during the Apex option.

Figure 6.178

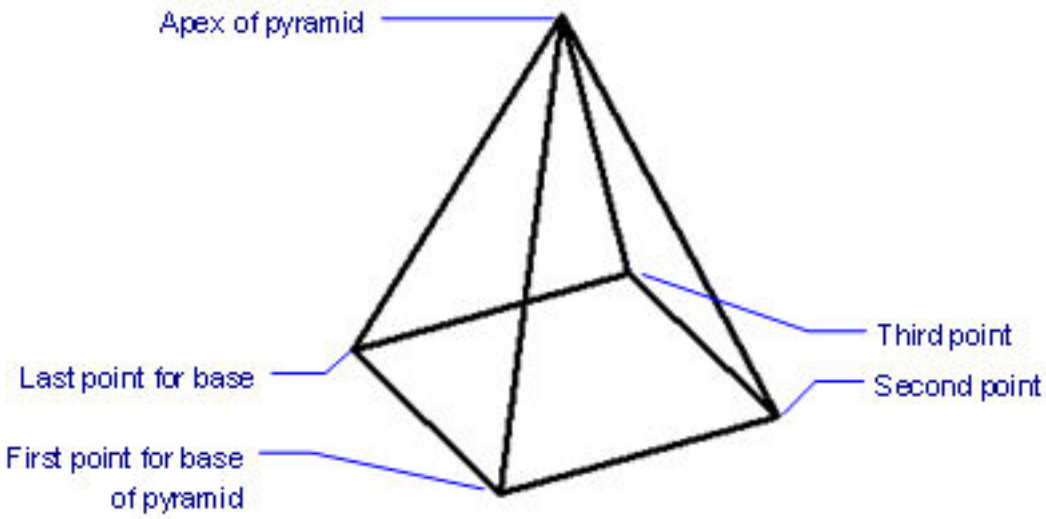
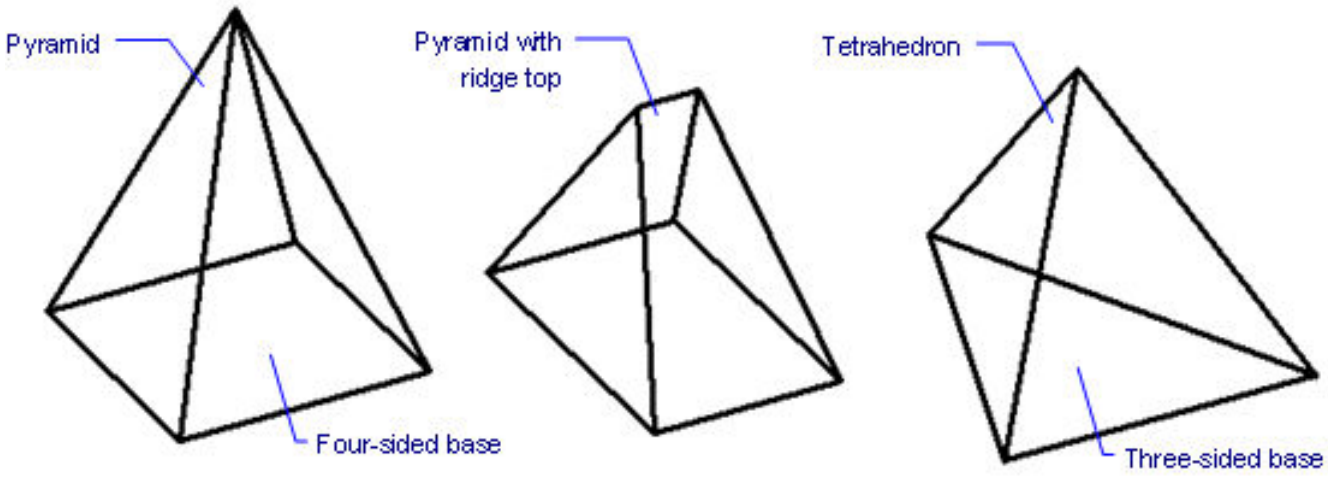


Figure 6.179



Command Options

Option	Description
First point for base of pyramid	Specifies the first corner of the pyramid's base.
Second point	Specifies the second corner of the base.
Third point	Specifies the third point of the pyramid's base, or the last point for a tetrahedron's base.
Last point for base	Specifies the fourth point for the pyramid's base.
Apex of pyramid	Specifies the height and angle of the pyramid; draws a pointy top.
Tetrahedron (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image17.jpg})	Draws tetrahedrons; prompts you: **Apex of tetrahedron - (specify the top of the tetrahedron.)**
Ridge (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image18.jpg})	Draws a ridge at the top of the pyramid. Prompts you: 1. **Start of ridge - (specifies the height and location of one end of the ridge.)** 2. **End of ridge - (specifies the height and location of the other end of the ridge.)**
Top surface (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image19.jpg})	Draws a flat surface at the top of the pyramid; prompts you: 1. **First point on top surface of pyramid - (specifies the starting corner of the pyramid's flat top.)** 2. **Second point - (specifies the next corner.)** 3. **Third point - (specifies the third corner for pyramid, and the last corner for a tetrahedron.)** 4. **Last point of top surface - (specifies the fourth corner for the pyramid.)**

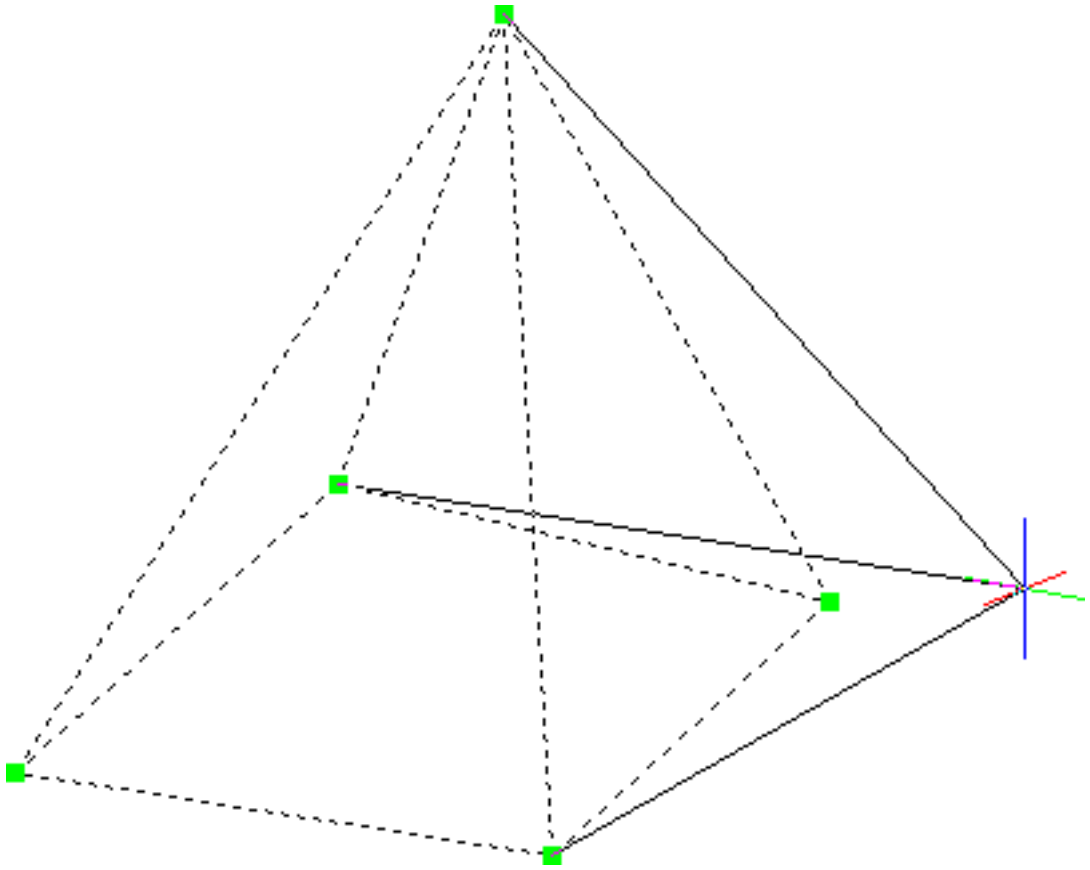
Grips Editing

Mesh-formed pyramids can be edited directly through grips:

1. Select the pyramid. Notice that it has one grip at each vertex.
2. Drag the grip to move the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image767.png})

Figure 6.180



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.181

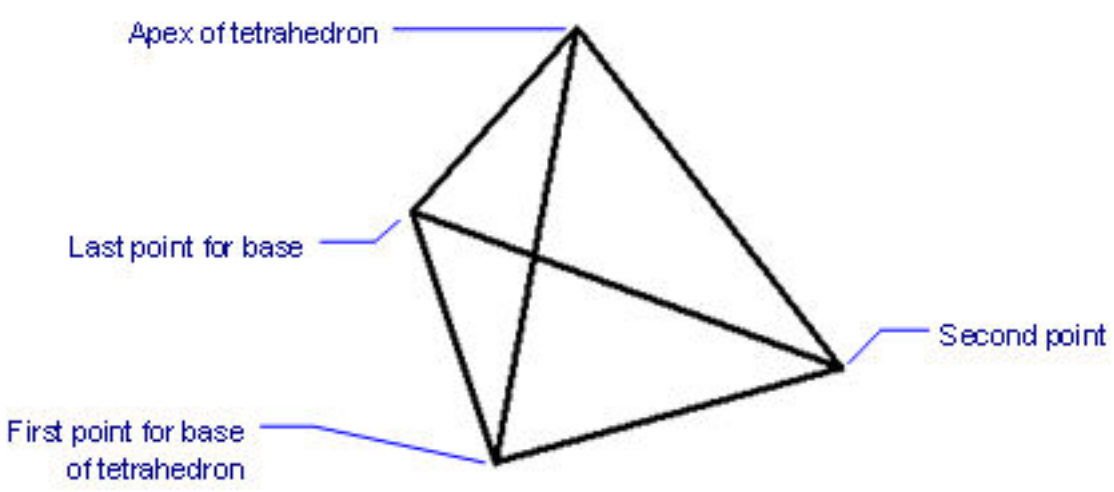


Figure 6.182

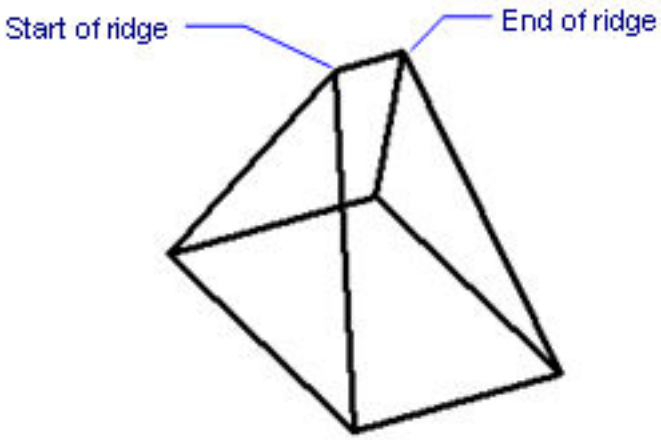


Figure 6.183

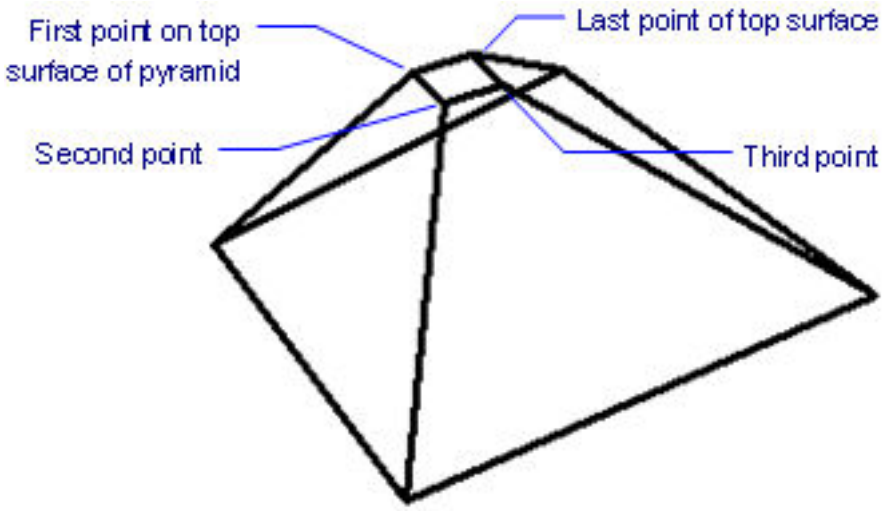


Figure 6.184



6.5.11 Ai_RevSurf

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) This command is obsolete. It still exists for backward compatibility only. Please use RevSurf (6.26.32) instead.

Figure 6.185



6.5.12 Ai_RuleSurf

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) This command is obsolete. It still exists for backward compatibility only. Please use RuleSurf (6.26.45) instead.

Figure 6.186



6.5.13 Ai_Sphere

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws 3D spheres as 3D polyface meshes.

Accessing the Command

 command bar: **ai_sphere** menu bar: **Model | Meshes | Sphere** toolbar: **Meshes |**  ribbon:**Model | Meshes | Cylinder | Sphere** : **ai_sphere Prompts you in the command bar:** Center of sphere: **(Pick a point, or enter coordinates)** Set radius of sphere or [Diameter]: **(Specify a distance, or type D)** Number of longitudinal segments <16>: **(Type a value or press Enter to accept the default number)** Number of latitudinal segments <16>: **(Type a value or press Enter to accept the default number)** (\vref {fig:C:/help/enUS/V18/CmdRef/source/image25.jpg}) If Dynamic Dimensions (1.15.9) are active, you can enter the radius in the dynamic dimension entry field. (\vref {fig:C:/help/enUS/V18/CmdRef/source/AiSphereDynDim.png})

Figure 6.187

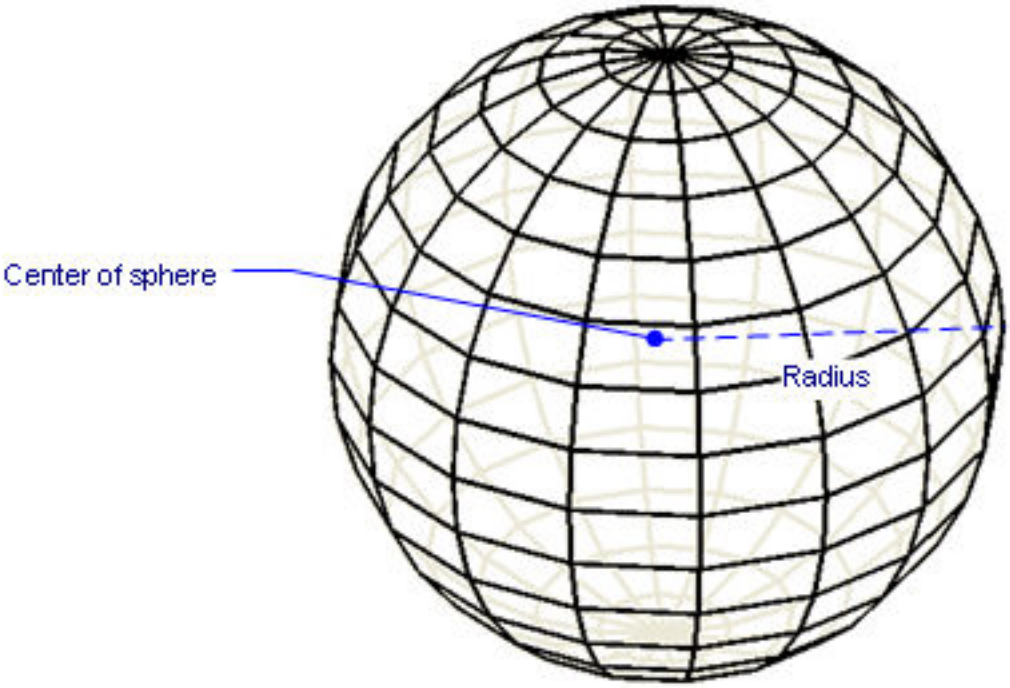
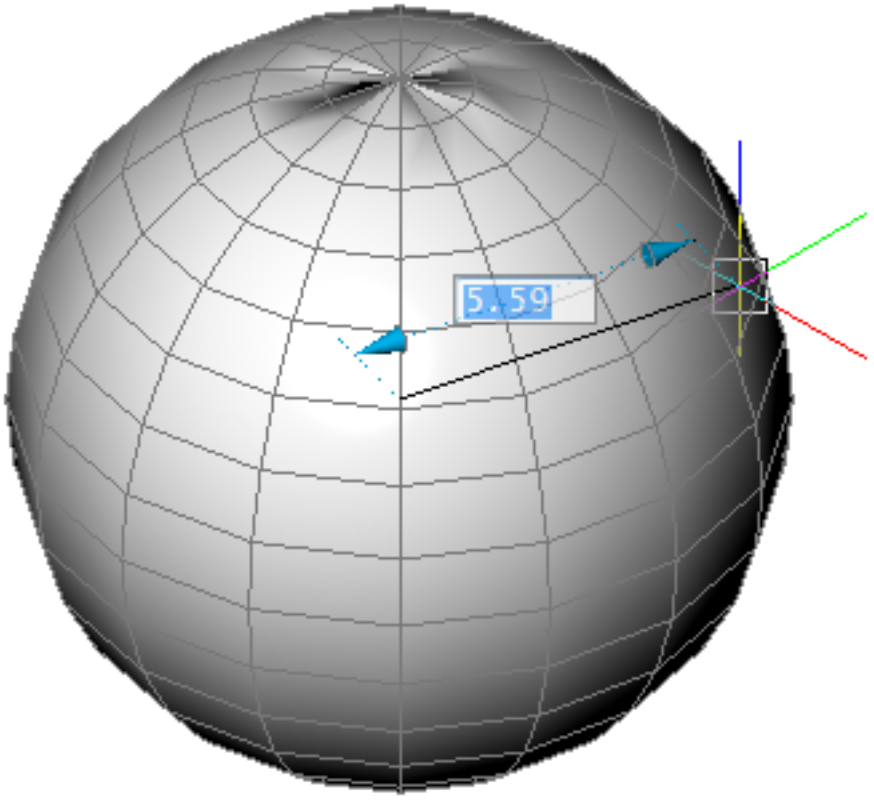


Figure 6.188



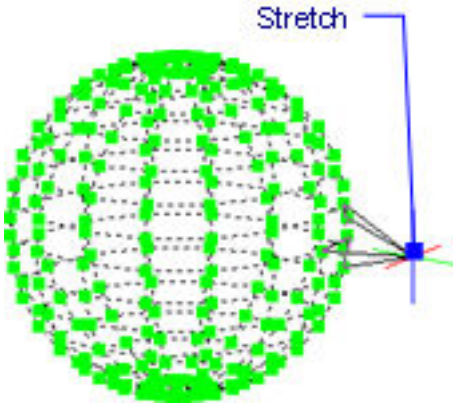
Command Options

Option	Description
Center of sphere	Specifies the center point of the sphere; enter x,y- or x,y,z-coordinates, or pick a point in the drawing.
Radius of sphere	Specifies the radius.
Diameter	Specifies the diameter.
N umber of longitudinal segments	Specifies the number of segments of the circles perpendicular to the vertical axis of the sphere.
Number of latitudinal segments	Specifies the number of segments of the circles through to the vertical axis of the sphere.

Grips Editing

(\vref {fig:C:/help/enUS/V18/CmdRef/source/image491.jpg})

Figure 6.189



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.190



6.5.14 Ai_TabSurf

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) This command is obsolete. It still exists for backward compatibility only. Please use TabSurf (6.29.8) instead.

Figure 6.191



6.5.15 Ai_Torus

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws tori (donut shapes) as 3D polyface meshes.

Accessing the Command



command bar: **ai_torus** menu bar: **Model | Meshes | Torus** toolbar: **Meshes |** ribbon: **Model | Meshes | Cylinder | Torus** : **ai_torus Prompts you in the command bar:** Center of whole torus: <0,0,0> **(Pick a point, or enter coordinates)** Set Radius of whole torus or [Diameter]: **(Specify the radius, or type D)** Set radius of body of torus [Diameter]: **(Specify the radius, or type D)** Segments around body of torus <16>: **(Type a value or press Enter to accept the default number)** Segments around whole torus <16>: **(Type a value or press Enter to accept the default number)**

Command Options

Option	Description
Center of whole torus	Specifies the center of the torus; you can press Enter to accept the default coordinates.
Radius of whole torus	Specifies the overall radius of the torus; note that the distance is measured to the center of the tube (body).
Radius of body of torus	Specifies the radius of the torus body, or tube. It is not necessary for the body radius to be smaller than the whole radius.
Diameter	Specifies the diameters of the whole torus and/or body.
Segments around body of torus	Specifies the number of segments of the circles around the body of torus.
Segments around body whole torus	Specifies the number of segments of the circles around the whole torus.

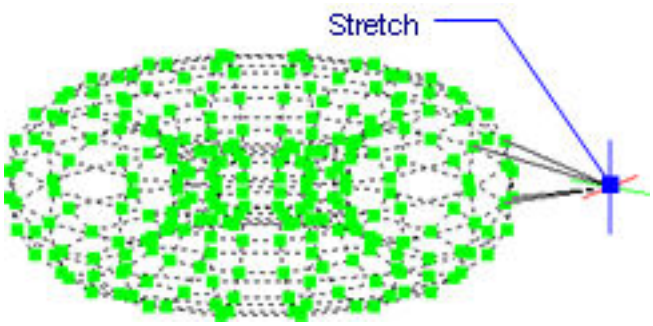
Grips Editing

Mesh-formed torus surfaces can be edited directly through grips:

1. Select the torus. Notice that it has one grip at each vertex.
2. Drag the grip to move the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image493.jpg})

Figure 6.192



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.193



6.5.16 Ai_Wedge

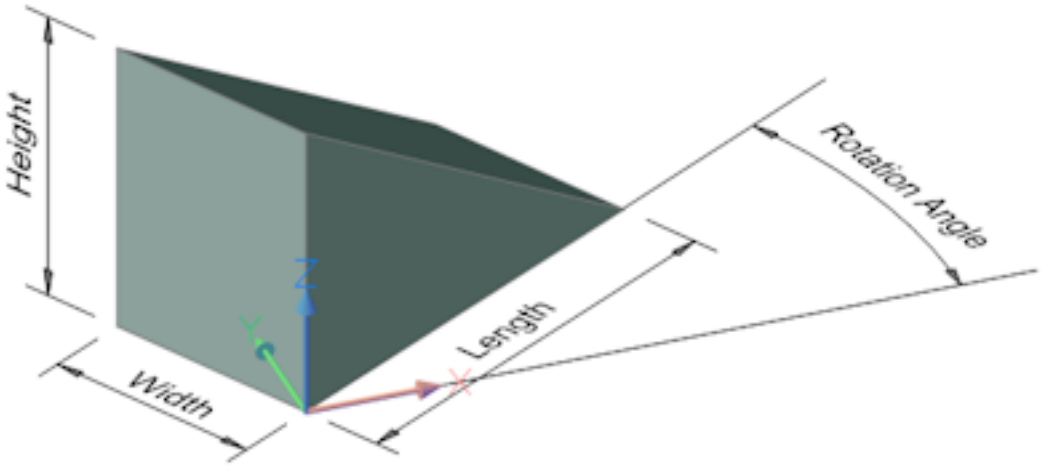
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws a wedge as a 3D mesh surface.

Accessing the Command



command bar: **ai_wedge** menu bar: **Model | Meshes | Wedge** toolbar: **Meshes |** ribbon: **Model | Meshes | Box | Wedge** : **ai_wedge** Prompts you in the command bar: **Corner of wedge: (Pick a point, or enter coordinates) Length of wedge: (Pick a point, or enter a value) Width of wedge: (Pick a point, or enter a value) Height of wedge: (Pick a point, or enter a value) Rotation angle for wedge <0>: (Pick a point, or enter a value)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/AiWedge.png})

Figure 6.194



Command Options

Option	Description
Corner of wedge	Pick a point, or x,y,z enter coordinates
Length of wedge	Pick a point, or enter a value
Width of wedge	Pick a point, or enter a value
Height of wedge	Pick a point, or enter a value
Rotation angle for wedge	Pick a point, or enter a value

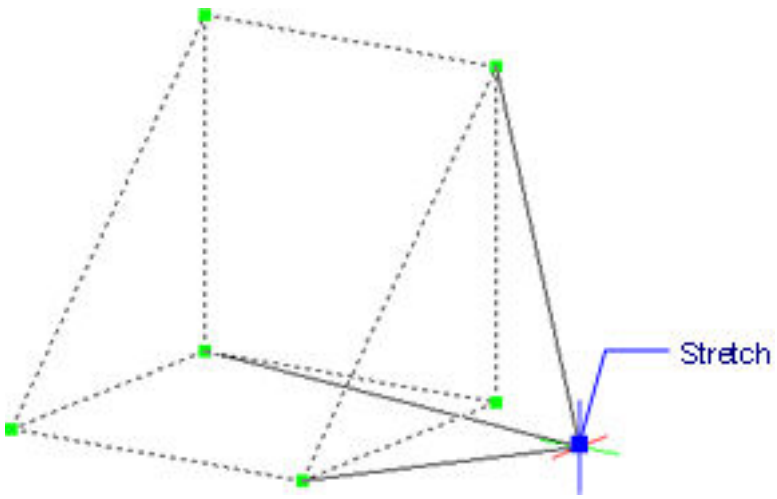
Grips Editing

Mesh-formed wedges can be edited directly through grips:

1. Select the wedge. Notice that it has one grip at each vertex.
2. Drag the grip to move the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image494.jpg})

Figure 6.195



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces.

6.6 B

6.6.1 Background

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Sets a background for the current viewport, such as gradient colors and images.

Figure 6.196



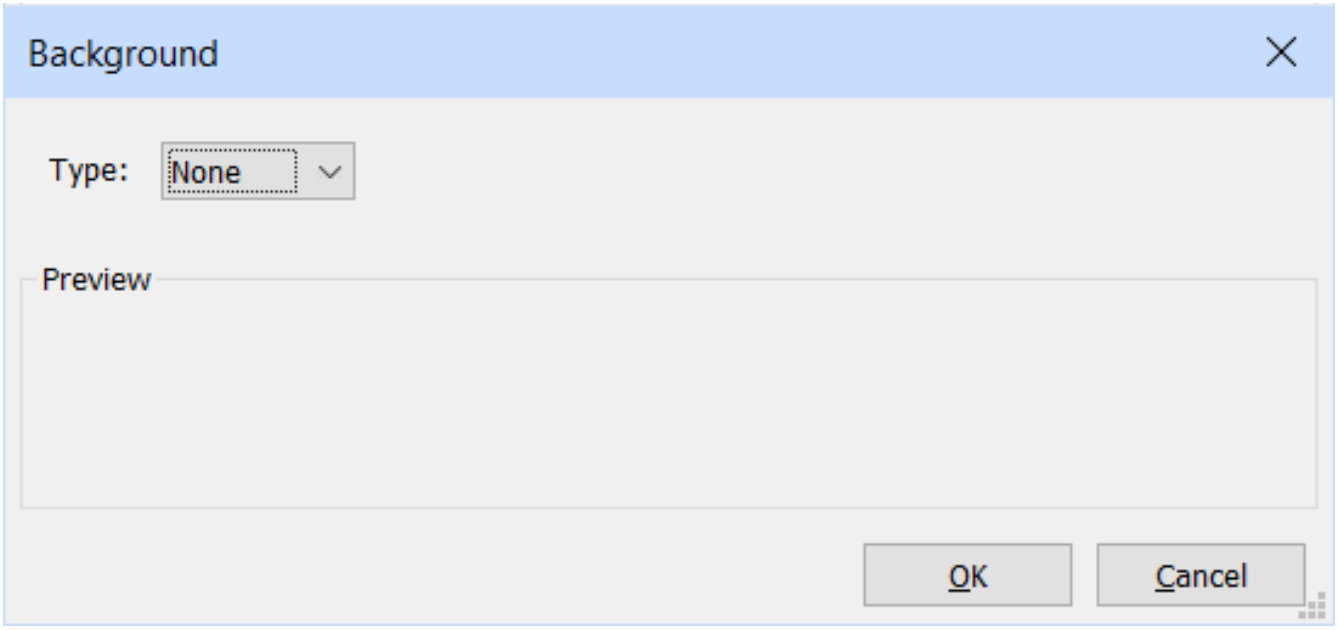
Accessing the Command

 command bar: **background** transparent: **'background** alias: **backgrounds** This command has an effect only

- When the**Perspective**variable is on
- In model space.

: **background** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/background-dlg.png}) Choose a **Type**from the droplist, and then select options. Click **OK** when done. The background appears in the viewport. Backgrounds are viewport-specific, so you can have a different type of background in every viewport.

Figure 6.197



Command Options

None Options	**Description**
Type = None	Displays the default background in model space, which depends on the **Perspective** variable: **Off** - background is the color specified by the **BkgColor** variable, most often black or white; the Background command has no effect **On** - two-color gradient that simulates the ground with sky; the colors are specified by variables that begin with Gradient-, such as GradientColorTop. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/E
Solid Options	**Description**
Color	Type = **Solid** sets a single color for the background of the current viewport. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/ba solid-dlg.png}) **Color**button opens the Color (6.9.33) dialog box: 1. Select a color from the dialog box, either an index color or a true color. 2. Click **OK** to close the Color dialog box. The Preview area shows the color. 3. Click **OK** to close th Background dialog box. The background color changes to the new color.
Gradient Options	**Description**
	Type = **Gradient** sets a two- or three-color for the background of the current viewport.
Rotate	Specifies the angle at which to rotate the gradient; this option, however, does not work.

Colors	Choose the number of colors for the gradient: **Two** - displays a two-color gradient; click the **Top** and **Bottom** buttons to choose the colors from the Color (6.9.33) command's dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/ba gradient-2-dlg.png}) **Three** -displays a three-color gradient; click the **Top**, **Middle**, and **Bottom** buttons to choose the colors from the Color (6.9.33) command's dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/ba gradient-3-dlg.png})
Image Options	**Description**
Type = **Image**sets an image, such as a photograph, for the background of the current viewport. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/ba image-dlg.png})	
Image Settings	Specifies the path to the raster (image) file used for the background;click the **Browse** button ( ) to select a BMP, GIF, JPDG/JPEG, or PNG file. Until you select a file, none of the other options in the dialog box are available to you.
Position	Positions the image in the viewport; chose from: • **Center** - centers the image in the viewport (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/ba image-center.png}) • **Stretch** - stretches small images to fit the size of the viewport (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/ba image-stretch.png}) • **Tile** - repeats small images a sufficient number of times to fill the viewport (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/ba image-tile.png}) As you choose an option, the Preview window shows the result.
Lock aspect ratio	This option is available only when the **Sliders Adjust** option is set to "Scale." • **On** -maintains the image's aspect ratio; the x and y sliders move together • **Off** - allows the image to be distored; the x and y sliders move independently of each other
Sliders adjust	Determines the function of the two slides: • **Offset** - move the image in the viewport • **Scale** - resizes the image; allows you to make the image the same size as the viewport

X and Y sliders	Adjust the offset and/or the scale of the image in the viewport: • **X** - move the slider left and right, or else enter a value • **Y** - move the slider up and down, or else enter a value You can use both the Offset and the Scale sliders to adjust the image's position and size in the viewport.
Reset	Resets the offset and scale to the default values: • **Offset** - default = 0 • **Scale** - default = 1

Tutorial: Backgrounds for visualizing your 3D models Click here (1.7.1) to watch

Related Commands

GradientBkgOff (6.15.18) - switches the gradient background color of the current viewport off. Restores the default view background. GradientBkgOn (6.15.19) - restores the default gradient background of the current viewport. MatchPerspective (6.21.7) - matches the current model space perspective view with a background image by selecting 3 or more matching point pairs. Render (6.26.22) - generates a photorealistic rendering using materials and lights through a dialog box. -Render (6.26.23) - generates a photorealistic rendering using materials and lights through the command bar. View (6.31.1) - creates and sets named views through the Drawing Explorer.

Figure 6.198

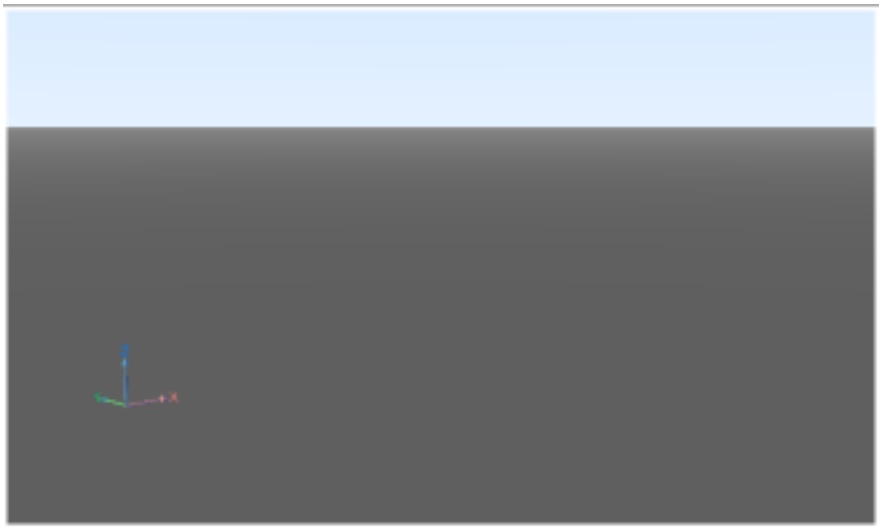


Figure 6.199

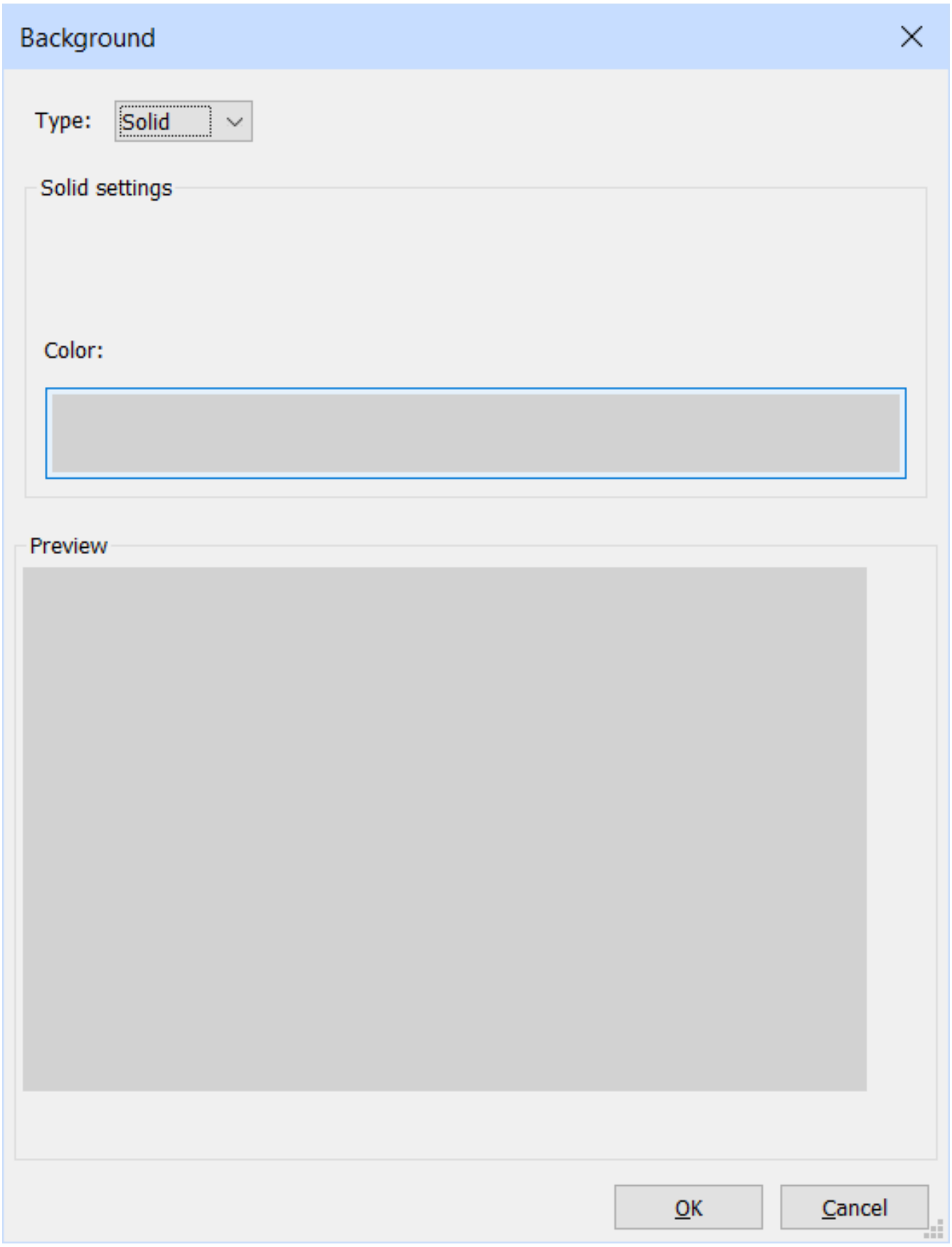

Background
Type:
Solid
Solid settings
Color:
Preview
OK
Cancel

Figure 6.200

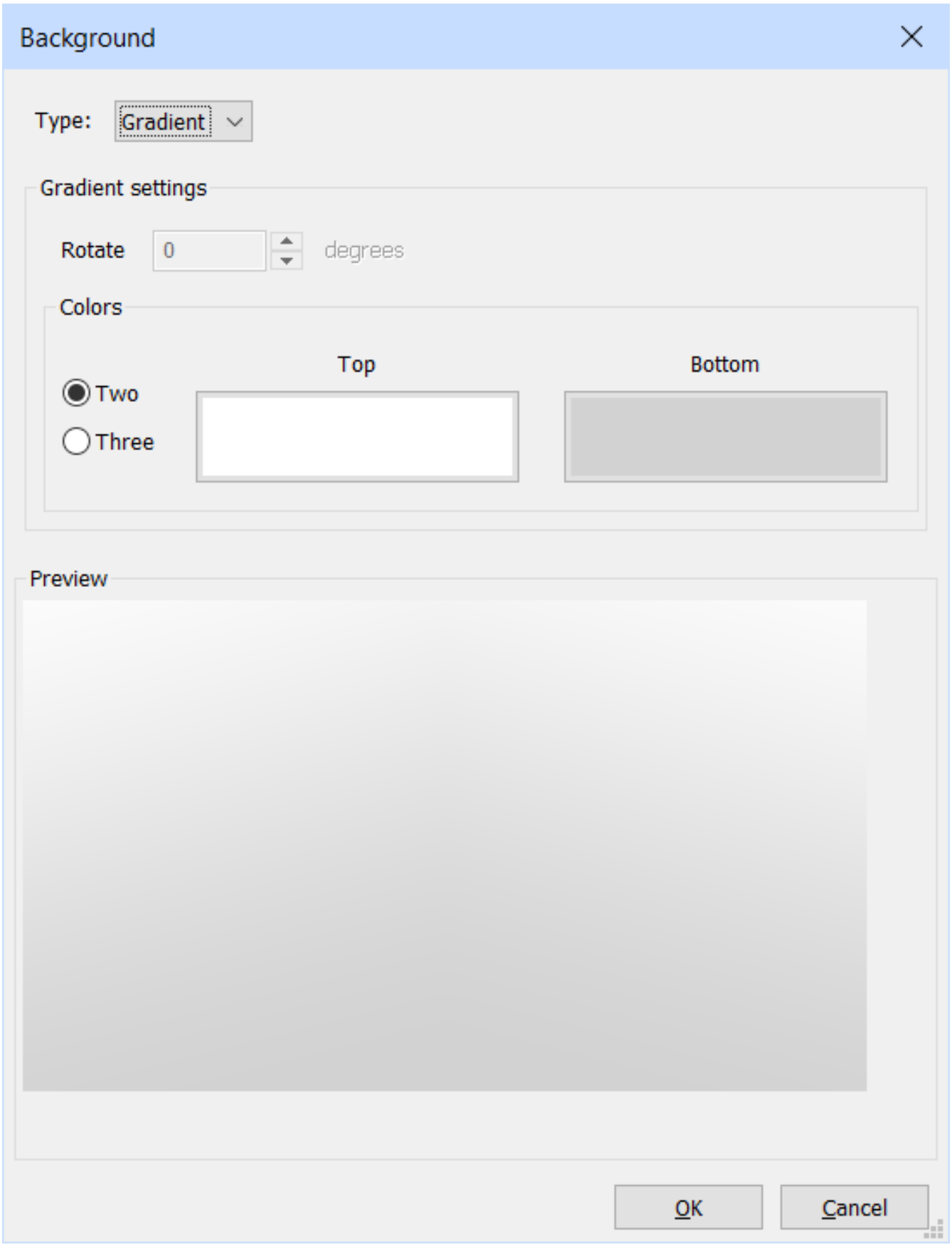

Background
Type:
Gradient
Gradient settings
Rotate
0
degrees
Colors
Top
Bottom
Two
Three
Preview
OK
Cancel

Figure 6.201


Background
Type:
Gradient
Gradient settings
Rotate
0
degrees
Colors
Top
Middle
Bottom
Two
Three
Preview
OK
Cancel

Figure 6.202

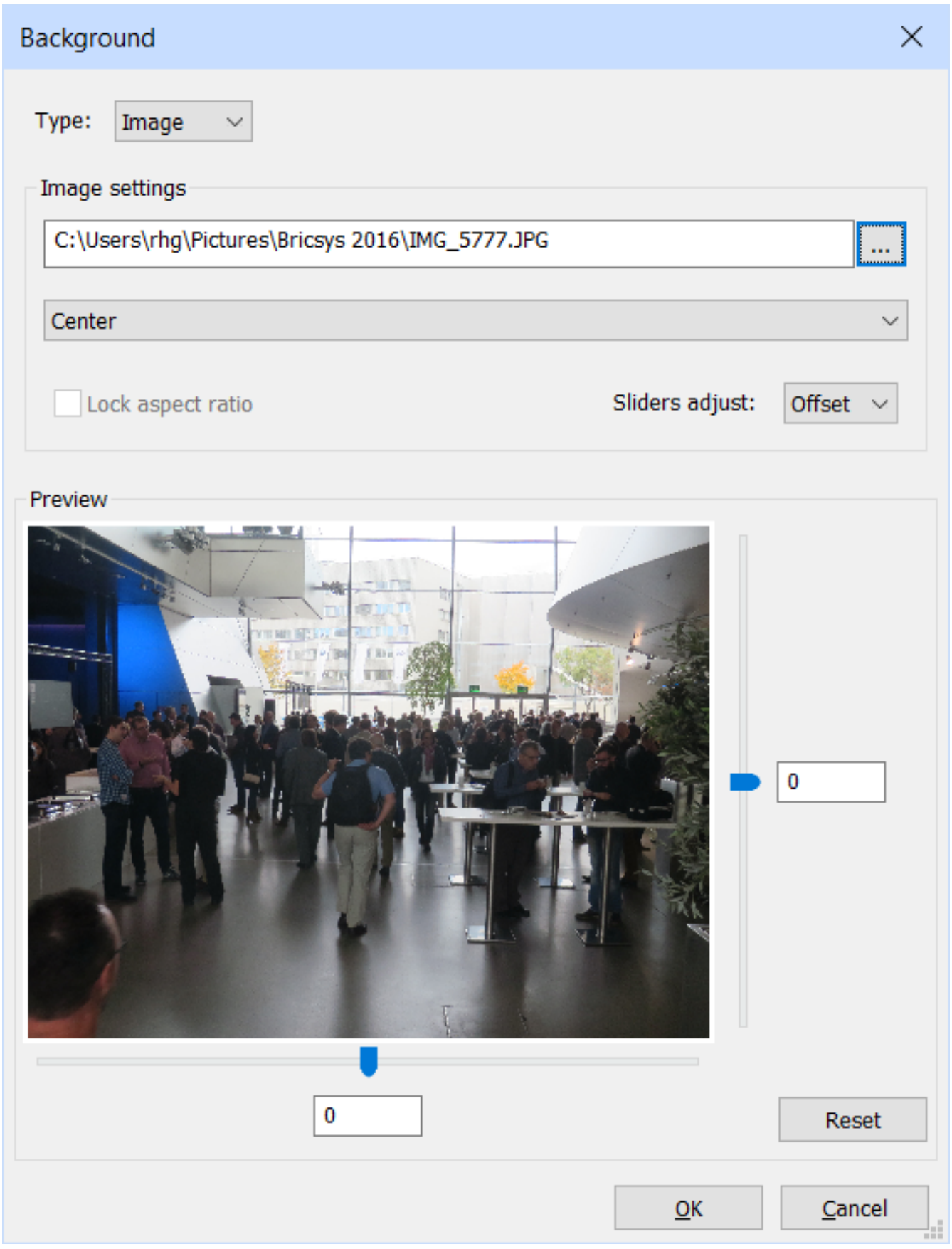

Background
Type:
Image
Image settings
C:\Users\rhg\Pictures\Bricsys 2016\IMG_5777.JPG
...
Center
Lock aspect ratio
Sliders adjust:
Offset
Preview
0
0
Reset
OK
Cancel

Figure 6.203



Figure 6.204



Figure 6.205



Figure 6.206



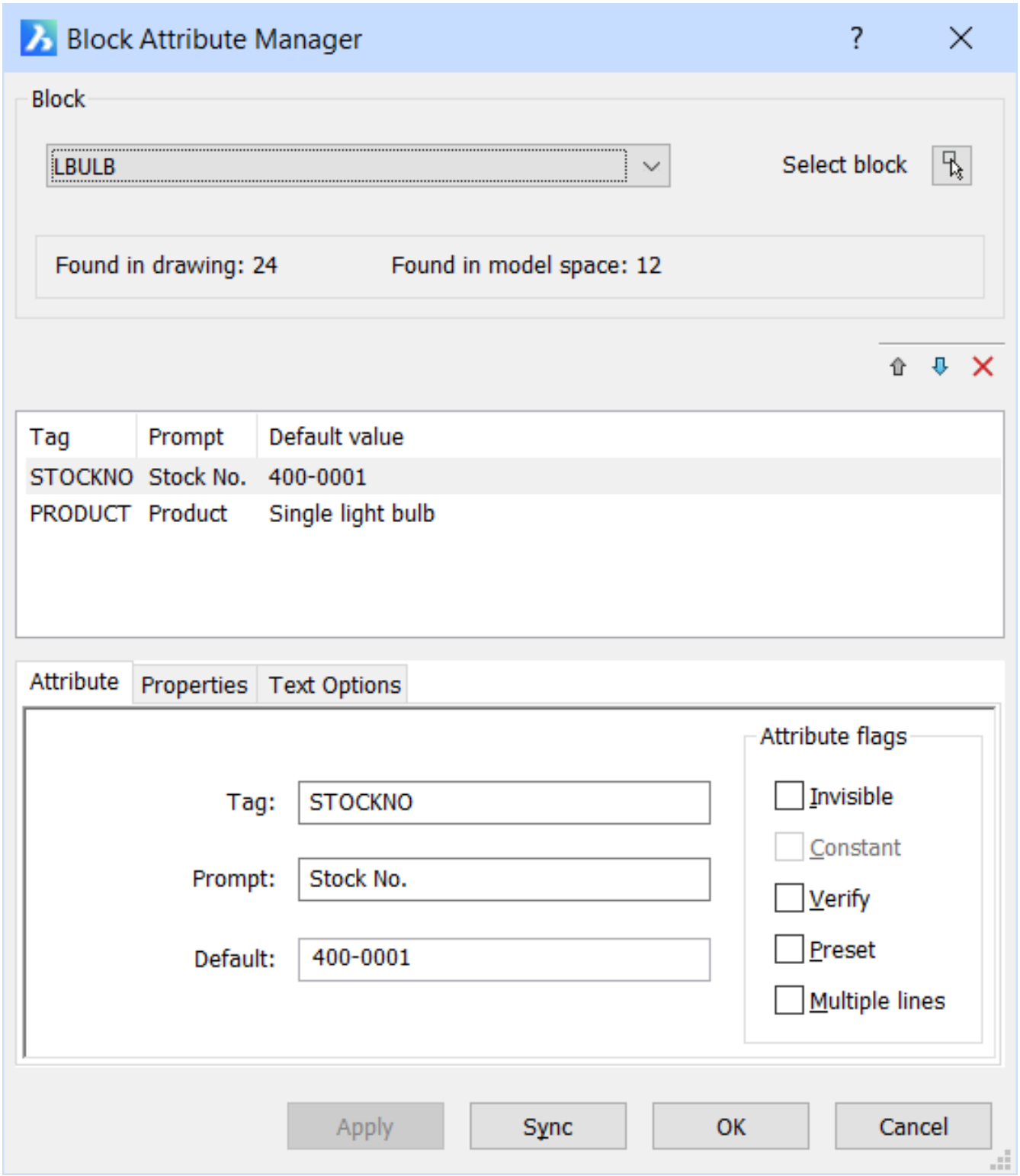
6.6.2 BattMan

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits all aspects of attributes in a block definition (short for "block attribute manager), and then optionally applies the changes to all blocks of the same name in the drawing.

Accessing the Command

 command bar: **battman** menu bar: **Tools | Attributes | Block Attributes Manager** toolbar: **Attributes** | ribbon:**Insert | Blocks | Block Attribute Manager** This command works only when the drawing contains attributes. : **battman** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/battmandlg.png})

Figure 6.207


Block Attribute Manager
Block
LBULB
Select block
Found in drawing: 24
Found in model space: 12
Tag
Prompt
Default value
STOCKNO
Stock No.
400-0001
PRODUCT
Product
Single light bulb
Attribute
Properties
Text Options
Attribute flags
Tag:
STOCKNO
Prompt:
Stock No.
Default:
400-0001
Invisible
Constant
Verify
Preset
Multiple lines
Apply
Sync
OK
Cancel

Command Options

Option	Description
Block	Select a block from the drawing whose attributes you want to edit: 1. Do one of the following: • Click the **Block**droplist, then select a block in the list. • Click the **Select Block** button ( ).The Block Attribute Manager dialog box temporarily closes to let you select a block in the drawing; prompts you: **Select a block** - pick one block in the drawing 2. Select an attribute from the Attribute list to edit.
Attribute list	Choose an attribute from the list to edit it: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/E • - moves the attribute up the list • - moves the attribute down the list • - deletes the attribute from the block
Attribute tab	Click the **Attribute** tab to edit thedefinition of the selected attribute: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/E See the **AttDef (6.4.27)** command for the meaning of these options.
Properties tab	Click the **Properties** tab to edit theproperties of the selected attribute: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/ba properties-tab.png})
Text Options tab	Click the **Text Options** tab to edit the text properties of the selected attribute: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/ba textoptions-tab.png}) See the **Style (6.27.53)** command for the meaning of these options.
Apply button	Applies the changes to the block definition, and keeps the dialog box open; choose another block whose attributes you wish to edit.
Sync button	Applies the changes to all instances of the same blockin the drawing.
OK button	Saves the changes you made, and then closes the dialog box.
Cancel button	Undoes all changes you made, and then closes the dialog box.

Figure 6.208

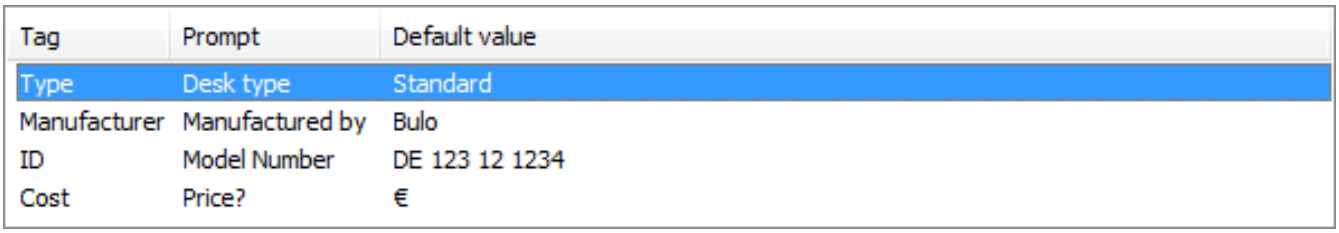
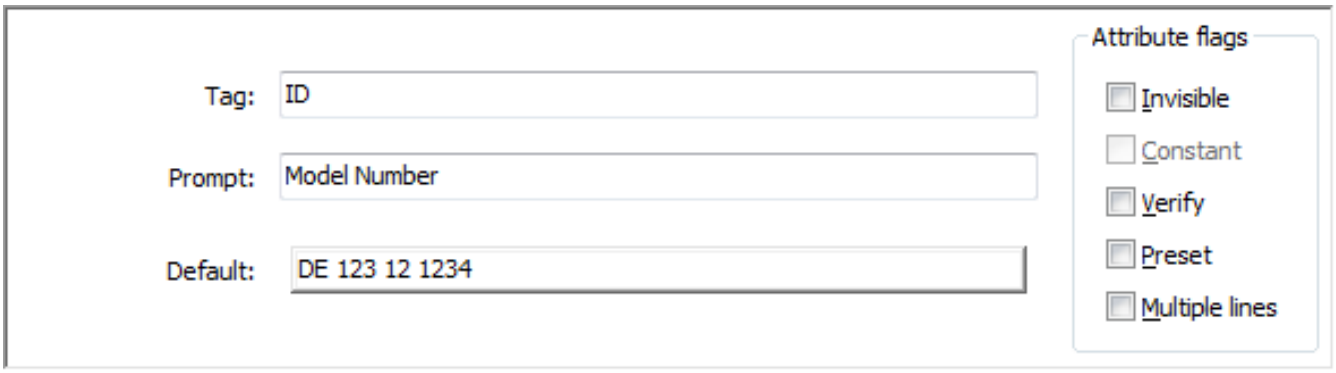


Figure 6.209



Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttDisp (6.4.35) - toggles the display of attribute text. AttEdit (6.4.36) - edits attribute values and properties. -AttExt (6.4.29) - extracts attributes from drawings to data files through the command bar. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. DataExtraction (6.10.1) - exports entity properties, block attributes and drawing information to a an external file. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. InsertAligned (6.17.15) - inserts blocks repeatedly, and inserts mirrored blocks. Properties (6.24.49) - edits attribute values and properties.

Figure 6.210

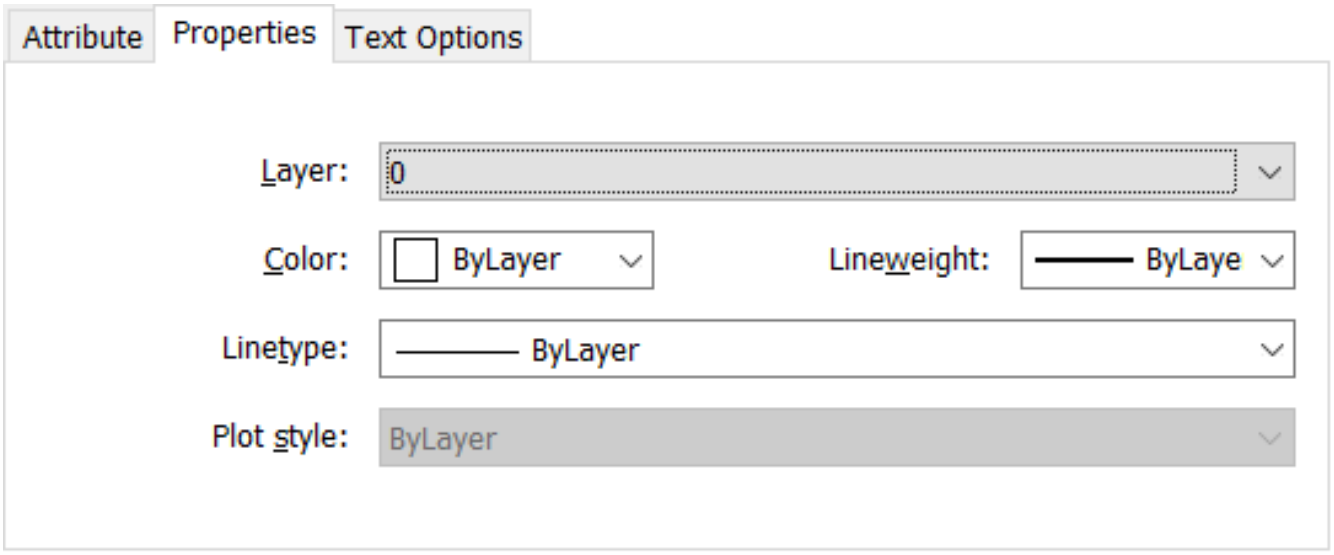


Figure 6.211

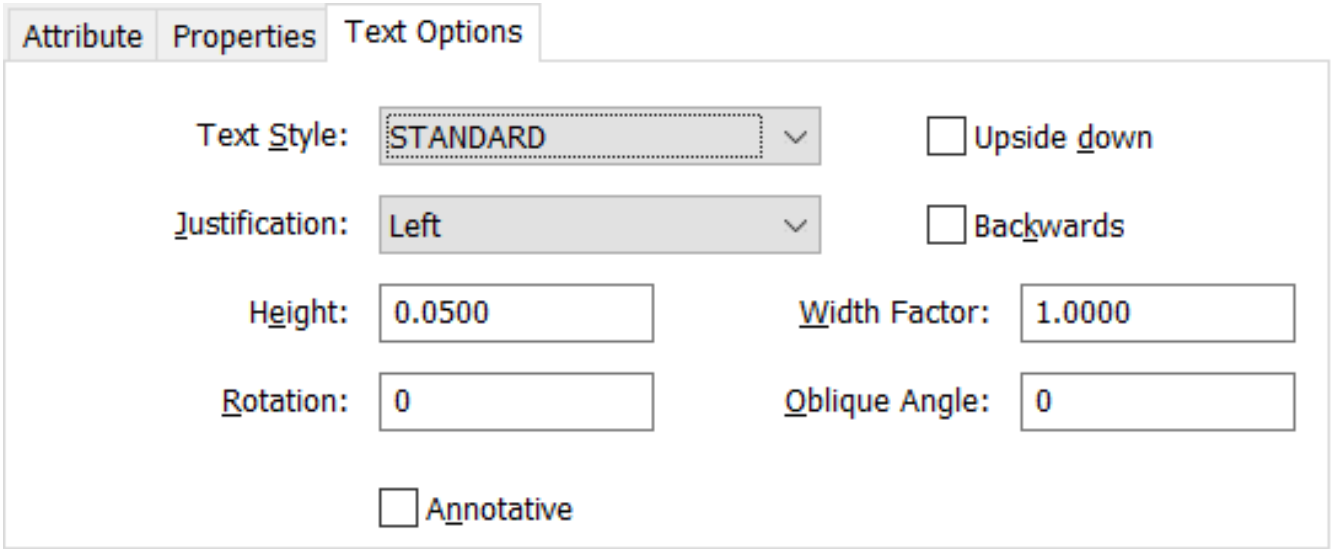


Figure 6.212



6.6.3 Base

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Changes the drawing's base (insertion) point for when it inserted into other drawings, usually as a block or external reference.

Accessing the Command

 command bar: **base** transparent: **'base** alias: **ba** menu bar: **Settings | Base Point** toolbar: **Settings |**  ribbon: **Settings | Base Point** : **base Prompts you in the command bar:** Base point for inserting this drawing <0.0000,0.0000,0.0000>: **(Enter x,y,z coordinates to relocate the base point for the drawing.)**

Command Options

Option	Description
Base point for inserting this drawing	Specifies x,y,z coordinates.The coordinates of the base point are in the current UCS. When this drawing is inserted into another drawing, these coordinates specify the insertion point, whether this drawing is inserted as a block or as an external reference. This allows the inserted drawing to match the destination drawing.

Related Commands

-Insert (6.17.14) - inserts DWG drawing files into the current drawing through the command bar. Insert (6.17.13) - inserts DWG drawing files into the current drawing through a dialog box. WBlock (6.33.1) - writes all or part of the drawing to a DWG file. XRef (6.34.8) - inserts DWG files as external references.

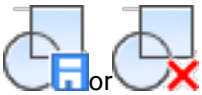
Figure 6.213



6.6.4 Bclose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes a Block Edit session and saves or discards the changes.

Accessing the Command


or **Keyboard**: bclose **Toolbar**: Block Edit **bclose: Prompts you:** Enter option [Save/Discard block editing changes] <Save>: **(Choose an option.)**

Command Options

Option	Description
Save	Save the changes and close the Edit Block session.
Discard	Discard the changes and close the Edit Block session.

Related Commands

Bedit (6.6.7) -Edits the blocks' entities in a temporary drawing session, using the local coordinate system of the block.

Figure 6.214



6.6.5 BHatch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Fills closed areas with repeating patterns or gradated colors; displays the Hatch and Gradient dialog box.

Accessing the Command

command bar: **BHatch :bhatch** Displays a dialog box. See the Hatch (6.16.1) and Gradient (6.15.17) commands.

Related Commands

Boundary (6.6.15) and -Boundary (6.6.16) place a polyline boundary inside closed areas. Hatch (6.16.1) - place hatch patterns through a dialog box. -BHatch (6.6.6) and -Hatch (6.16.2) - places hatch patterns through the command bar. HatchGenerateBoundary (6.16.5) - generates a boundary around a hatch or gradient fill. Properties (6.24.49) - changes the properties of hatch patterns. Settings (6.27.23) - specifies the default parameters for hatch patterns. Solid (6.27.37) - draws 3- and 4-side solid-filled areas.

Figure 6.215



6.6.6 -BHatch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Fills closed areas with repeating patterns; displays prompts at the command bar.

Accessing the Command

command bar: **-bhatch** : **-bhatch** Prompts you in the command bar. See the -Hatch (6.16.2) command.

Related Commands

Boundary (6.6.15) and -Boundary (6.6.16) place a polyline boundary inside closed areas. BHatch (6.6.5) and Hatch (6.16.1) - place hatch patterns through a dialog box. -Hatch (6.16.2) - places hatch patterns through the command bar. HatchGenerateBoundary (6.16.5) - generates a boundary around a hatch or gradient fill. Properties (6.24.49) - changes the properties of hatch patterns. Settings (6.27.23) - specifies the default parameters for hatch patterns. Solid (6.27.37) - draws 3- and 4-side solid-filled areas.

Figure 6.216



6.6.7 BEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits the entities that make up blocks in a temporary drawing session called the Block Editor, which uses the local coordinate system of the block. Optionally creates a new block.

Accessing the Command

 Keyboard: **bedit** Menu: **Tools | Edit Block** Toolbar: **Block Edit** Quad Tab: **Edit** Ribbon Tab | Panel: **Home | Blocks and Insert | Blocks :bedit** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19 dlg.png})

To edit an existing block, do the following:

1. Select a block in the list, then double click or click the OK button. The Edit Block session starts. The **Block Edit** toolbar displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/BlockEditToolb
2. Edit the block.
3. Do one of the following:
   - On the **Block Edit** toolbar:
     - : Save the changes and close the Edit Block session.
     - : Discard the changes and close the Edit Block session.
   - Execute the BCLOSE (6.6.4) command.

To create a new block, do the following:

1. Type a name in the **Block to create or edit** field.
2. The Edit Block session starts. The **Block Edit** toolbar displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
3. Create the block entities.
4. Do one of the following:
   - On the **Block Edit** toolbar:
     - : Save the changes and close the Edit Block session.
     - : Discard the changes and close the Edit Block session.
   - Execute the BCLOSE (6.6.4) command.

Figure 6.217

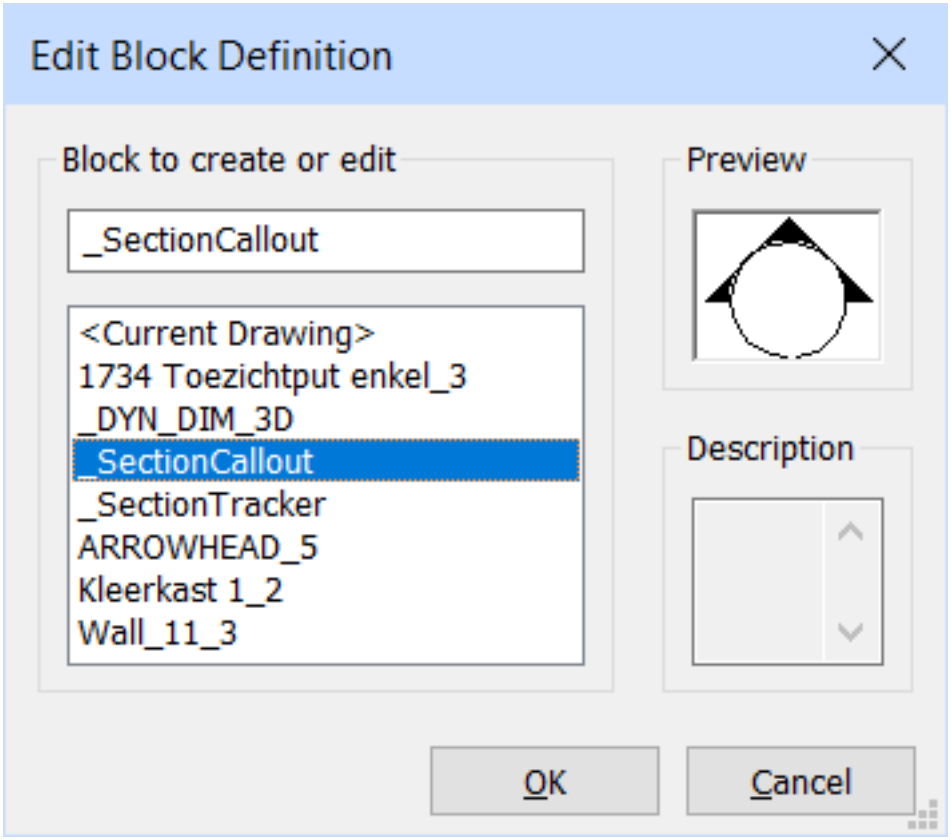


Figure 6.218

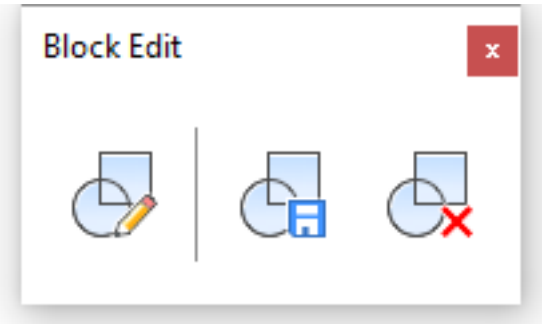
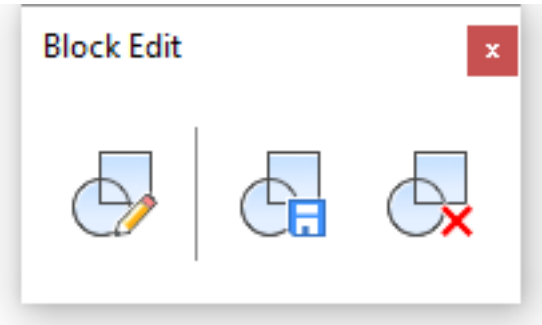


Figure 6.219



Related Commands

Bclose (6.6.4) - closes a Block Edit session and saves or discards the changes. Block (6.6.11) - groups entities into a block (symbol), which can be reused in drawings. RefEdt (6.26.12) - edits block references and externally-referenced drawings through a dialog box.

Figure 6.220



6.6.8 -BEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Prompts for the name of a name at the command line, and then opens the Block Editor environment

Accessing the Command

Keyboard: **-bedit :-BEdit Prompts you:** Block name or ? to list existing blocks: **(Enter the name of a block, or else enter ? to see the list of block names in the current drawing)**

Command Options

Option	Description
Block name	Enter the name of a block; or enter an unused name to create a new block
? to list existing blocks	Lists the names of blocks in the drawing; prompts you: **Block(s) to list [* for all]: (Use wildcards, such as ? and *, to specify the block names to list)** Enter * to see the names of all blocks in the current drawing.

Procedures

<links to other articles in the BricsCAD, BricsCAD BIM or BricsCAD Mechanical sections.>

Related Commands

BEdit (6.6.7) - opens the Block Editor environment through a dialog box. **BClose (6.6.4)** - closes the Block Editor environment, optionally saving the changes.

Figure 6.221



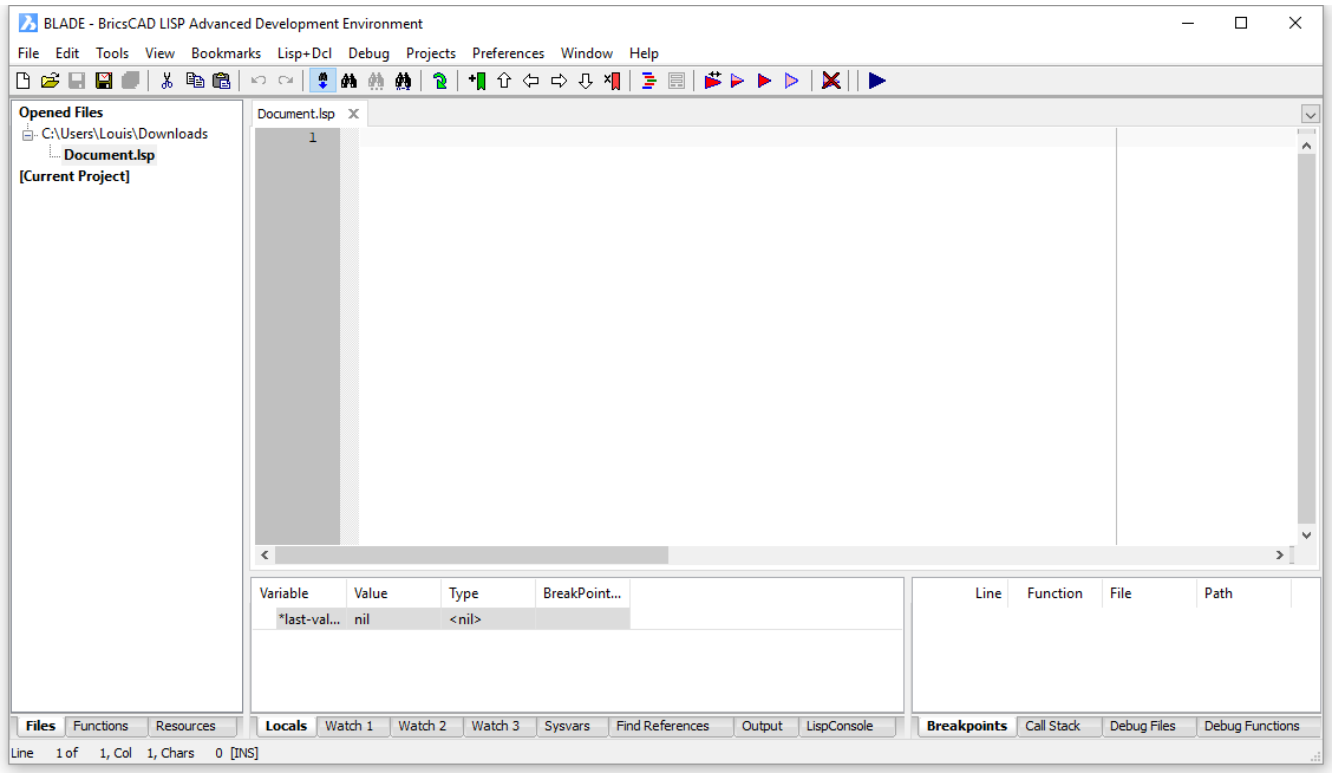
6.6.9 Blade

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the interactive environment for editing and debugging LISP applications (short for ”BricsCAD LISP advanced development environment”).

Accessing the Command

command bar: **blade** transparent: **'blade** : **blade** Displays BLADE - BricsCAD LISP Advanced Development Environment dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/BLADEdialog.png})

Figure 6.222



Command Options

No command options.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. VbaLoad (6.32.2) loads VBA projects. VbaMan (6.32.4) - manages VBA projects. VbaNew (6.32.5) - starts a new VBA project. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. Vbalde (6.32.1) - opens the VBA integrated development environment. VbaUnload (6.32.9) - unloads VBA projects. Security (**??**) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.

Figure 6.223



6.6.10 Blipmode

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Toggles the display of blips that are made as you pick points in the drawing.

Accessing the Command

command bar: **blipmode** transparent: **'blipmode** alias: **bm** : **blipmode Prompts you in the command bar:** Blipmode is currently off: ON/Toggle/<Off>: **(Enter an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/C
Examples of blips made when the user picks points in the drawing.

Figure 6.224



Command Options

Option	Description
ON	Turns on the display of blips.Use the Redraw command to erase blips from the screen.
Toggle	Turns on blip mode, if off; turns off, if on.
Off	Turns off the display of blips.

Related Commands

Redraw (6.26.9) - removes blips from the screen. Settings (6.27.23) - specifies the default parameters for blipmode.

Figure 6.225



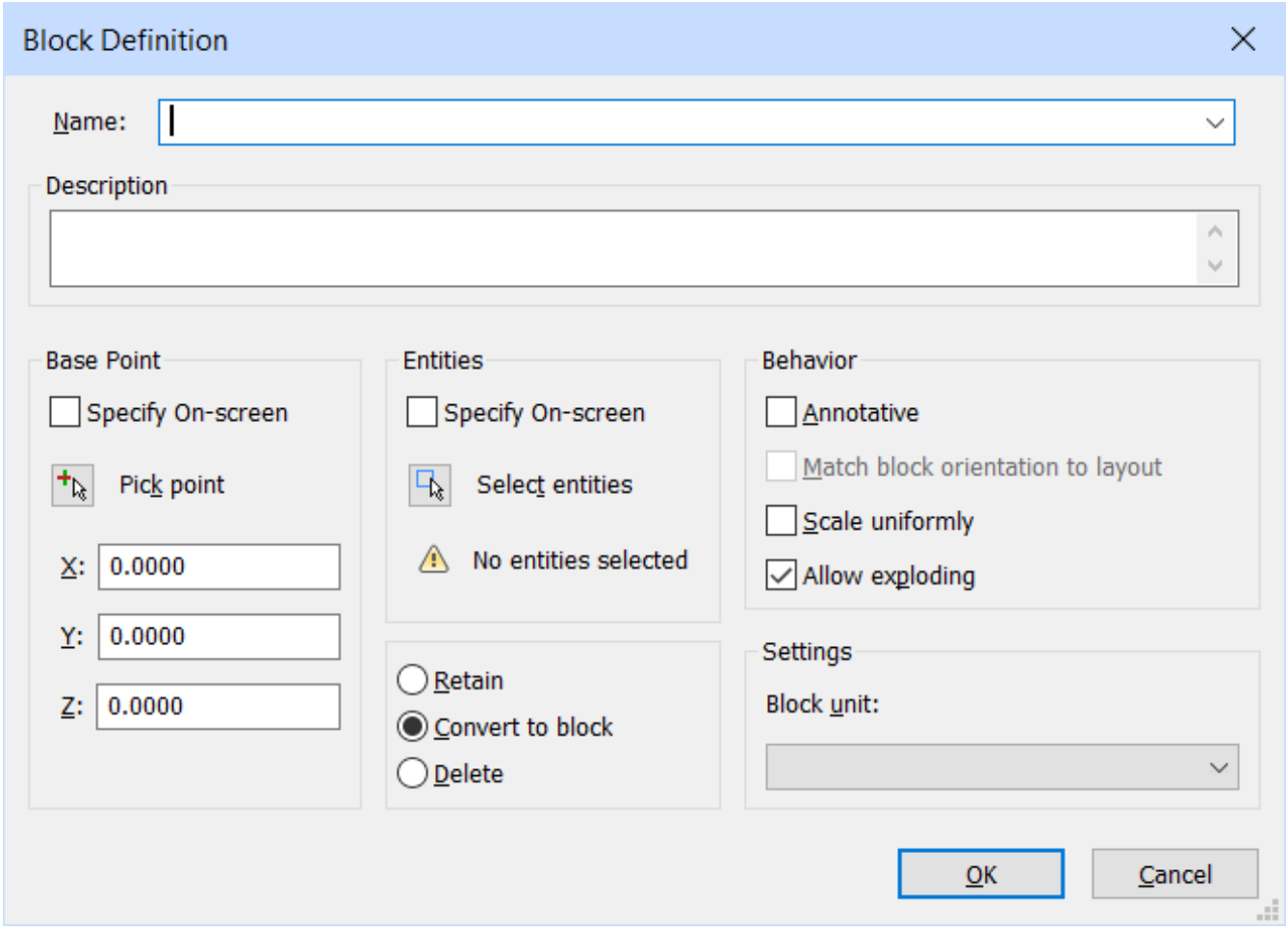
6.6.11 Block

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Groups entities into a block (symbol), which can be reused in drawings.

Accessing the Command


command bar: **block** alias: **b** Menu bar: **Tools | Create Block** toolbar: **Tools |**  ribbon:**Insert | Blocks | Create Block** : **block** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Bdlg.png})

Figure 6.226


Block Definition
Name:
Description
Base Point
Specify On-screen
Pick point
X: 0.0000
Y: 0.0000
Z: 0.0000
Entities
Specify On-screen
Select entities
No entities selected
Retain
Convert to block
Delete
Behavior
Annotative
Match block orientation to layout
Scale uniformly
Allow exploding
Settings
Block unit:
OK
Cancel

Command Options

Option	**Description**
Name	Specifies the name of the block.
Description	(optional) Describes the block.
Base Point Options	**Description**
Specify On-Screen	Specifies the block's base point, the point at which it will be inserted into drawings with the **Insert (6.17.13)** command. Choose how you want to specify the base point's coordinates: • **On** - you will be prompted later to define the base point after you click the **OK** button; the Pick Point button and X, Y, Z fields are grayed out (unavailable) • **Off** - click the Pick Point button, or else enter x, y, z coordinates in the X, Y, Z fields.
 Pick point	Pick a point in the drawing for the block's base point: 1. Click the **Pick point**  button. 2. The Block Definition dialog box temporarily closes, and the program prompt you to specify the base point on screen: **Insertion point for new block or [Annotative]** - **(pick a point, or else enter x,y,z coordinates, and then press Enter.)**
X, Y, Z	Enter x, y, z coordinates for the base point in the X, Y, and Z fields. The z coordinate is usually optional.
Entities Options	**Description**
Specify On-screen	The Entities section selects the entities that make up the block.Choose how you want to select the entities: • **On** - you will be prompted later to select the entities after you click the **OK** button; the Select Entities button is grayed out • **Off** - click the Select Entities button

 Select Entities

Select one or more entities in the drawing:

1. Click the **Select Entities**  button
2. The Block Definition dialog box temporarily closes, and the program prompt you to specify the base point on screen:

Select entities for block- (select one or more entities using a selection method) Select entities for block- (select more entities, or else press Enter to return to the dialog box) Once entities are selected, decide what happens to them once they are turned into a block. Choose an option:

- **Retain** - selected entities are retained as entities, and the block definition is created in the drawing
- **Convert to block** - selected entities are converted to a block
- **Delete** - selected entities are deleted after the block definition is created

The Convert to Block option is the most efficient one.

Behavior Options
Description

Annotative

Sets the annotative property of the block. This kind of block should be created when the annotation scale in model or paper space is 1:1. By being annotative, the block automatically scales itself according to the current annotation scale factor. Choose whether you want block to follow annotative scaling:

- **On** - block will scale itself to whatever annotative scale is in effect; the Scale Uniformly option is grayed out (unavailable)
- **Off** - block follows the scale factor you give the block during the **Insert (6.17.13)** command

Match block orientation to layout

Determines if annotative blocks match the orientation of the layout:

- **On** - annotative blocks display upright, regardless of the orientation of the viewport
- **Off** - annotative blocks match the orientation of the viewport

Scale Uniformly

Determines if blocks can be scales non-uniformly. This option is not available to annotatively-scaled blocks. Choose one:

- **On** - X, Y, and Z scale factors of blocks are the same as one another; this prevents blocks from being distorted
- **Off** - blocks can be inserted with different X, Y, and Z scale factors; this is useful for unit blocks that are meant to become a variety of objects, such as differently sized table tops

Allow Exploding	Determines if users can explode the block after it is inserted. When a block is exploded, it loses its block status, and the individual entities can be edited. (To edit the entities of an unexploded block, use the **BEdit (6.6.7)** command.) Choose one: • **On** - block can be exploded after being inserted, with the **Explode (6.13.13)** command • **Off** - blocks cannot be exploded; you can change this property with the **Explorer (6.13.11)** command, Blocks section.
Settings Option	**Description**
Block Unit	Scales the block correctly when the block definition is inserted in a drawing whose units are different from the drawing in which the block was created. More specifically, in which the **INSUNITS**variable differs. 1. Click the Block Unit droplist. 2. Choose a unit from the list (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/blockunit-droplist.png})
OK	Closes the dialog box, and if necessary prompts you at the command line for the base point and the entities to include in the block, then saves the block definition in the drawing.
Cancel	Exits the dialog box without saving the block definition; no block is created.

Procedures

- General procedure to create a block (1.22.5)
- To create a block in a drawing (1.22.5)
- To create a block using the Blocks Explorer (1.22.5)
- To save a block to a separate file (1.22.5)

Related Commands

BEdit (6.6.7) - edits blocks Blockify -Automatically detects equally shaped solids in a model and replaces them by block references.Searches the drawing for an identical set of a 2D entities and replaces them by block references. CopyBase (6.9.51) - copies entities to the Clipboard with a base point. ExpBlocks (6.13.11) - opens the Blocks section of the Drawing Explorer dialog Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. InsertAligned (6.17.15) - inserts blocks repeatedly, and inserts mirrored blocks. Properties (6.24.49) - edits attribute values and properties. WBlock (6.33.1) - writes blocks and other drawing parts to files on disk through a dialog box -WBlock (6.33.2) - writes blocks and other drawing parts to files on disk through a dialog box

Figure 6.227

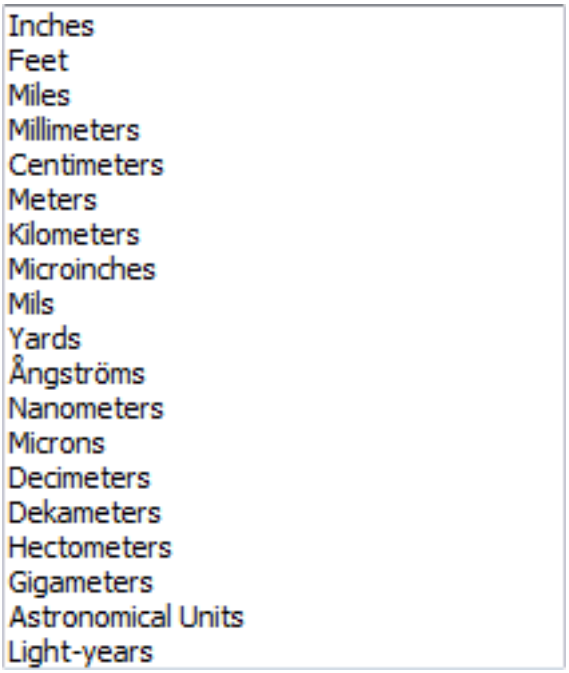

Inches
Feet
Miles
Millimeters
Centimeters
Meters
Kilometers
Microinches
Mils
Yards
Ångströms
Nanometers
Microns
Decimeters
Dekameters
Hectometers
Gigameters
Astronomical Units
Light-years

Figure 6.228



6.6.12 -Block

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Groups entities into a block (symbol).

Accessing the Command

command bar: -**Block** alias: **-b** : -**block Prompts you in the command bar:** Name for new block or [? to list existing blocks(?)]: **(Enter a name, or ?)** Insertion point for new block or [Annotative]: **(Pick a point, or enter x,y coordinates)** Select entities for block: **(Select one or more entities)** Select entities for block: **(Select more entities, or press Enter to exit command)**

Command Options

Option	Description
Name for new block	Specifies the name of the block.
? to list existing blocks	Lists the names of blocks already in the drawing, so that you don't use an existing name by accident.Prompts you: **Block(s) to list [* for all]** - **(use wildcards to specify groups of names, or else press Enter to display all names.)** The program provides this report:

Defined blocks. "BSWITCH" "CONNECT" "DOMESW" "ELECTRIC" "FUSE40"

User Blocks	External References	Dependent Blocks	Unname
5	0	0	0

When the list of block names is long, press **F2** to see it in the Text window. If you enter an existing name, the program ask you if you want to redefine the block: **The block ground already exists. Do you want to redefine it? [Yes/No] <No>** - **(enter an option)**:

- **Yes** - changes the block definition to the new set of entities you select in the next steps. **Caution:** all instances of this block name will be changed!
- **No** - exits the command

Insertion point for new block	Specifies the block's base point, the point at which it will be inserted into drawings with the Insert command. Prompts you: **Insertion point for new block or [Annotative]** - **(enter an option)**: • **Insertion point** - pick a point in the drawing, or else enter x,y, and z coordinates • **Annotative** - specifies that the block will follow annotative scaling; prompts you: **Create annotative block[Yes/No] <Yes>** - **(enter an option)**: • **No** - avoids making the block annotative • **Yes** - gives the block the annotative property, and then goes on to the following prompt: **Orient relative to sheet in paper space viewports[Yes/No] <No> - (determines if annotative blocks match the orientation of the layout)**: • **On** - annotative blocks display upright, regardless of the orientation of the viewport • **Off** - annotative blocks match the orientation of the viewport
Select entities for block	The Z coordinate is optional, and is usually left at 0. Selects the entities that make up the block; you may use any selection method, including entity snaps and windowed selection. Repeats the command: **Select entities for block** - **(press Enter to finish selecting entities and to exit the command.)** The entities disappear from view, but are now part of the block; use the **Oops (6.23.7)** command to return the entities to the drawing.

Procedures

- General procedure to create a block (1.22.5)
- To create a block in a drawing (1.22.5)
- To create a block using the Blocks Explorer (1.22.5)
- To save a block to a separate file (1.22.5)

Related Commands

BEdit (6.6.7) - edits blocks Blockify -Automatically detects equally shaped solids in a model and replaces them by block references.Searches the drawing for an identical set of a 2D entities and replaces them by block references. CopyBase (6.9.51) - copies entities to the Clipboard with a base point. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. InsertAligned (6.17.15) - inserts blocks repeatedly, and inserts mirrored blocks. WBlock (6.33.1) - writes blocks and other drawing parts to files on disk through a dialog box. -WBlock (6.33.2) - writes blocks and other drawing parts to files on disk through a dialog box.

Figure 6.229



6.6.13 Blockify

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Blockify automatically detects equally shaped solids in a model and replaces them by block references. It searches the drawing for an identical set of entities (2D and 3D) and replaces them by block references. Youtube

Accessing the Command


Keyboard: **blockify** Menu: **Tools | Blocks | Blockify** Quad Tab**: Modify| Blockify** Ribbon Tab | Panel:**Blocks | Blockify : blockify Prompts you:** Select input entities* or [Find all groups] <Find all groups>: **(Select an option.)** BricsCAD reports the number of blocks created in the command line. Press F2 if the command line is closed. * Includingtext, multiline text, solids, traces, hatches, leaders, multileaders and existing block references.

Command Options

Option	Description
Select input entities	Searches the drawing for identical sets of a selection of entities* (2D and 3D). Compose a set of entities, then press Enter. Prompts you: **Select search space or [Entire drawing] <Entire drawing>:** • Entire drawing: searches the entire drawing • Select search space: select the part of the drawing to be analyzed, then press Enter.
Find all groups	Analyses the model space of the current drawing and automatically detects equally-shaped solids and replaces them by block references.Prompts you: **Select search space or [Entire drawing] <Entire drawing>** - **(choose an option.)** • Entire drawing: searches the entire drawing • Select search space: select the part of the drawing to be analyzed, then press Enter.

Related Commands

-block (6.6.12) -Groups entities into a block (symbol). block (6.6.11) -Groups entities into a block (symbol) through a dialog box. ExpBlocks (6.13.11)- opens the Blocks section of the Drawing Explorer dialog

Figure 6.230



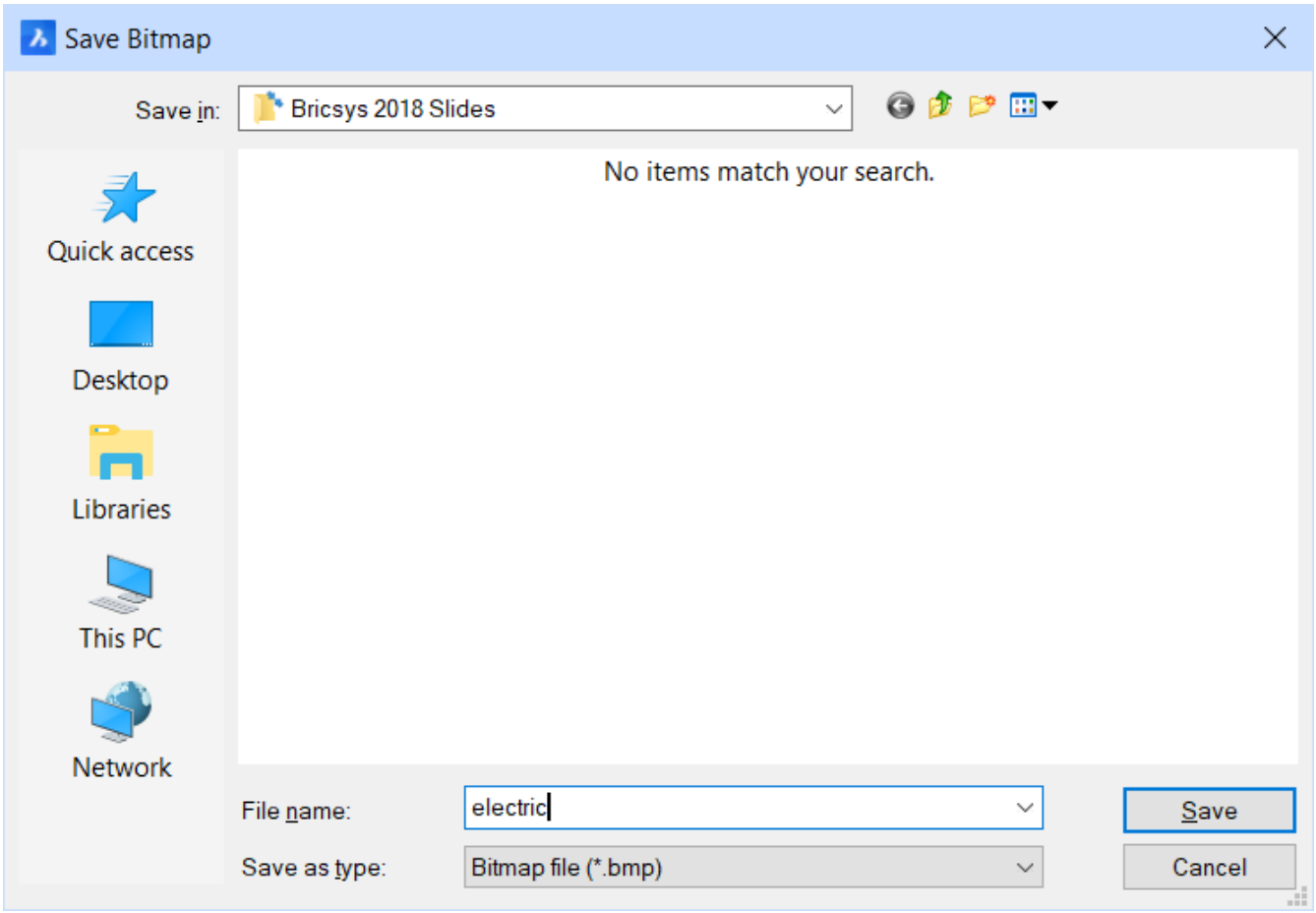
6.6.14 BmpOut

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Exports the current viewport as a BMP (bitmap or raster) file.

Accessing the Command

command bar: **bmpout** menu bar: **File | Export**, and then choose BMP from the File of Type droplist ：**bmpout** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/bmpout-dlg.png})

Figure 6.231



Command Options

Option	Description
Save in	Selects the folder and drive in which to save the file.
File name	Specifies the name of the file.
File of type	Specifies the format in which to save the image.
Save	Saves the content of the current viewport as a raster image.

Related Commands

Export (6.13.15) - exports drawings in BMP, DWF, DXF, DWT, PDF, WMF, EMF, SLD, and older DWG formats. MSlide (6.21.31) - exports drawings in WMF, EMF, and SLD formats. WmfOut (6.33.12) - exports drawings in WMF, EMF, and SLD formats.

Figure 6.232



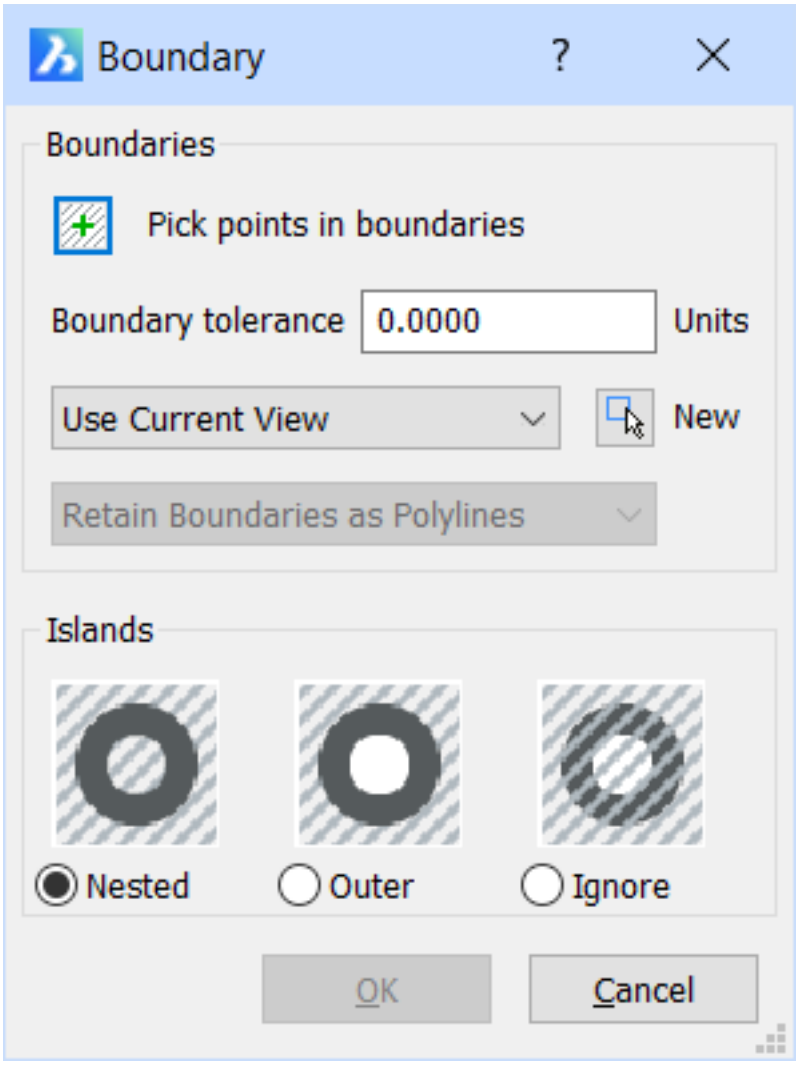
6.6.15 Boundary

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a polyline that forms a boundary around closed areas, through a dialog box.

Accessing the Command

 command bar: **boundary** aliases: **bo, bpoly** menu bar: **Draw | Boundaries | Boundary Polyline** toolbar: **Draw | Boundaries|**  ribbon:**Draw | Draw | Polygons | Boundary** : **boundary** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/B/boundary-dlg.png}) Choose options, and then click **OK**.

Figure 6.233


Boundary
?
Boundaries
Pick points in boundaries
Boundary tolerance
0.0000
Units
Use Current View
New
Retain Boundaries as Polylines
Islands
Nested
Outer
Ignore
OK
Cancel

Command Options

Boundaries	Description
Pick points in boundary	Specifies the closed areas in which to place the boundary; temporarily dismisses the dialog box, and then prompts you at the command bar: **Pick a point to define a boundary or hatch area or [Select entities/Undo]:**- **(enter an option)**: • **Pick a point to define a boundary or hatch area** - pick a point inside an area closed by entities (ignore the reference to hatch area) • **Select entities** - allows you to select individual entities forming a closed area • **Undo** - un-selects the last selected entities **Notice** that BricsCAD highlights draws the boundary, allowing you to preview it.The prompt repeats, allowing you to choose additional areas, entities, or undo the selection. When done selecting areas, press Enter return to the dialog box.
Boundary tolerance	Determines the largest gap that can exist in the boundary for the program to consider the area closed; enter a value: • 0 - no gaps allowed to exist
Boundary set	Specifies where BricsCAD should search for entities that make up the boundary. Select an option from the droplist: • **Current Viewport** - searches all entities in the current viewport • **Existing Set** - searches only the current selection set for possible boundaries
New	Creates a new selection set of entities that make up the boundary; temporarily dismisses the dialog box, and then prompts you at the command bar: **Select entities** - **(use any selection method to choose one or more entities)** The prompt repeats, allowing you to choose additional entities. When done selecting entities, press Enter to return to the dialog box.
Retain Boundaries as	Specifies how to handle the boundary entity; currently, the only option is to make the boundary out of a polyline, and then retain the boundary after you exit the dialog box.

Islands	Determines how the boundary search algorithm treats "islands", boundaries that are found inside the boundary. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Choose one: • **Nested** -the program treats every island as a boundary  • **Outer** - the program creates a boundary of only the outermost entities  • **Ignore** - the program creates one boundary, like the Outer option; the Ignore option means only something for hatches and gradients, not boundaries 

Procedures

- Drawing polylines (1.17.12)
- Creating boundary polylines (1.17.12)

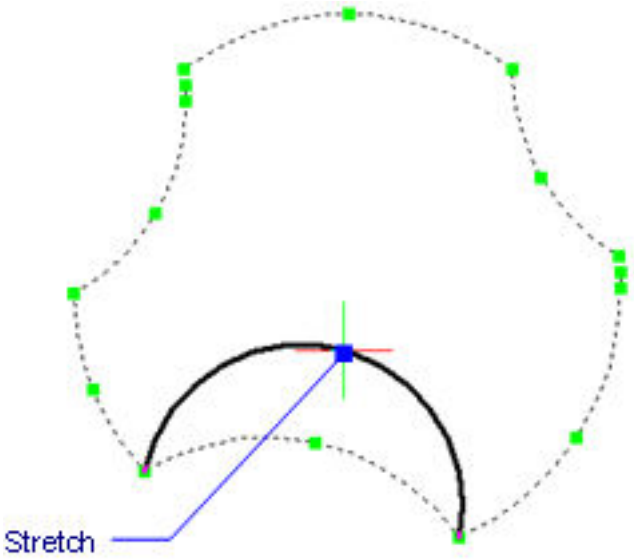
Grips Editing

Boundaries can be edited directly through grips:

1. Select the boundary. Notice that it has many grips.
2. Drag a grip to stretch the boundary.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image418.jpg})

Figure 6.234



Related Commands

BHatch (6.6.5) and Hatch (6.16.1) - place boundaries and hatch patterns through a dialog box. -BHatch (6.6.6) and -Hatch (6.16.2) - places boundaries and hatch patterns through the command bar. -Boundary (6.6.16) creates a polyline boundary inside closed areas through the command bar. GenerateBoundary (6.15.14) - Creates closed polylines from faces of 3D solids as well as the boundaries detected when the Enable Boundary Detection open of SELECTIONMODES is activated. HatchGenerateBoundary (6.16.5) - generates a boundary around a hatch or gradient fill. Properties (6.24.49) - changes the properties of hatch patterns. Region (6.26.18) - creates boundary-like regions. XEdges (6.34.3) - creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, surfaces, etc. and regions.

Figure 6.235

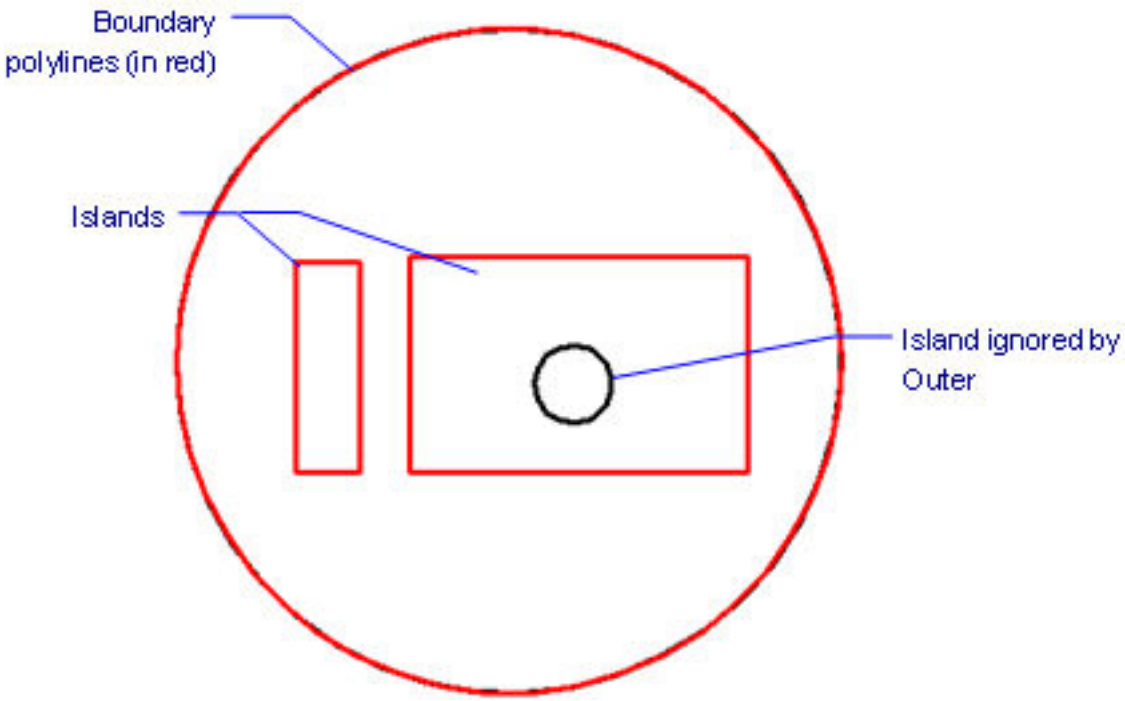


Figure 6.236



6.6.16 -Boundary

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a polyline boundary inside closed areas, through the command bar.

Accessing the Command

command bar: **-Boundary** alias: **-bo** : **-boundary Prompts you in the command bar:** Internal point or [Advanced/Undo]: **(Pick a point inside a closed area, or type A)** Internal point or [Advanced/Undo]: **(Pick a point inside another closed area, enter A, or press Enter to end the command)**

Command Options

Option	**Description**
Internal point	Specifiers the area by picking a point inside a closed area: **Internal point or [Advanced/Undo]** - **(enter an option)**: • **Internal point** - pick a point in the drawing that you hope is an enclosed area, or else enter x,y coordinates. The prompt repeats until you press Enter. • **Advanced** - switches to advanced mode; see below • **Undo** - un-selects the last selected entities When the program cannot find an enclosed area around your pick point, it reports: Selecting everything visible... Valid hatch boundary not found. Adjust the gap tolerance value if the boundary should be considered closed.
Advanced Options	**Description**
Advanced	Entering **a** for the Advanced option produces the following prompt: **[Boundary/Island/eXit] <eXit>** - **(enter an option)** • **Boundary** - jumps to the options for specifying the boundary; see below • **Island** - jumps to the options for specifying how islands should be treated; see below • **eXit** - returns to the ”Internal point or [Advanced/Undo]” prompt

Boundary	Specifies where the program should search for the entities making up the boundary: **Specify boundary set [New/Everything] <Everything>:** - **(enter an option)**: • **New** - starts a new boundary selection process; prompts you ”Select entities” • **Everything** - uses all entities in the current viewport for the boundary
Island	Determines whether islands are hatched over or not; prompts you: **Do you want Island Detection? [Yes/No/Outer only] <Yes>:**- **(enter an option)**: • **Yes** - places the boundary entity at the outermost area, as well as around all islands • **No** - islands are ignored; boundary is created of the outermost area only • **Outer only** - ignores innermost areas; boundary is created between outermost area and islands
eXit	(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ Returns to the previous prompt.

Procedures

- Drawing polylines (1.17.12)
- Creating boundary polylines (1.17.12)

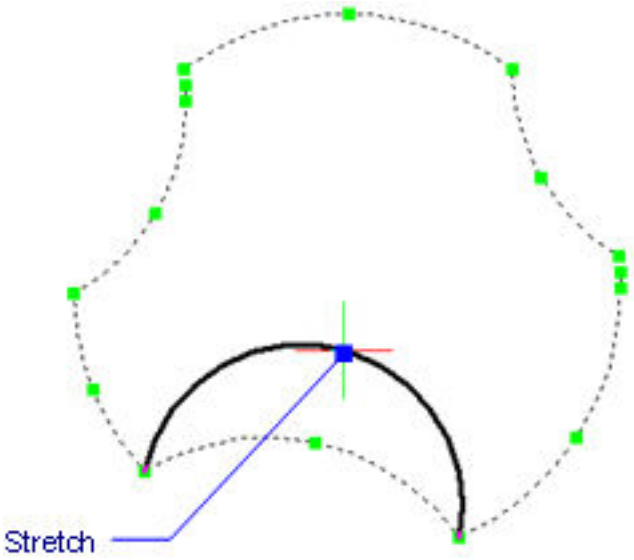
Grips Editing

Boundaries can be edited directly through grips:

1. Select the boundary. Notice that it has many grips.
2. Drag a grip to stretch the boundary.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image418.jpg})

Figure 6.237



Related Commands

BHatch (6.6.5) and Hatch (6.16.1) - place boundaries and hatch patterns through a dialog box. -BHatch (6.6.6) and -Hatch (6.16.2) - places boundaries and hatch patterns through the command bar. Boundary (6.6.15) places a polyline boundary inside closed areas through a dialog box. GenerateBoundary (6.15.14) - Creates closed polylines from faces of 3D solids as well as the boundaries detected when the Enable Boundary Detection open of SELECTIONMODES is activated. HatchGenerateBoundary (6.16.5) - generates a boundary around a hatch or gradient fill. Properties (6.24.49) - changes the properties of boundaries. Region (6.26.18) - creates boundary-like regions. Settings (6.27.23) - specifies the default parameters for boundaries. XEdges (6.34.3) - creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, surfaces, etc. and regions.

Figure 6.238

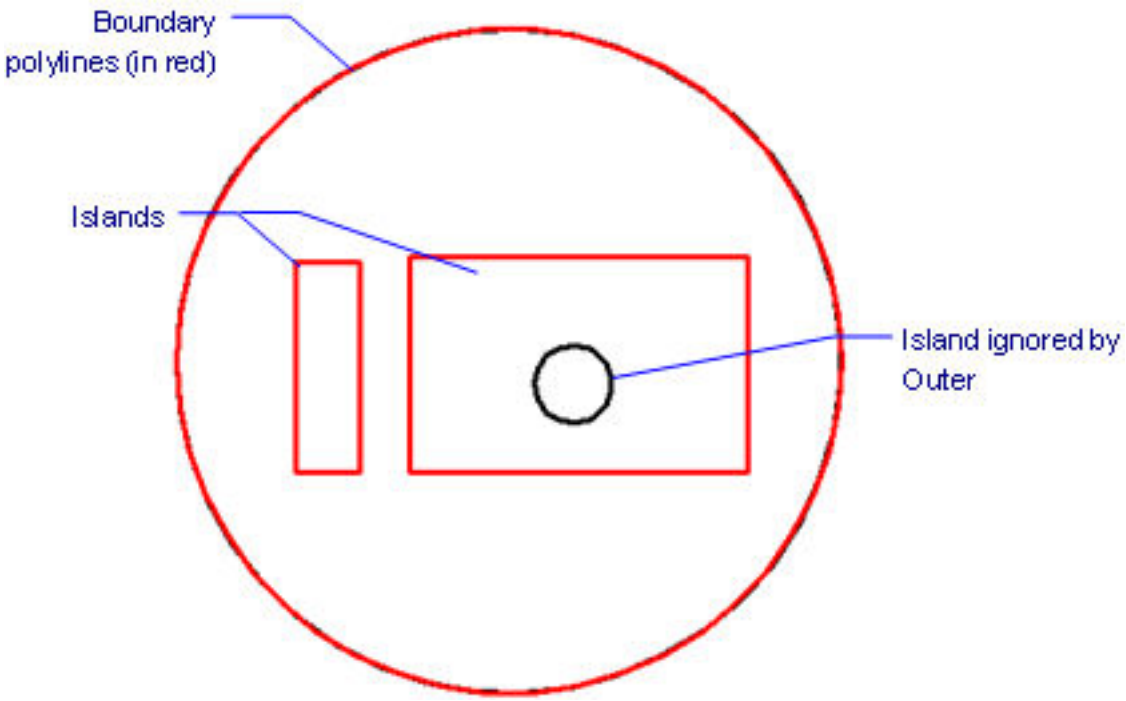


Figure 6.239



6.6.17 Box

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates 3D boxes:

- Classic: box is drawn from a 3D polyface mesh.
- Shape, Pro and Platinum: box is drawn as a 3D solid.

Accessing the Command

 command bar: **box** menu bar: **Model | 3D Solids | Box** toolbar: **3D Solids |**  ribbon:**Solid | Create | Primitive | Box** : **box Prompts you in the command bar;** in the Classic version, the prompts follow those of the **Ai_Box (6.5.3)** command: Set corner of box or [Center] <0.00,0.00,0.00>: **(Pick a point, or enter an option)** Set opposite corner or [Cube/Length)]: **(Pick another point, or enter an option)** Height of box [2Point]: **(Pick two points, or enter a height)** (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image60.jpg}) If Dynamic Dimensions (1.15.9) are active, you can enter the dimensions in the dynamic dimensions entry fields, as illustrated below. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/BoxDynDir

Figure 6.240

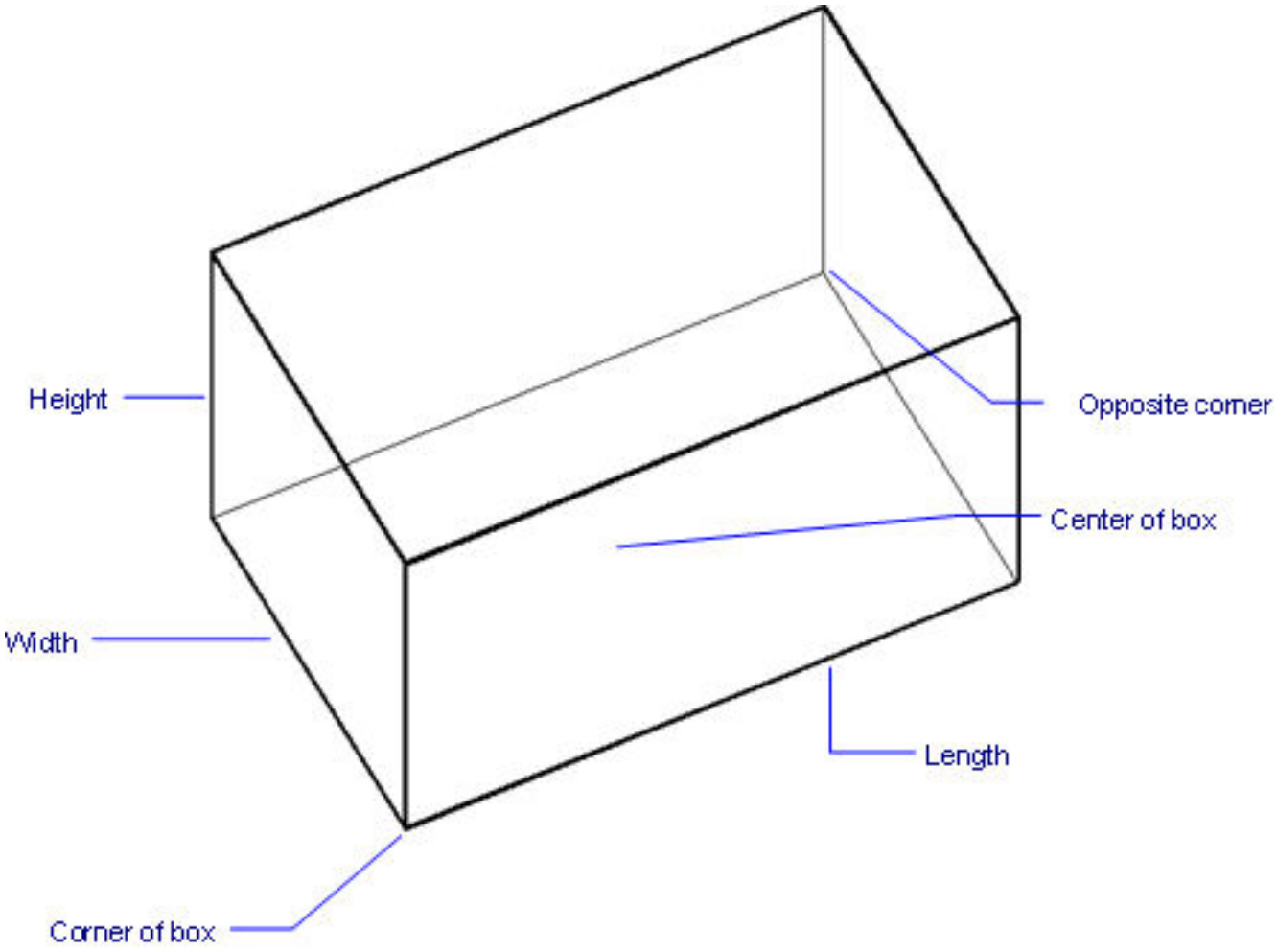
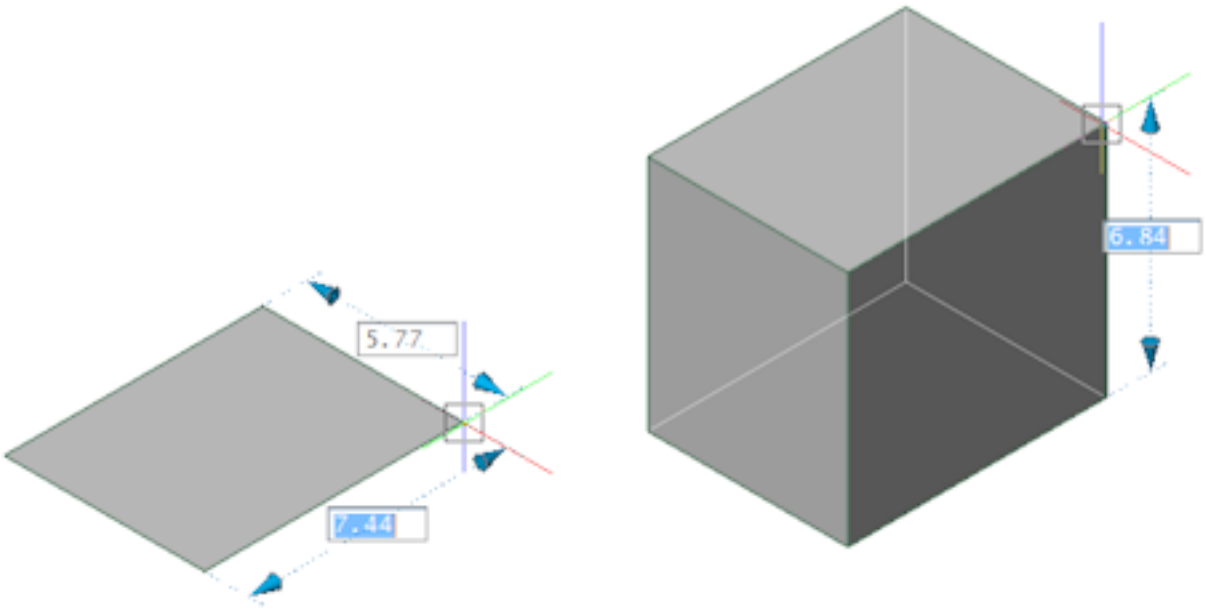


Figure 6.241



Command Options

Option	Description
Corner of box	Specifies one corner of the box's base.Enter x,y,z coordinates or pick a point in the drawing. Press Enter to reuse the previously-entered coordinates.
Center	Specifies the center of thebox's base; prompts you: **Center <1,2,3>** - **(Enter x,y,z coordinates, pick a point in the drawing, ore press Enter to accept the previously-used coordinates.)**
Cube	Specifies a box with three sides of equal length; prompts you: **Length of sides of cube <6>** - **(enter a length, or pick two points, or press Enter to accept the previously-used length.)** The command ends at this point, drawing the cube box.
Length of side	Specifies the length of each side: • **Length of side of box <3>** - **(specify the length in the x direction, pick a point, or press Enter to accept the previously-used length)** • **Width of box <2>** - **(specify the length in the y direction, or pick a point, or press Enter to accept the previously-used width)** • **Height of box <1>** - **(specify the length in the z direction, or pick a point, or press Enter to accept the previously-used height)** You may enter negative values to draw the box in the downward directions.
Set opposite corner	Specifies the opposite corner of the box's base; enter x,y,z coordinates or pick a point in the drawing
Height of box	Specifies the height of the box; enter an option: • **Height of box** - **(specify the height, or pick a point to indicate the height)** • **2Point** - **(pick two points in the drawing)**

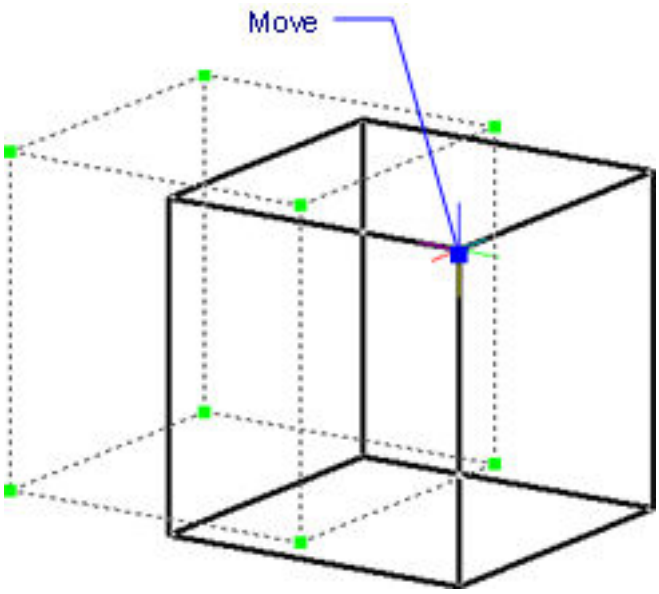
Grips Editing

Boxes can be edited directly through grips:

1. Select the box. Notice that it has many grips.
2. Drag any grip to move the box. (All eight grips perform the same function: move.)

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image429.jpg})

Figure 6.242



Related Commands

Cone (6.9.63) - creates 3D cones as solid models. Cylinder (6.9.60) - creates 3D cylinders as solid models. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Pyramid (6.24.58) - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polyface mesh (Classic) Revolve (6.26.30) - revolves 2D entities into 3D solid models. Section (6.27.11) - creates 2D regions from 3D models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Sphere (6.27.64) - creates 3D balls as solid models. Subtract (6.27.56) - removes one solid model from a second. Torus (6.29.22) - creates 3D donuts as solid models. Union (6.30.6) - joins two or more solid models into a single entity. Wedge (6.33.7) - creates 3D wedges as solid models.

Figure 6.243



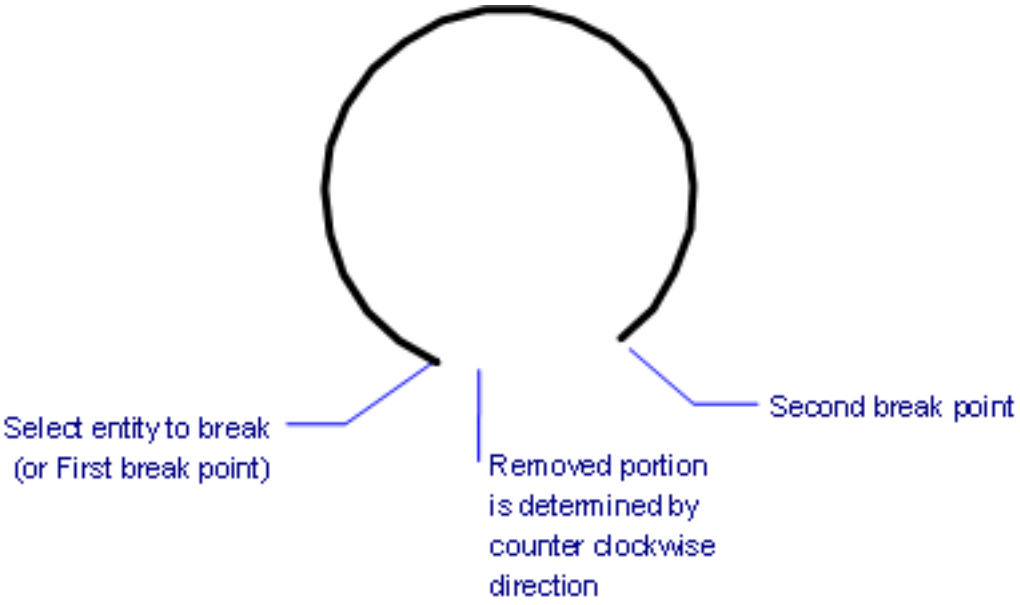
6.6.18 Break

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Removes a portion of an entity.

Accessing the Command

 command bar: **break** alias: **br** menu bar: **Modify | Break** toolbar: **Modify |**  ribbon:**Change | Modify | Break** : **break Prompts you in the command bar:** Select entity to break: **(Pick an entity)** Select second breakpoint or [First break point/@]: **(Pick a point on the entity or enter an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU

Figure 6.244



Command Options

Option	Description
Select entity to break	Selects the entity to break; only one entity may be selected.The pick point is also the first break point, unless you use the **First break point** option.
First break point	Specifies the start of the portion of the entity to remove.
Second break point	Specifies the end of the broken out section.On arcs and circles, the break occurs in the counter-clockwise direction from first to second break points.
@	Specifies that the first and second break points are at the same location on the entity.The selected entity is broken into two connected parts.

Procedures

- To break an entity (1.23.26)

Related Commands

DimBreak (6.11.7) - breaks dimension lines, extension lines and leaders at locations where they cross other entities. Join (6.18.1) - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.. PEdit (6.24.20) - joins lines and arcs into a single polyline. Trim (6.29.26) - cuts entities where they cross other entities.

Figure 6.245



6.6.19 Browser

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Opens the default Web browser.

Accessing the Command

command bar: **browser** alias: **url** : **browser Prompts you in the command bar:** Enter URL <http://bricsys.com>: **(Enter address of a Web page or press Enter to open the Bricsys website)**

Command Options

Option	Description
Enter URL	Specifies a location on the Internet, such as www.bricsys.com.After you press Enter, your Web browser opens to that page. (URL is short for ”universal resource locator.”)

Related Commands

OnWeb (6.23.6) - opens the Bricsys home page in your computer’s default Web browser. Mail (6.21.1) - attaches the current drawing to a message in the default email client.

6.7 BIM (Building Information Modeling)

6.7.1 BimAddEccentricity

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Allows to control the relative position of the axis of a linear solid.

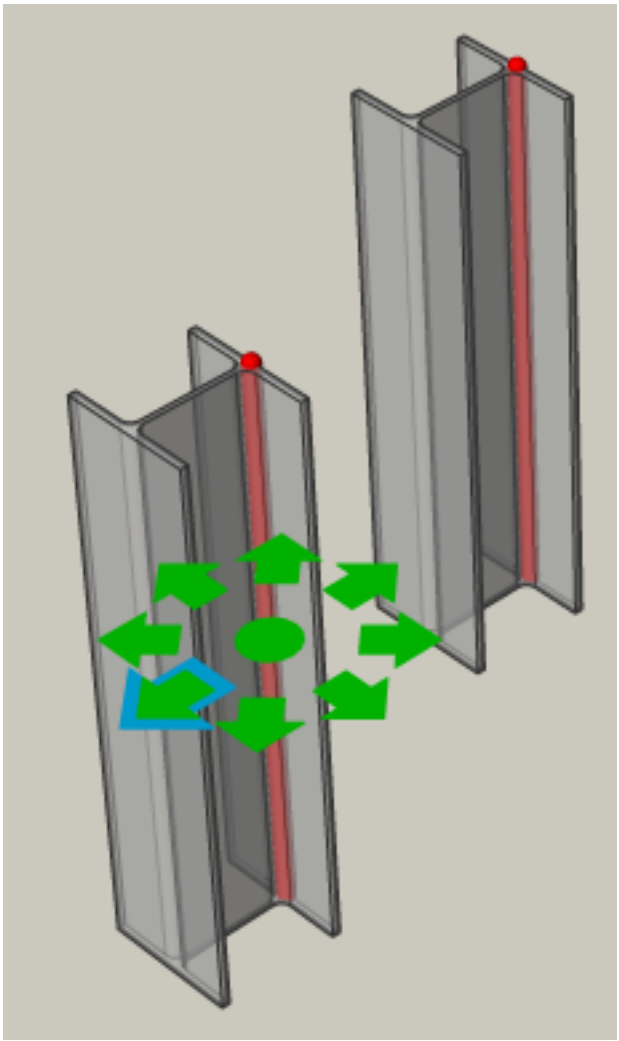
Figure 6.246



Accessing the Command


command bar: **bimaddeccentricity** Quad: **Model** :**BimAddEccentricity Prompts you in the command bar:** Select several entities/subentities: **(Select a linear solid.)** Entities in set: 1 Select several entities/subentities: **(Select more linear solids or press Enter.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/E
Choose base location [Accept] <Accept>: **(Click a new location; type a location or press Enter.)**

Figure 6.247



Command Options

Option	Description
Click the widget	Click one of the nine locations, then right click to confirm.
Type a location	The options are: • **TR**: Top Right • **TM**: Top Middle • **TL**: Top Left • **MR**: Middle Right • **C**: Center (resets the axes of the selection set to the default location) • **ML**: Middle Left • **BR**: Bottom Right • **BM**: Bottom Middel • **BL**: Bottom right

Procedures

- Add eccentricity to linear solids (2.4.2)
- Altering the axis of proiles
- Making a new steel connection

Related Commands

BimApplyProfile (6.7.2) - applies a profile to linear entities and linear solids. BimLinearSolid (6.7.15) - creates linear solids. BimProfiles (6.7.17) - opens the profiles library in the current project and the referenced library. BimRecalculateAxis (6.7.44) - recalculates the axis of linear building elements and repositions it back on the center line of a linear element.

Figure 6.248



6.7.2 BimApplyProfile

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Applies a profile to linear entities and linear solids.

Accessing the Command

command bar: **bimapplyprofile** menu bar: **BIM | Apply Profile** toolbar: **BIM |** : **BimApplyProfile Prompts you in the command bar:** Select path(s): **(Select a linear entity* or linear solid.)** Entities in set: 1 Select path(s): **(Select more linear entities or linear solids or press Enter.)** Select profile [in Dialog] <in Dialog>: **(Select a profile entity in the drawing or press Enter to display Profile dialog box.)** Apply profile? [Ok/Quarter turn/Rotate/Delete clippings] <Ok>: **(Press Enter to apply the profile or choose an option.)** *Lines, polylines, arcs, circles, elliptical arcs and ellipses are accepted. Open and closed splines are accepted only if they have default tangents and are not self-intersecting.

Command Options

Option	Description
Select profile	Allows to select a profile in the model; prompts you: **Pick a profile** - **(Do one of the following)**: • Click a closed 2D entity or a region in the model. • Click an existing linear solid.
In dialog	Opens the **Profiles** dialog box. Select a profile, then click the OK button on the Profiles dialog box.
OK	Applies the profile to the selection.
Quarter Turn	Rotates the profile 90° counter clockwise.
Rotate	Allows to rotate the profile about a user defined angle; prompts you: **Rotation angle <90>** - **(type a value)**. Positive values rotate the profile counter clockwise.
Delete Clippings	Removes all clippings from the selected linear solids.

Related Commands

BimAddEccentricity (6.7.1) - allows to control the relative position of the axis of a linear solid. BimLinearSolid (6.7.15) - creates linear solids. BimProfiles (6.7.17) - opens the profiles library in the current project and the referenced library.

Figure 6.249



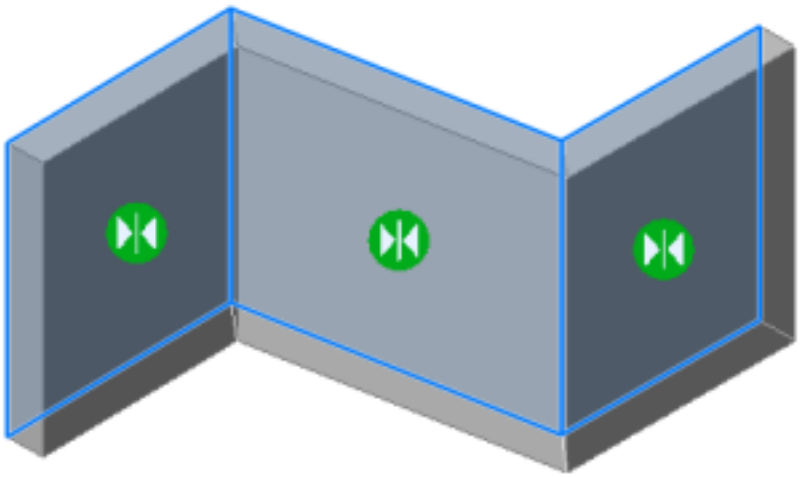
6.7.3 BimAttachComposition

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Attaches a BIM composition to solids.

Accessing the Command

 command bar: **bimattachcomposition** menu bar: **Bim | Attach Composition** Quad Tab:**Bim** Ribbon Tab | Panel: **Data | Attach** : **BimAttachComposition Prompts you in the command bar:** Select entities to attach composition:**(Select solids.)** Entities in set: 3 >Select entities to attach composition:**(Select more solids or press Enter.)** The reference face of the selected solids highlights. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/C Enter composition name or [Dialog/Entity] <Dialog>: **(Type a composition name or choose an option)** Flip or manually select reference faces, when done [Accept]: **(Press Enter to accept the indicated reference face or click the Flip Reference face** **or Select Reference face**  **icons.)** The composition has been assigned to 3 element(s)**.**

Figure 6.250



Command Options

Option	Description
Dialog	Opens the **Compositions** dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ **Option:** Click the type list button to select a composition type: • **All** : Lists all compositions • **Wall** : Lists compositions of the type **Wall** only. • **Slab** : Lists compositions of the type **Slab** only. • **Roof** : Lists compositions of the type **Roof** only. • **Generic** : List compositins of the type **Generic** only. The composition type is defined by the **Type** property of the composition. Although Compositions are grouped into **Wall** , **Slab** , **Roof** and **Generic** compositions, any type of composition can be attached to any dwg entity, including 2D entities. Do one of the following: • Double click a composition. • Select a composition, then click the Select button.
Entity	Applies the composition of an existing solid. Prompts you: **Select other entity to get composition - (select a solid)**. **Flip or manually select reference faces, when done [Accept] -(Press Enter to accept the indicated reference face or click the Flip Reference face or Select Reference face icons.)**
	Click the icon toflip the reference face to the opposite face of the solid. You can use window selection to select multiple icons: click, then drag the mouse and click again.
	Click the icon to select the reference face. Prompts you: **Select reference face - (select a face*) Successfully selected reference face Select opposing face - (select a face*) Successfully selected opposing face** * Hit the TAB key to select an obscured face.
Compositions panel (1.12.7) 	Drag a composition from the Compositions panel onto a selection of solids.

Procedures

- Attaching compositions to solids (2.6.2)
- To remove a composition from a solid (2.6.2)
- Creating a new composition (2.6.2)
- Displaying composition plies (2.6.2)
- Editing a composition (2.6.2)
- Modifying the thickness of a ply (2.6.2)
- Copying compositions (2.6.2)

- Deleting compositions (2.6.2)

Related Commands

Bimflip (6.7.10) - flips the starting face from which the layers of a composition are set out. BimProjectInfo (6.7.19) - opens the BIM Project Info dialog box. BimUpdateThickness (6.7.35) - re-applies the overall thickness of a composition to the solid. BimSetReferenceFace (6.7.43) - defines a reference and an opposing face to control the ply layout of a composition. BLCompositions (6.7.38) - opens the compositions library in the current project and the referenced library. BLMaterials (6.7.39) - opens the physical materials library in the current project and the referenced library. BimClassify (6.7.39) - Classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier).

Figure 6.251

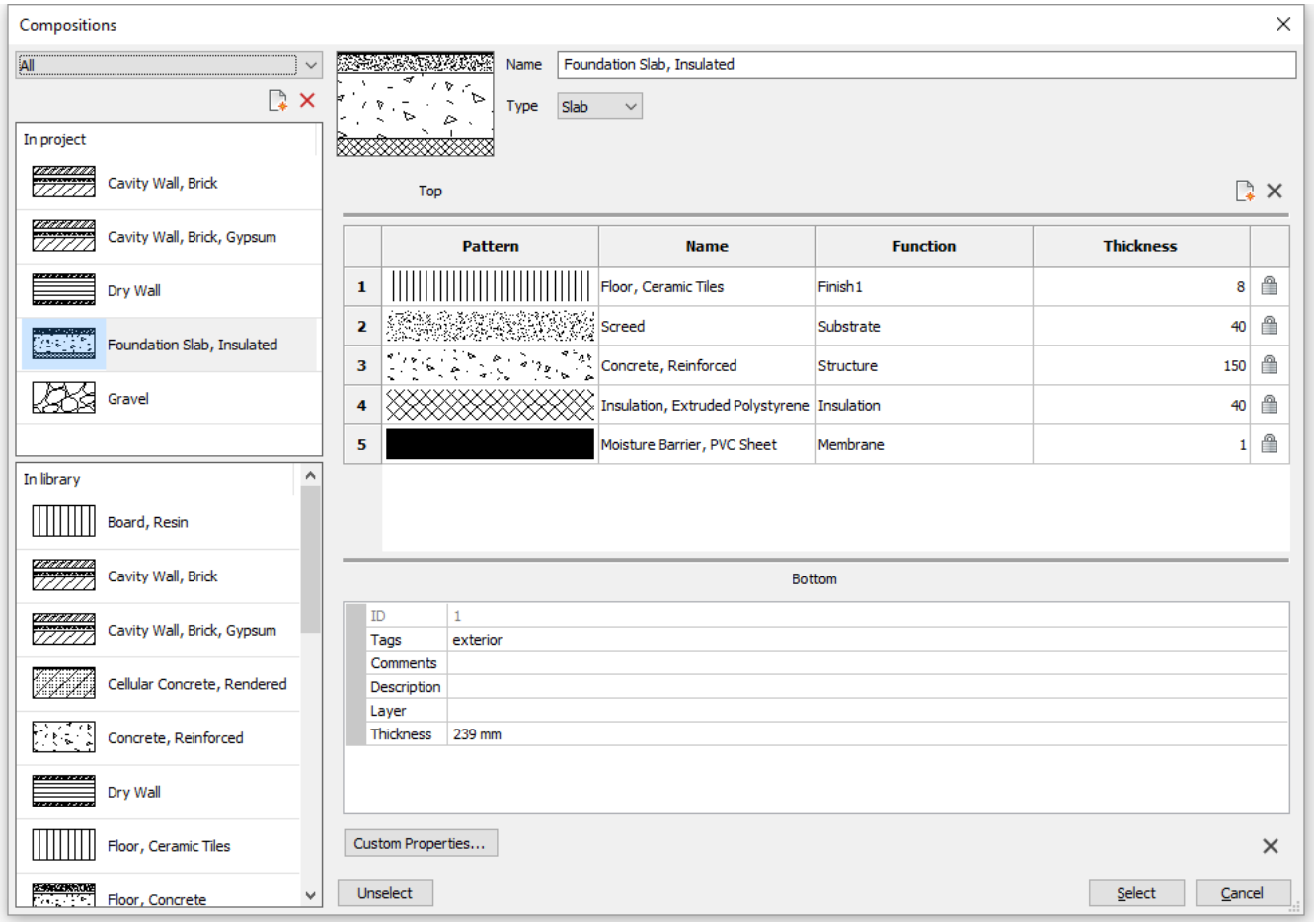

Compositions
All
In project
Cavity Wall, Brick
Cavity Wall, Brick, Gypsum
Dry Wall
Foundation Slab, Insulated
Gravel
In library
Board, Resin
Cavity Wall, Brick
Cavity Wall, Brick, Gypsum
Cellular Concrete, Rendered
Concrete, Reinforced
Dry Wall
Floor, Ceramic Tiles
Floor, Concrete
Name
Foundation Slab, Insulated
Type
Slab
Top
Pattern
Name
Function
Thickness
1
Floor, Ceramic Tiles
Finish1
8
2
Screed
Substrate
40
3
Concrete, Reinforced
Structure
150
4
Insulation, Extruded Polystyrene
Insulation
40
5
Moisture Barrier, PVC Sheet
Membrane
1
Bottom
ID
1
Tags
exterior
Comments
Description
Layer
Thickness
239 mm
Custom Properties...
Unselect
Select
Cancel

Figure 6.252



6.7.4 BimAttachSpatialLocation

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Allows to attach or remove a spatial location to a selection of entities.

Accessing the Command

 command bar: **bimattachspatiallocation** Quad: **BIM** |  : **BimAttachSpatialLocation Prompts you in the command bar*:** Locations: 1. Building-2 2. Building-2 - Floor 0 3. Building-2 - Floor 1 ... * It might be necessary to press the F2 function key to open the Prompt History (6.29.13) window. Enter location number or [Auto attach locations/Unattach current location] <Auto attach locations>: **(Enter a location number or choose an option.)** Select entities to update: **(Select the entities you want to attach a spatial location to.)** Entities in set: xx Select entities to update: **(Select more entities or press Enter.)** The spatial location has been assigned to xx element(s).

Command Options

Option	Description
Auto attach locations	Allows to automatically assign a spatial location; prompts you: **Select entities to update** - **(select the entities to be processed.) Entities in set: xx Select entities to update** - **(select more entities or press Enter.) Building 'Building': <Existed> The spatial location has been assigned to yy element(s). Skipped: zz element(s).** If necessary, new building(s) and stories are created in the Building and Story Manager (6.7.31) dialog box.
Unattach current locations	Allows to remove the current spatial location from a selection set; prompts you: **Select entities to update** - **(select the entities to be processed.) Entities in set: xx Select entities to update** - **(select more entities or press Enter.) The spatial location has been removed from xx element(s).**

Related Commands

BimClassify (6.7.5) - classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier). Bimify (6.7.13) - analyzes the model and runs automatic classification and spatial location assignment for the entire model. BimSpatialLocations (6.7.31) - opens the Buildings & Stories manager. This explorer allows to create and edit site, buildings and stories.

Figure 6.253



6.7.5 BimClassify

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier). Any dwg entity can be classified as a Building Element.

Accessing the Command


command bar: **bimclassify** menu bar: **BIM | Classify As Building Element** toolbar: **BIM** Quad: **BIM** : **BimClassify Prompts you in the command bar:** Classify entities as: [Wall/Column/Slab/Beam/wIndow/Door/buildingElement/Other/Auto/Unclassify] <Other>: **(Select an option.)** Select entities to classify <Drawing>: **(Select ore or more entities or press Enter to classify the drawing as a building element.)**

Command Options

Option	Icon	Description
Wall		Classifies the selection as wall elements. This option is applicable to any 3D Solid. Attaching a Wall composition (6.7.3) to a solid automatically classifies the solid as a wall.
Column		Classifies the selection as column elements. This option is applicable to any 3D Solid.
Slab		Classifies the selection as slab elements. This option is applicable to any 3D Solid. Attaching a Slab or Roof composition (6.7.3) to a solid automatically classifies the solid as a slab.
Beam		Classifies the selection as beam elements. This option is applicable to any 3D Solid.
Window		Classifies the selection as windows. This option is applicable to any block reference
Door		Classifies the selection as doors. This option is applicable to any block reference.

Building Element		Classifies the selection as unspecified building elements.
Other		Allows to classify the selection as building elements or convert the selection to a block and classify the block reference as a building element. Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 Do the following: • (option) Type a search string in the **Search** field to filter the available element types. • Select a building element type. • (option) Check the **Convert to block ...** option and type a name in the **Block name** field. • Click the **OK** button.
Auto		Automatically classifies the selected entity or entities. Section (6.27.12) entities become BimSection (6.7.27) entities.
Unclassify		Removes all BIM data from the selected entities.
Drawing		Classifies the entire drawing as a building element, e.g. a window, a door or a curtain wall. When the entire drawing is classified a number of appropriate properties can be defined in the Properties Bar (6.24.49). The following properties are available for windows: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 Glazing.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 Shading.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 Common.png})

Definitions

Type	Description
	Building Core Elements
Wall	Vertical construction that subdivides spaces. Example: separation wall, bearing wall, interior wall

Beam	Structural member that carries loads between or beyond points of support, usually narrow in relation to its length and (nearly) horizontal. Example: I-beam, L-beam, T-beam, double T-beam
Column	Structural member of slender form, usually vertical, that transmits to its base the forces, primarily in compression, that are applied to it. Example: Doric column, Ionic column, Corinthian column
Slab	Component of the construction that normally encloses a space vertically. The slab may provide the lower support (floor) or upper construction (roof slab) in any space in a building Example: floor slab, roof slab
Stair	Construction comprising a succession of horizontal stages (steps or landings) that make it possible to pass on foot to other levels. Example: stairs, escalator, spiral staircases interrupted with landings
Stair Flight	Assembly of building components in a single ”run” of stair steps (not interrupted by a landing). The stair steps and any stringers are included in this object. Example: stairs, escalator, spiral staircases NOT interrupted with landings
Ramp	Inclined way or floor joining two surfaces at different levels. Example: Slope/Ramp interrupted with landings
Ramp Flight	Inclined slab segment, normally providing a human circulation link between two landings, floors or slabs at different elevations. Example: Slope/Ramp NOT interrupted with landings
Railing	The railing is a frame assembly adjacent to human circulation spaces and at some space boundaries where it is used instead of walls or to complement walls. It is designed to aid humans, either as an optional physical support, or to prevent injury by falling. Example: handrail, balustrade, grab bar, guard rail
Curtain Wall	Exterior wall of a building which is an assembly of components, hung from the edge of the floor/roof structure rather than bearing on a floor.
Roof	Construction enclosing the building from above Example: flat roof, gable roof, gambrel roof, hip roof
Building Element	The building element comprises all elements that are primarily part of the construction of a building. Example: structural and space separating system like walls, beams, doors
Site	A defined area of land, possibly covered with water, on which the project construction is to be completed.
BIM XReference	The identification of information that is not explicitly represented in the current model or in the project database. Such information may be contained in classifications, documents or libraries within the external source.
Building Architecture Elements	
Window	Construction for closing a (near) vertical opening in a wall or pitched roof that will admit light and may admit fresh air. The window is a building element that is predominately used to provide natural light and fresh air. A window consists of a lining and one or several panels. Example: skylights, light domes, swinging, pivoting, sliding, or revolving panels and fixed panels
Door	Construction for closing an opening. The door is a building element that is predominately used to provide controlled access for people and goods. A door consists of a lining and one or several panels. Example: hinged, pivoted, sliding, revolving, folding doors

Covering	A covering is an element which covers some part of another element and is fully dependent on that other element. Example: wall claddings, floorings, suspended ceilings, finish trim, base molding
Furnishing Element	Generalization of all furniture related objects. Example: closet, table, chair, bed
Building Service Elements	
Distribution Control Element	The occurrence elements of a building automation control system that are used to impart control over elements of a distribution system. It senses elements and measures changes in the controlled variable such as temperature, humidity, pressure, or flow. Example: thermostat
Distribution Port	The occurrence of a specialized port for use within the context of distribution elements. Example: A gas-powered hot water heater may have three ports: GAS, DOMESTICCOLDWATER, and DOMESTICHOTWATER. The heater is a member of two systems (GAS and DOMESTICCOLDWATER), and hosts one system (DOMESTICHOTWATER) at the corresponding port.
Distribution Flow Element	The occurrence elements of a distribution system that facilitate the distribution of energy or matter, such as air, water or power. Example: ducts, pipes, wires, fittings
Distribution Chamber Element	A place at which distribution systems and their constituent elements may be inspected or through which they may travel. Example: sump, trench
Energy Conversion Device	The occurrence of a device used to perform energy conversion or heat transfer and typically participates in a flow distribution system. Example: boiler, chiller, or a cooling coil
Flow Fitting	The occurrence of a junction or transition in a flow distribution system. Example: elbow, tee, a junction box in an electrical distribution system
Flow Controller	The occurrence of elements of a distribution system that are used to regulate flow through a distribution system. Example: damper, valve, switch, relay
Flow Moving Device	The occurrence of an apparatus used to distribute, circulate or perform conveyance of fluids, including liquids and gases, and typically participates in a flow distribution system . Example: pump, fan
Flow Segment	A flow segment is a section of a distribution system that typically has only two ports. Example: section of aduct, pipe, conduit, etc.
Flow Storage Device	Device used for the temporary storage of a fluid such as a liquid or a gas, the voltage potential induced by the induced electron flow. Example: tank, battery
Flow Terminal	The occurrence of a permanently attached element that acts as a terminus or beginning of a distribution system. A terminal is typically a point at which a system interfaces with an external environment. Example: air outlet, drain, water closet, sink
Flow Treatment Device	Device used to change the physical properties of the medium, flow treatment types (or the instantiable subtypes) may be exchanged without being already assigned to occurrences. Example: air, oil or water filter, duct silencer
Building Structure Elements	
Member	Structural member designed to carry loads between or beyond points of support. It is not required to be load bearing. The orientation of the member (being horizontal, vertical or sloped) is not relevant to its definition. Example: diagonal element of a truss construction
Footing	A part of the foundation of a structure that spreads and transmits the load directly to the soil. Example: wall footing, stepped footing, sloped footing, raft footing

Pile	A slender timber, concrete, or steel structural element, driven, jetted, or otherwise embedded on end in the ground for the purpose of supporting a load. A pile is also characterized as deep foundation, where the loads are transfered to deeper subsurface layers. Example: pile foundation
Reinforcing Bar	A steel bar, usually with manufactured deformations in the surface, used in concrete and masonry construction to provide additional strength. Example: reinforced concrete
Reinforcing Mesh	A series of longitudinal and transverse wires or bars of various gauges, arranged at right angles to each other and welded at all points of intersection. Example: reinforced concrete floor slab
Tendon	A steel element used to impart prestress to concrete when the element is tensioned. Example: wire, cable, bar, rod, strand
Tendon Anchor	A tendon anchor is the end connection for tendons in prestressed or posttensioned concrete.
Plate	A Plate is a planar and often flat part with constant thickness. A plate can be a structural part carrying loads between or beyond points of support, however it is not required to be load bearing. The location of the plate (horizontal, vertical or sloped) is not relevant to its definition.
Discrete Accessory	A discrete accessory is a representation of different kinds of accessories included in or added to elements. Example: visible steel corbel (corbel system made from steel components protruding from the master element), corner fixing plate (fixing plate attached to the corner of an element), steel plate (steel plate used as an accessory in a joint)
Fastener	Representations of fixing parts which are used as fasteners to connect or join elements with other elements. Example: jointing mortar, glue, weld
Mechanical Fastener	Fasteners connecting building elements mechanically. Example: bolt, screw, nail, nut, washer, rivet

Procedures

- Classifying entities (2.6.1)
- Creating a simple window (2.4.5)

Related Commands

BimAttachComposition (6.7.3) - attaches a BIM composition to solids. BimProperties (6.7.41) - allows to create, edit and delete property definitions in a dialog box, and to organize properties in property sets. BimSpatialLocations (6.7.31) - opens the Buildings & Stories manager. This explorer allows to create and edit site, buildings and stories. Bimify (6.7.13) - analyzes the model and runs automatic classification and spatial location assignment for the entire model. Export (6.13.15) - exports drawings in a variety of vector and raster formats, including *.ifc

Figure 6.254

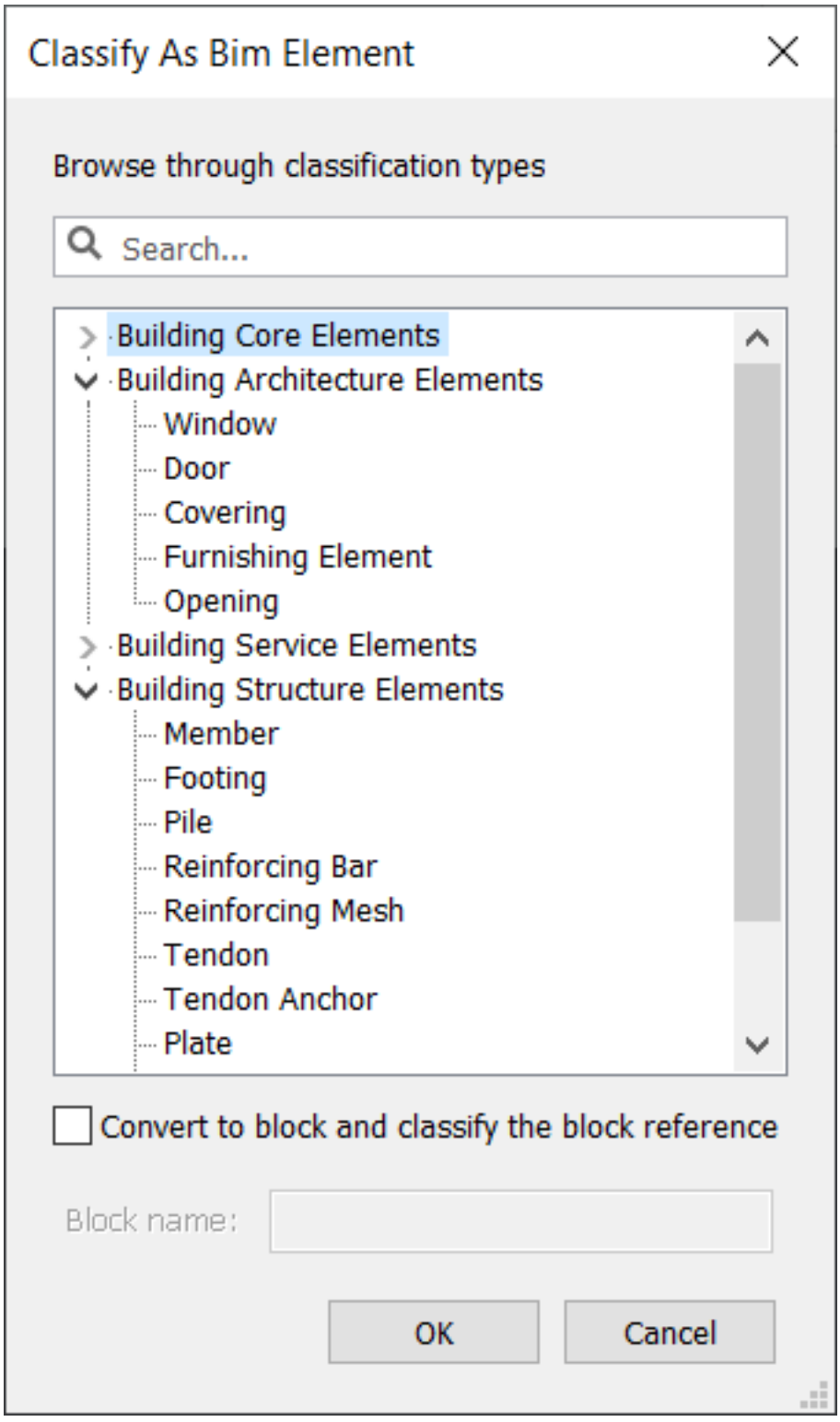

Classify As Bim Element
Browse through classification types
Search...
Building Core Elements
Building Architecture Elements
Window
Door
Covering
Furnishing Element
Opening
Building Service Elements
Building Structure Elements
Member
Footing
Pile
Reinforcing Bar
Reinforcing Mesh
Tendon
Tendon Anchor
Plate
Convert to block and classify the block reference
Block name:
OK
Cancel

Figure 6.255

BIM	
Type	Window
Name	W_3x1_L
Building	
Story	
Composition	none
Display composition	Off
GUID	{D7FB5026-4C
Construction type	ALUMINIUM
Operation type	SINGLE_PANEL
Window insert	
Overall height	1200 mm
Overall width	1500 mm
Sill height	800 mm
Head height	0 mm
Assembly code	
Assembly subtype	

Figure 6.256

IFC Glazing	
Glass layers	2
Glass color	
Tempered	Off
Laminated	Off
Coated	Off
Wired	Off
Fill gas	
Translucency	1
Reflectivity	0
Beam radiation transmittance	0
Solar heat gain transmittance	0
Thermal transmittance summer	0
Thermal transmittance winter	0

Figure 6.257

IFC Shading	
External shading coefficient	0
Internal shading coefficient	0
Inset shading coefficient	0
Manufacturer	
Manufacturer	
Manufacturer URL	
Manufacturer product name	
Manufacturer product No	
Cost	0
Construction Specifiers Institute	

Figure 6.258

IFC Common	
Reference	
Acoustic rating	
Fire rating	
Thermal transmittance	0
External	On
Smoke stop	Off
Security rating	
Infiltration	0
Glazing area fraction	0

Figure 6.259



6.7.6 BimCopy

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Creates copies from the planar faces of 3D solids, and the sides and ends of linear solids.

Accessing the Command

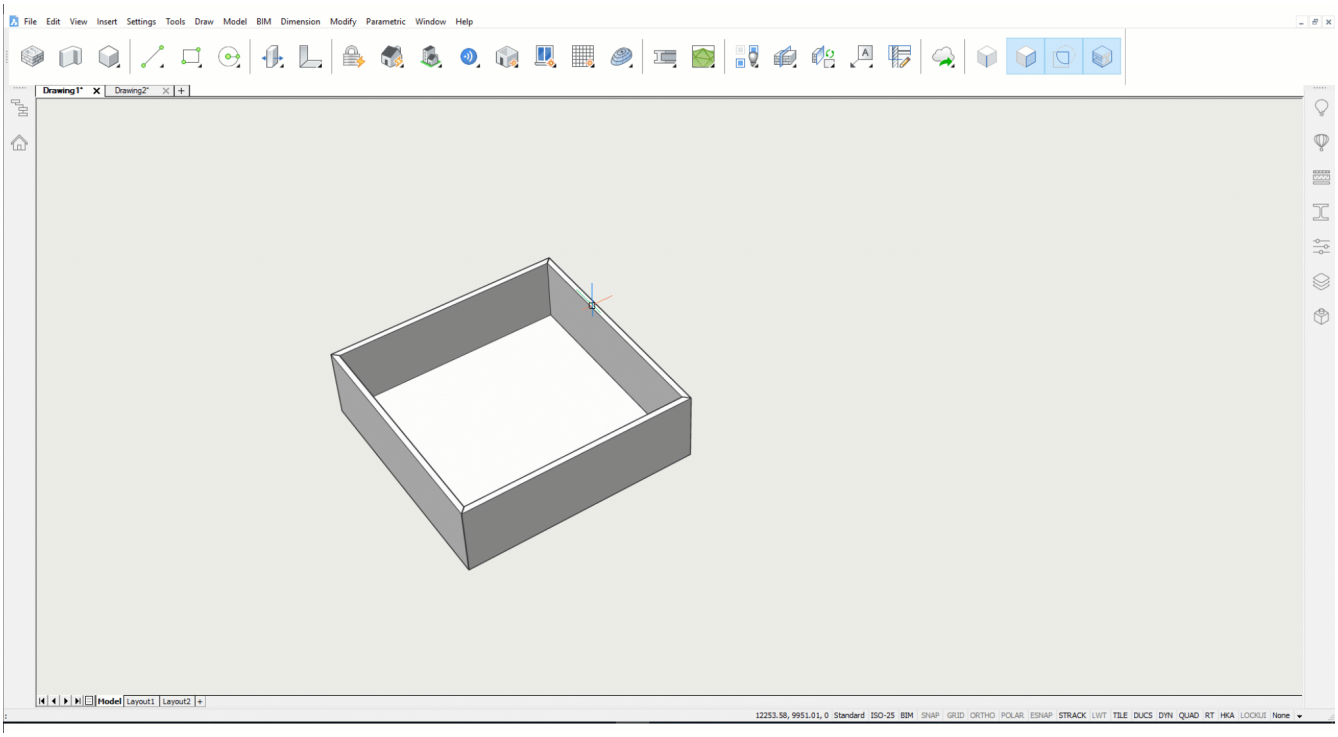

Keyboard:**Bimcopy** Menu:**BIM | BIM copy** Toolbar:**BIM Toolbar| BIM copy** Quad Tab:**Model tab | BIM copy** Ribbon Tab | Panel: **Model | Modify 3D| BIM copy : BimCopy** Move the solid, specify a distance in the dynamic dimension field, and right-click to create a single copy. **Prompts you in the command bar:** Copy mode: single copy (in the Prompt History window) **Prompts you in the command bar:** Specify distance to create copy or switch to [Copy/Repeat/Number/Accept] <Accept>: **(Enter the distance value to make a copy of the solid at the specified distance or use one of the available options in the command bar)**

Command Options

Option	Description
Copy	Creates copies of the planar faces of 3D solids, and the sides and ends of linear solids.
Repeat	Repeats the copy using the same displacement. Move the cursorin the displacement direction. The further you move away from the source solid, the more copies are created. Click when the desired number of copies displays.
Number	Creates a number of copies you want to have in your model; prompts you: **Specify number of repeated copies:(specify the number of entities you want to create in the dynamic dimension field and press Enter to accept it.)**
Accept	Accepts the current distance you entered in the dynamic dimension field. This is the default option. Either press Enter to use this option or type A in the command line.

The following animated gif shows how to use bimcopy for 3D solids and linear solids. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BIMCopy3DSolidsLinearSolids.png})

Figure 6.260



Related Commands

Copy (6.9.50) - makes copies of entities.

Figure 6.261



6.7.7 BimCurtainWall

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Creates a curtain wall from the face of a 3D Solid as a block.

Accessing the Command

 Keyboard: **bimcurtainwall** Menu: **BIM | Curtain Wall** Toolbar: **BIM and BIM Curtain Wall** Quad Tab: **Model** Ribbon Tab | Panel: **Model | Curtain Wall :BimCurtainwall Prompts you:** Select a face: **(Select the face of a solid or surface)** Create grid or [Length U panels/Length V panels/Number U panels/Number V panels]: **(Press Enter or choose an option.)** ===ERROR WITH DOWNLOADED IMAGE=== Change parameters or accept [Accept/Width/Depth/Glass thickness/Smooth connections] <Accept>: **(Press Enter to create the curtain wall using the current values or choose an option.)** ===ERROR WITH DOWNLOADED IMAGE=== Delete defining entities? [Yes/No] <Yes>:**(Press Enter or choose an option)** This prompt only displays when the value of the DELOBJ (1.13.11) system variable is**-1** or **-2**.

Command Options

Option	Description
Grid Options	
Length U panels	Sets the length of the panels in U direction (height). Prompts you: **U grid length <550>: (Type a value or press Enter.)**
Length V panels	Sets the length of the panels in V direction (width). Prompts you: **V grid length <3000>: (Type a value or press Enter.)**
Number U panels	Number of panels in U direction (rows). Prompts you: **U grid size <5>:(Type a number of press Enter.)**
Number V panels	Number of panels in V direction (columns). Prompts you: **V grid size <3>:(Type a number of press Enter.)**
Parameter options	
Width	Defines the width of the curtain wall frame, mullions and transoms. Prompts you: **Frame width <60.000000>:(Type a value or press Enter.)**
Depth	Defines the depthof the curtain wall frame, mullions and transoms. Prompts you: **Frame depth <60.000000>:(Type a value or press Enter.)**
Glass thickness	Defines the thickness of the glass panels. Prompts you: **Glass thickness <20.000000>:(Type a value or press Enter.)**
Smooth connections	Defines whether the connections between the frames, mulliions and transoms are smooth (left) or not (right) ===ERROR WITH DOWNLOADED IMAGE=== ===ERROR WITH DOWNLOADED IMAGE=== Prompts you: **Smooth connections [Yes/No] <No>:(Choose an option or press Enter.)**

Delete defining entities	Whether the selected source entity is retained or deleted depends on the value of theDELOBJ (1.13.11) system variable: • 0 : retained • 1 or 2 : deleted • -1 or -2 : You are prompted:**Delete defining entities? [Yes/No] <Yes>:**

Related Commands

BimWindowCreate (6.7.36) - creates a fully parametric window or opening based on a closed contour. Panelize (6.7.40) - creates a grid on the face of a 3D Solid as a block.

Figure 6.262



6.7.8 BimDrag

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Moves 3D solids perpendicular to a selected face by dragging that face. Optionally preserves connections to other solids.

Accessing the Command

 Command bar: **bimdrag** Menu bar: **BIM | Drag** Toolbar: **BIM** Quad: **BIM** Ribbon tab | panel: **Model | Modify 3D** :**BimDrag Prompts you in the command bar:** Select several entities/subentities: **(select a face of a solid)** Entities in set: 1 Select several entities/subentities: **(select more faces, then right click or press Enter to stop selecting.)** The selected face(s) move(s) dynamically. The manipulator (6.21.2) displays. When multiple faces are selected, the solids are moved perpendicular to the face selected last. ===ERROR WITH DOWNLOADED IMAGE=== Specify distance to drag solid or [Disable connectivity mode]: Optionally disable/enable connectivity mode, then do one of the following:

- (option)The current reference face highlights.Hit the TAB key to select a different reference face. All faces which are parallel to the selected one are recognized, the nearest first. Press and hold the SHIFT key when hitting the TAB key to cycle reference faces in reverse order. To select a reference face on a different solid: move the cursor over that face, then hit the TAB key. The selected face highlights.
- Click a point in the model.
- Type a value in the dynamic distance field.
- Move the cursor to adjust the value in the dynamic distance field, then click. Optionally zoom in/out to set the snap increment size of the ruler.

Command Options

Option	Description
Connectivity mode	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/BimDra The options are: •  : Enable connectivity •  : Disable connectivity Do one of the following: • Hit the Ctrl-key to toggle the connectivity mode. • Type **d** to disable connectivity • Type **e** to enable connectivity

Related Commands

dmPushPull (6.12.20) - adds or removes volume from a solid by moving a face. dmMove (6.12.17) - moves the selected solids, or faces or edges of a solid using a vector.

Figure 6.263



6.7.9 BimExport

BIM BIM BIM Obsolete. Use Export (6.13.15) instead and select file type *.ifc.

Figure 6.264



6.7.10 BimFlip

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Flips the starting face from which the layers of a composition are set out. Mirrors an insert (e.g. a window or door) left/right or flips it inside/outside.

Accessing the Command

command bar: **bimflip** menu bar: **BIM | Flip** toolbar: **BIM |** Quad: **BIM Classification |** (solids) Quad: **BIM |** (inserts) : **BimFlip Prompts you in the command bar:** Select BIM entity to flip: **(Select a wall, slab or an insert)** Entities in set: 1 Select BIM entity to flip: **(Select more walls, slabs or inserts, or press Enter.)** If the selection contains no inserts: Done on xx object(s). The result is visible in drawings generated by the BimSectionUpdate (6.7.29) command if a multi-layered composition is attached to the sectioned solid.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimFlipBefore.png}) Before	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimFlipAfter.png}) After

If the selection contains at least 1 insert: Flip insert(s) Left-right/In-out <Left-right>: **(Press Enter to flip left-right or type I and press Enter to flip inside-outside.)**

Command Options

Option	Description
Select BIM entity to flip	The **BimFlip** command accepts inserts, walls or slabs. Solids that have no BIM classification (6.7.5) yet, cannot be selected. The command only succeeds for solids that meet both of the following conditions: • have a BIM composition (6.7.3) attached. • are in a calculated BIM section (6.7.27). BricsCAD reports the number of solids that could not be flipped: '... failed on xx object(s)'
Left-right	Mirrors the insert about a vertical axis in the face of the solid.
In-out	Flips the insert to the opposite face of the solid.

Related Commands

BimAttachComposition (6.7.3) - attaches a BIM composition to solids. BimClassify (6.7.5) - classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier). BimUpdateThickness (6.7.35) - re-applies the overall thickness of a composition to the solid. BimSection (6.7.27) - creates a BIM section entity. BimSectionUpdate (6.7.29) - exports or updates the result of a BIM section in a drawing. BimSetReferenceFace (6.7.43) - defines a reference and an opposing face to control the ply layout of a composition.

Figure 6.265

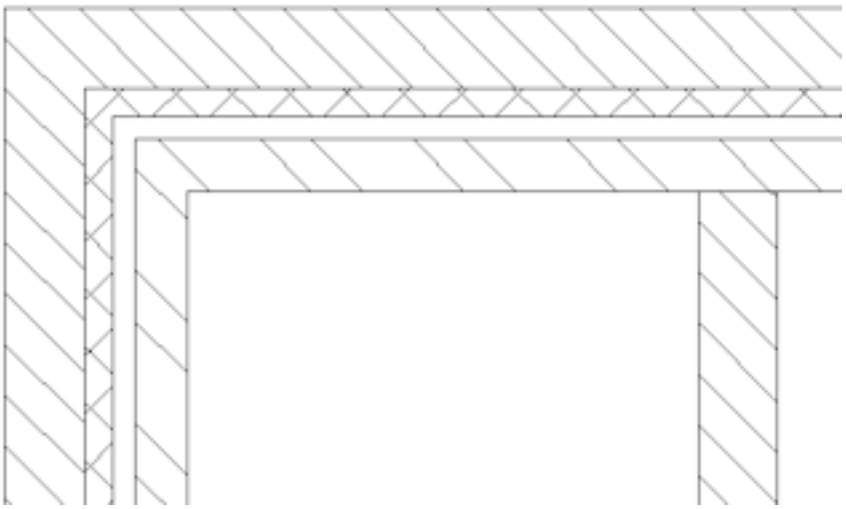


Figure 6.266

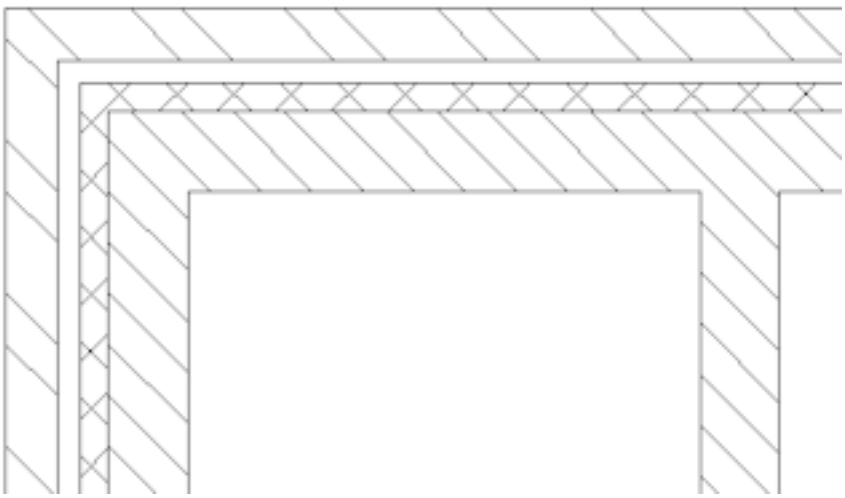


Figure 6.267



6.7.11 BimFlowConnect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Creates a connection between flow segments, such as pipes or HVAC ducts.

Accessing the Command



Keyboard:**bimflowconnect** Menu:**BIM | Bim Connect | Flow Connect** Toolbar:**BIM Connect** Quad Tab:**Model** Ribbon Tab | Panel: **Model | Modify 3D | Connect : BimFlowConnect Prompts you:** Select solids to connect: **(Select one or more flow segments*.)** Entities in set 1: Select solids to connect: **(Select more flow segments* or press Enter.)** * When more than 2 flow segments are selected, the axes of the selected segments must be coplaner. When multiple coplanar segments are selected, the connections are created. When 2 non coplaner solids are selected: Apply connection or switch to the next alternative [Switch] <Accept>:**(Press Enter to accept the connection or press the Ctrl-key for alternative connection(s))**

Command Options

Option	Description
Switch	Switches between possible connections. Type s + Enter or tap the Ctrl-key to cycle the various alternatives. Alternatives for circular flow segments: ===ERROR WITH DOWNLOADED IMAGE=== ===ERROR WITH DOWNLOADED IMAGE=== Alternatives for rectangular flow segments: ===ERROR WITH DOWNLOADED IMAGE=== ===ERROR WITH DOWNLOADED IMAGE=== ===ERROR WITH DOWNLOADED IMAGE===
Reducers	When the section of the selected flows is different, reducer elements are inserted. ===ERROR WITH DOWNLOADED IMAGE=== ===ERROR WITH DOWNLOADED IMAGE===

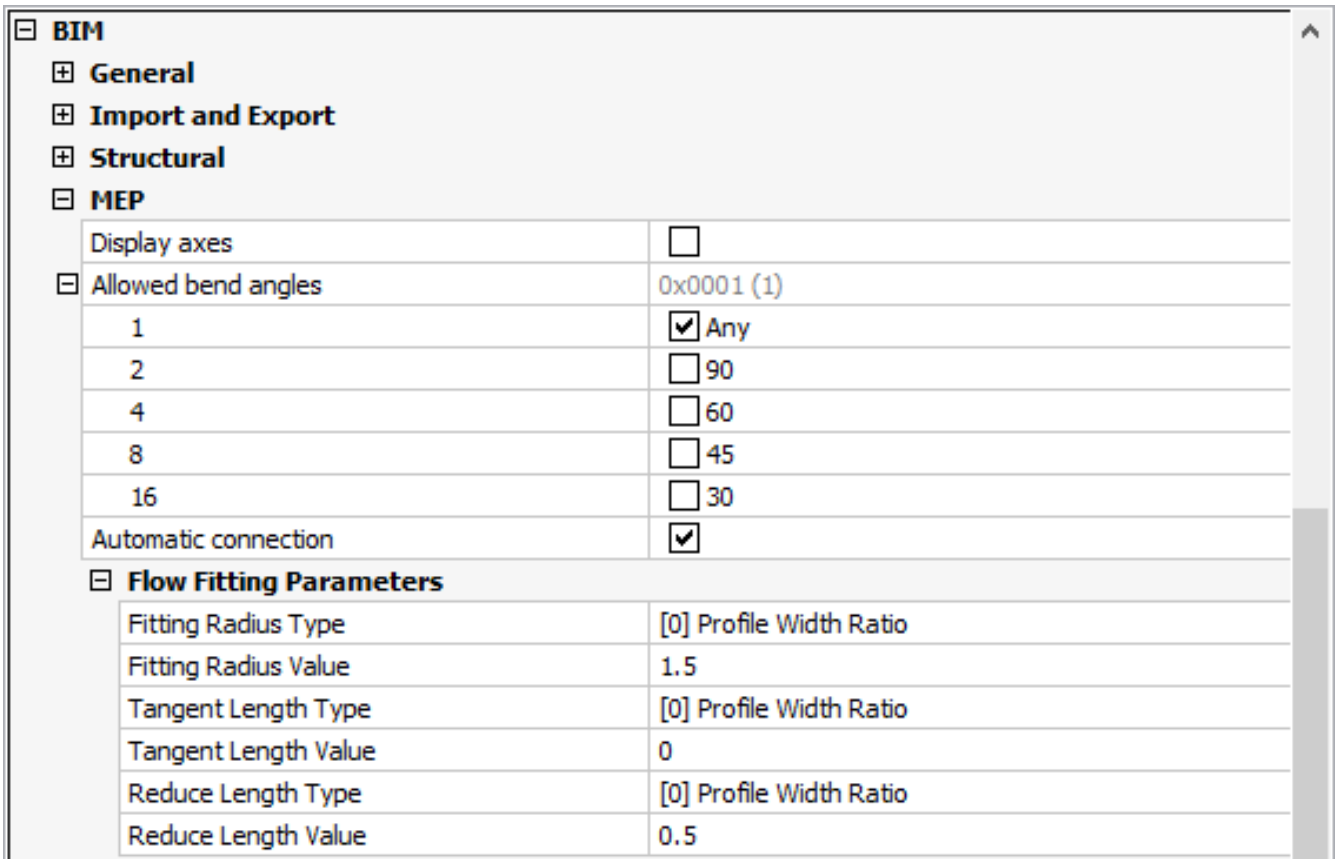
User Preferences

The creation of bends and reducers is controlled by user preferences (1.13.11). (\vref {fig:C:/Dev/Task-2448/images/staticbric

Display axes	Displays a red axis of the profile.
Allowed bend angles	Defines which angles are used for MEP bends. • **Any**: allows to connect any two ducts, • **90**, **60**, **45**, **30**: specifies which bend angles are allowed.
Automatic connection:	If set, APPLYPROFILE and LINEARSOLID commands automatically create bends when creating multiple flow segments in the same same workflow.

Flow Fitting parameters	These parameters control the creation of bends and reducers. • Fitting Radius: radius of the bend; more precisely the radius sweep path of the bend. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands • Tangent Length: length of the straight part at the end of a bend. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands • Reduce Length: Length of a reducer element which connects two different profiles. For each of the flow fitting parameter you can choose between a relative value (Profile Width Ratio) or an absolute value.

Figure 6.268



Related Commands

BimDrag (6.7.8) - moves 3D solids perpendicular to a selected face by dragging that face. Optionally preserves connections to other solids. BimStructuralConnect (6.7.30) - connects structural profile solids such as beams and columns. LConnect (6.20.19) - creates an L-connection between solids. TConnect (6.29.10) - connects solid faces to other solids.

Figure 6.269

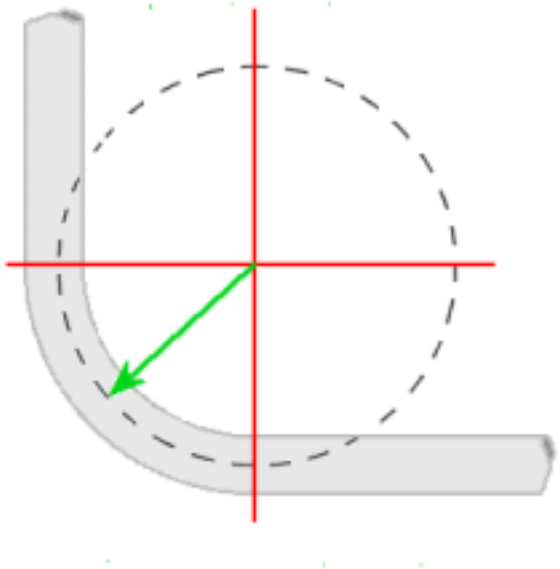


Figure 6.270

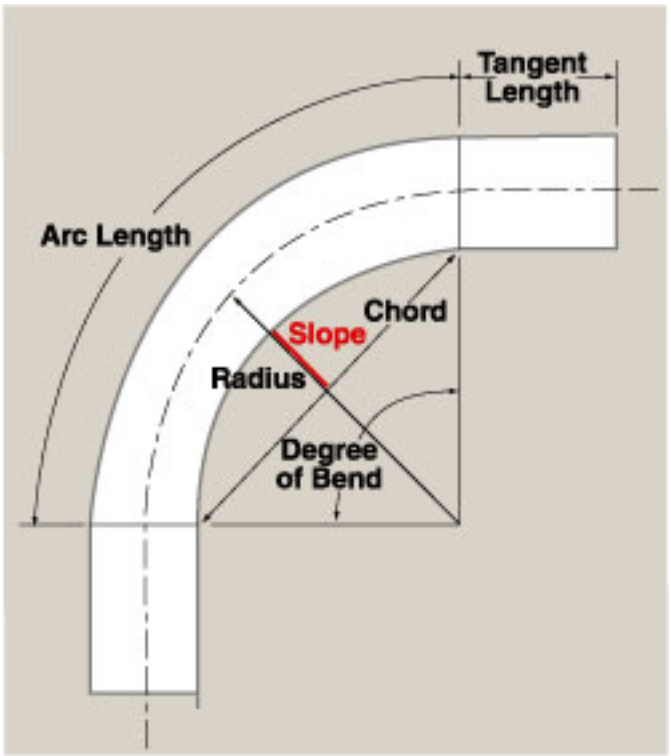


Figure 6.271

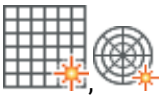


6.7.12 BimGrid

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Creates 2-dimensional rectangular and radial grids.

Tutorial: How to use Grids Click here (2.3.12) to watch

Accessing the Command

 , **Keyboard**: bimgrid **Menu**: BIM | Rectangular Grid **Toolbar**: BIM and BIM Grid **Quad Tab**: Model **RibbonTab | Panel**: Model | Grids **: BimGrid Prompts you:** Enter first point [offsetU/offsetV/offset Labels/Radial]: **(Specify the first point or choose an option.)** ===ERROR WITH DOWNLOADED IMAGE=== Enter second point [offsetU/offsetV/offset Labels]: **(Specify the second point or choose an option.)** ===ERROR WITH DOWNLOADED IMAGE===

Command Options

Option	Description
Offset U	Sets the line offset between U lines in V direction (interdistance between the U lines). Prompts you: **Enter offset between U grid axes:(Type a value or press Enter.)**
Offset V	Sets the line offset between V lines in U direction (interdistance between the V lines). Prompts you: **Enter offset between V grid axes:(Type a value or press Enter.)**
Offset Labels	Sets the label offset to grid axes. Prompts you: **Enter label offset:(Type a value or press Enter.)**
Radial ( )	Creates a radial grid. Prompts you: **Enter first point[offsetU/offsetV/offset Labels/Rectangular]:(Specify the center point of the radial grid or choose an option.)** ===ERROR WITH DOWNLOADED IMAGE=== **Enter second point [offsetU/offsetV/offset Labels]:(Specify the start lines of the radial grid or choose an option.)** ===ERROR WITH DOWNLOADED IMAGE=== **Enter third point [offsetU/offsetV/offset Labels]:(Specify the end lines of the radial grid or choose an option.)** ===ERROR WITH DOWNLOADED IMAGE===

Procedures

- Editing a rectangular grid (2.4.13)
- Renumbering the grid labels (2.4.13)

Related Commands

BimCurtainWall (6.7.7) -Creates a curtain wall from the face of a 3D Solid as a block. BimPanelize (6.7.40) -Creates a grid on the face of a 3D Solid as a block. Number (6.22.9) -Creates incremental number tags for BIM entities. BimWindowCreate (6.7.36) -Creates a fully parametric window or opening based on a closed contour.

Figure 6.272



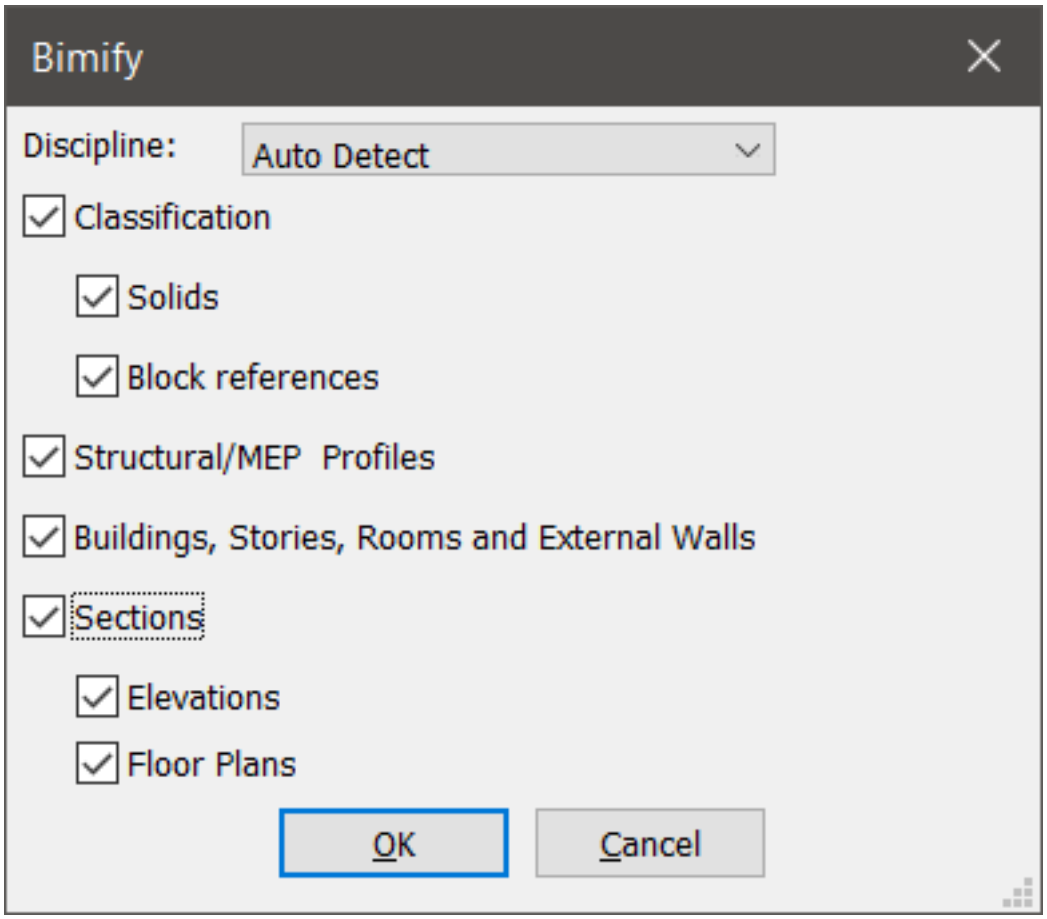
6.7.13 Bimify

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Analyzes the model and runs automatic classification and spatial location assignment for the entire model. Creates new buildings and/or stories and rooms if necessary. Optionally creates plan sections and elevations. Section entities (6.27.12) are converted to BimSection (6.7.27) entities.

Accessing the Command

 Menu: **BIM | Bimify** Quad: **BIM | Bimify** Toolbar: **BIM | BIMIFY** Keyboard: **bimify** : **Bimify** Displays a dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimifyDialog.png}) Check the options of your choice, then click the OK button. BricsCAD reports in the Prompt History (6.29.13) panel: BIM data assigned to 27 of 27 object(s), skipped 0 pre-assigned object(s) Time taken to classify 0.10 s Building 'Building-2': <Created> The spatial location (6.7.31) has been assigned to 27 element(s). Time taken for creation of hatches 0.02 s Time taken for creation of volumes 0.06 s Successfully created 3 rooms*. Time taken in total: 0.11 s Determined outer walls Time taken in total: 0.08 s Creating Sections ** * Room entities are created on the BIM_ROOMS layer. ** Section entities are created on the BIM_SECTIONS layer.

Figure 6.273



Command Options

Option	Description
Classification	See BimClassify (6.7.5). • Solids:Classifies 3D Solids as walls, slabs, ... • Block References: Classifies blocks as windows, doors, ...
Structural/MEP Profiles	Assignscolumns, beams, members or flow segments profiles that are available in BricsCAD profile library. If no match can be found for a profile element in the standard library, Bimify creates a new custom profile in the project's library.
Buildings, Stories, Rooms and External Walls	Assigns spatial locations (6.7.31) (buildings and floors) Creates rooms (6.7.26). Detects externalwalls and sets the property Wall Common / Is External = ON.
Sections	Creates BIM Section (6.7.27) entities. Sections: Creates a plan section for each floor. Elevations: Creates four elevations (Front, Back, Left and Right).

Procedures

Using BIMIFY in your BIM model (2.8.1)

Related Commands

BimAttachSpatialLocation (6.7.4) - allows to attach or remove a spatial location to a selection of entities. BimClassify (6.7.5) - classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier). BimRoom (6.7.26) -Creates BIM Room entities. BimSection (6.7.27) - creates a BIM section entity. BimSpatialLocations (6.7.31) - opens the Buildings & Stories manager. This explorer allows to create and edit the buildings and stories.

Figure 6.274



6.7.14 BimInsert

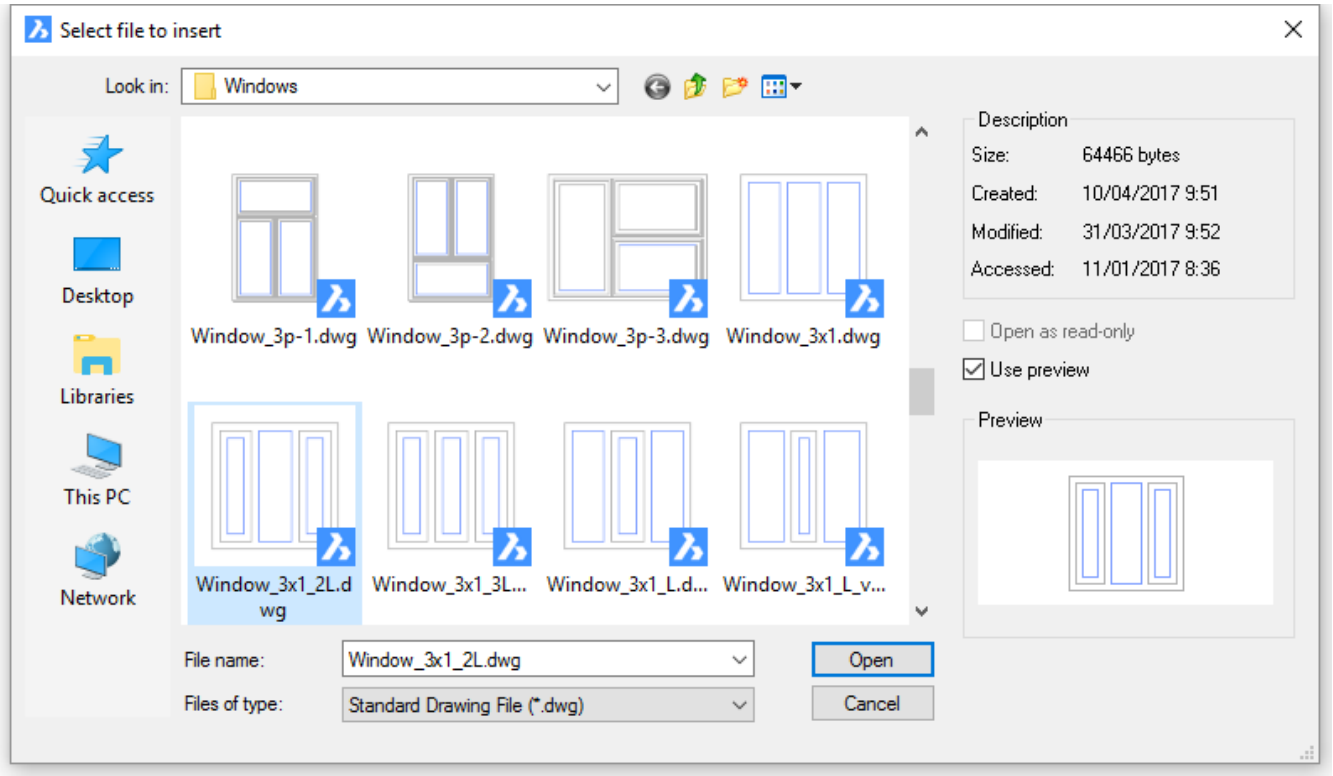
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Inserts a window or door in a solid. Inserts a copy of a window or door in a solid. Inserts a RFA* file as a BIM component into the current drawing. Make sure Dynamic UCS (DUCS (1.12.2)) and Dynamic Dimensions (DYN (1.12.2)) are active. * RFA import is a work-in-progress. Some general guidelines about the current V19 RFA import functionality are:

1. We don't support RFA files older than 2011.
2. Files from 2011 to 2014 get imported, but since there was a major format change from 2014 to 2015, there might be issues with nested families.
3. Files from 2015 to 2018 get imported correctly. At the moment, we import extrusions, blends, revolutions, and sweeps.
4. Only the geometry of the family is imported, no parameters nor properties.
5. We are working on RFA 2019 support.

Accessing the Command

Command bar: **biminsert** Menu bar: **BIM| BIM Insert** Toolbar: **BIM |** Quad: **BIM |** : **BimInsert** Displays a File dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimInsertSelectFileDialog Select a window or door to insert, then double click or click the **Open** button. Select insertion point or [Edit inserted entity] : **(Move the cursor over a solid; the face under the cursor highlights: The UCS aligns dynamically with the selected face; if the edge by which the cursor enters the solid is horizontal or vertical, the X-axis of the UCS is horizontal, otherwise the X-axis is aligned with the edge; the Z-axis is perpendicular to the face of the solid. Specify point to insert.)** Hit the Ctrl-key to switch between Select insertion (6.7.14) point and Edit inserted entity (6.7.14) mode.

Figure 6.275



Command Options

Option	Description
Select insertion point (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BIMEdit inspect.png})	Allows to specify the insertion point. Hit the Ctrl-key to switch to Edit inspect property (6.7.14) mode. From the middle of each edge of the placement rectangle dynamic dimensions display to the nearest parallel edge on the face of the solid. Press the TAB key to cycle the fields. Type a value in the dynamic dimension fields, then press the TAB key, to lock the field which then turns red. Only one of two parallel dimensions can be locked. If a dimension value is entered, and the other, parallel dimension was already locked, it will be unlocked.. Press Enter or click to insert the selected block. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/

1. Locked field
2. Current field
3. Unlocked fields
4. Unlocked fields

Edit inserted entity (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BIMInsertTipsOtherJ})	Allows to specify the width and height of the placement rectangle of the insert. Prompts you: **Edit [Height/Width/Done] <Height>:** Two dynamic dimension fields show the current width and height of the placement rectangle. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ Type a value in the current dynamic dimension field, then press the TAB key to switch to the other field. Hit the Ctrl-key to switch to Select Insertion Point (6.7.14) mode.

Procedures

Inserting windows or doors (5.3.9) Inserting a copy of an insert (**??**)

Related Commands

BmUnlink (6.8.23) - breaks the connection between a component insert (e.g. a Window) and a 3D solid (e.g. a Wall) retaining the opening. BimWindowCreate (6.7.36) - creates a fully parametric window based on a closed contour. BimWindowUpdate (6.7.37) - updates the opening made by a window or door in a solid, in case the definition changed in such way that the opening was not correctly updated automatically.

Figure 6.276

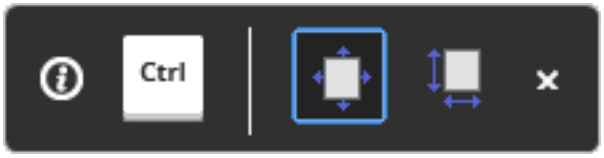


Figure 6.277

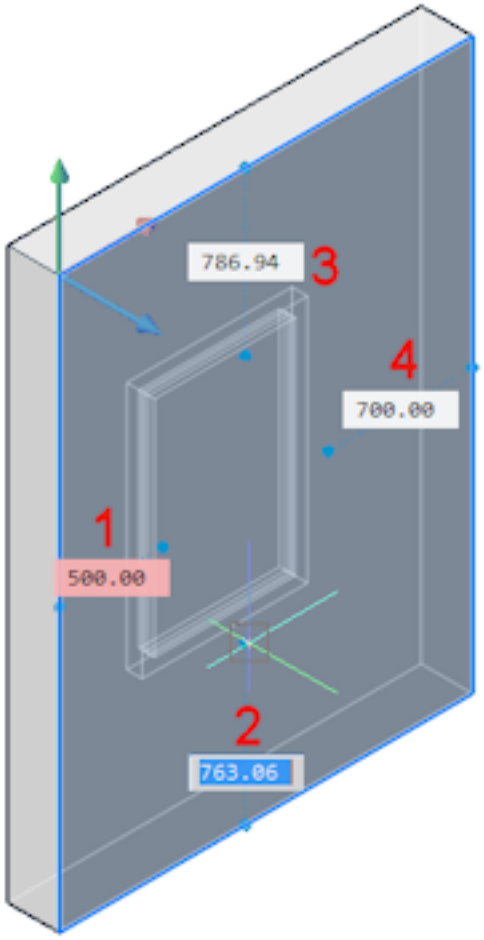


Figure 6.278



Figure 6.279

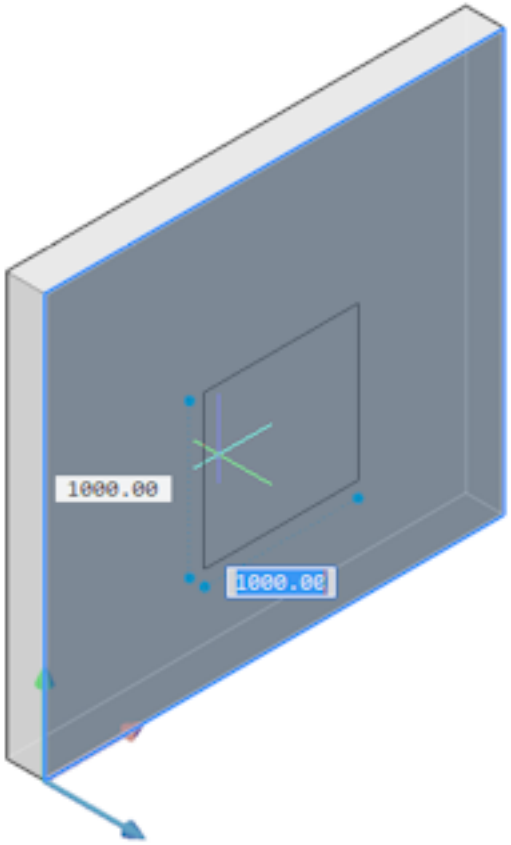


Figure 6.280



6.7.15 BimLinearSolid

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Creates a chain of linear solids.

Accessing the Command

Command bar: **bimlinearsolid** Menu bar: **BIM | Linear Solid** Toolbar: **BIM** Quad: **Model** Drag a profile from the **BIM Libaries - Profiles** panel onto the graphic screen. : **BimLinearSolid Prompts you in the command bar:** Set start point or [Follow/Rectangular/Circular/Dialog/select in Model/adJust profile] <Last point>: **(Pick a point or choose an option.)** Set next point or [Angle/Length/Undo/Quarter turn/Rotate/choose other Profile/adJust profile]: **(Pick a point or choose an option.)** Set next point or [Angle/Length/Undo/Quarter turn/Rotate/choose other Profile/adJust profile]: **(Pick more points, or choose an option, or press Enter to stop.)**

Command Options

Option	Description
Last point	Press Enter to use the most recently specified point.
Follow	Draws the next linear solid at the same angle as the previous one. **Length of line** - **(specify the length of the linear solid.)**
Rectangular	Switches to a default rectangular profile.
Circular	Switches to a default circular profile.
Dialog	Opens the Profiles (6.7.17) dialog box. Select a profile, then click the OK button on the Profiles dialog box. Returns to the previous prompt.
Select in model	Allows to select a profile in the model; prompts you: **Pick a profile** - **(Do one of the following)**: • Click a closed 2D entity or a region in the model. • Click an existing linear solid. Returns to the previous prompt.
Adjust profile	Allows to adjust the dimensions of the default rectangular and circular profile; prompts you: For rectangular profile: **Width of rectangle <250> [Height]** - **(Type a value or press Enter to accept the current value.) Height of rectangle <120> [Width]** - **(Type a value or press Enter to accept the current value.)** For circular profile: **Radius of circle <100>** - **(Type a value or press Enter to accept the current value.)** Returns to the previous prompt.
Angle	Draws at a user-specified angle and distance; prompts you: **Angle of line** - **(specify the angle (measured counter-clockwise from the positive x-axis)). Length of line** - **(specify the length of the linear solid.)**
Length	Draws at a user-specified distance and angle; prompts you: **Length of line** - **(specify the length of the linear solid.) Angle of line** - **(specify the angle (measured counterclockwise from the positive x-axis)).**
Undo	Erases the last linear solid.
Quarter turn	Rotates the profile 90° counter clockwise.

Rotate	Allows to rotate the profile by a user defined angle: prompts you: **Angle to rotate profile** - **(specify the rotation angle.)** Positive values rotate the profile counter clockwise.
Choose other profile	Allows to use a different profile for the next linear solid; prompts you: **Choose a profile [Rectangular/Circular/Dialog/select in Model]** - **(choose an option.)**
Adjust profile	Prompts you to pick a profile for the next linear solid in the model: **Pick a profile** - **(Do one of the following):** • Click a closed 2D entity or a region in the model. • Click an existing linear solid.

Related Commands

BimAddEccentricity (6.7.1) - allows to control the relative position of the axis of a linear solid. BimApplyProfile (6.7.2) - applies a profile to linear entities and linear solids. BimProfiles (6.7.17) - opens the profiles library in the current project and the referenced library.

Figure 6.281



6.7.16 BimMultiselect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Selects linear solids with coplanar and/or parallel axes.

Accessing the Command

command bar: **bimmultiselect** menu bar: **BIM | multiselect** : **bimmultiselect Prompts you in the command bar:** Select single entity/subentity:**(Select a solid or a solid face.)** Select type of selection [all Coplanar axes/coplAnar and parallel axes/all Parallel axes] <Accept>: **(Choose an option or press Enter to accept selection of the highlighted linear solids.)** If a face of a linear solid is selected, the corresponding faces of the linear solids with coplanar/parallel axes are selected.

Command Options

Option	Description
All coplanar axes	Selects all solids of which the axis is coplanar with the axis of the selected solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/
Coplanar and parallel axes	Selects all solids of which the axis is coplanar and parallel with the axis of the selected solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/
All parallel axes	Selects all solids of which the axis is parallel with the axis of the selected solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/

Related Commands

DmSelect (6.12.26) - selects edges and faces of 3D solids or surfaces based on their geometric properties. SelectAlignedFaces (6.27.16) - selects all faces in a model which are coplanar with a selected face. SelectAlignedSolids (6.27.18) - selects all solids in a model of which a face is coplanar with a selected face. SelectConnectedFaces (6.27.19) - selects all faces in a model which are connected to a selected face. SelectConnectedSolids (6.27.20) - selects all solids in a model which are connected to a selected face.

Figure 6.282

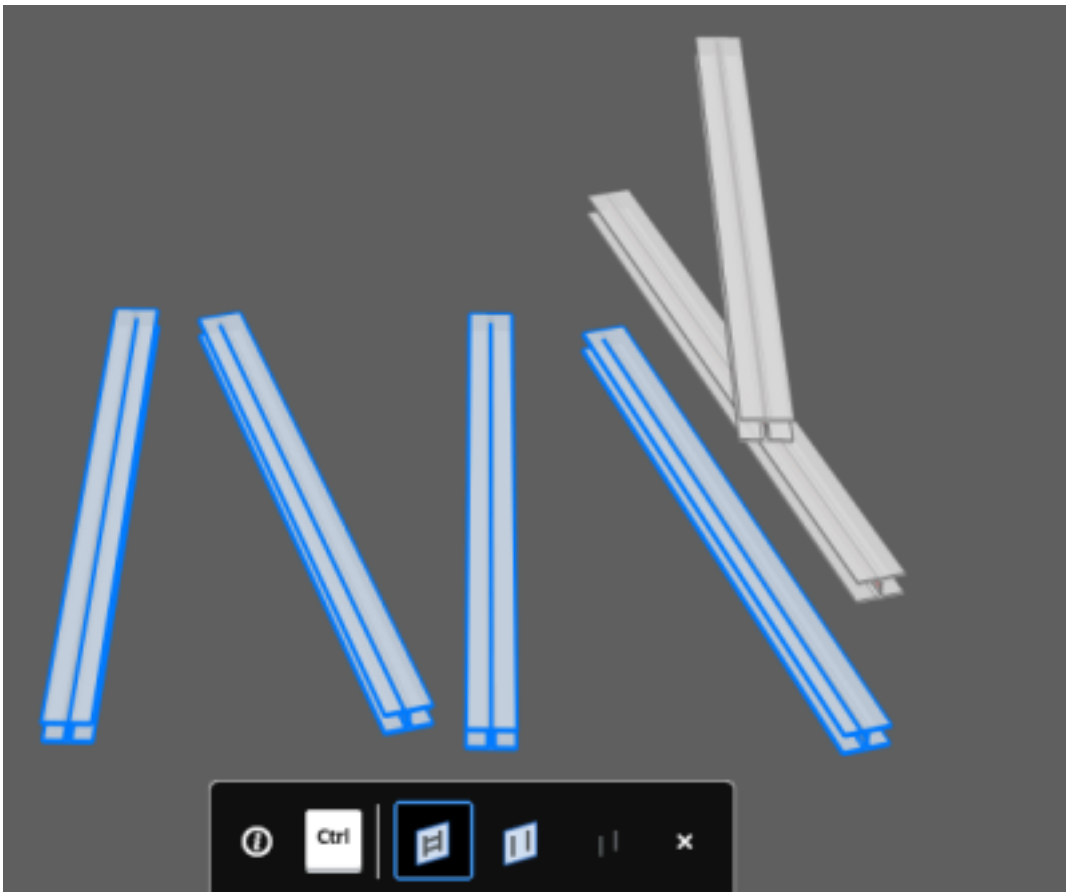


Figure 6.283

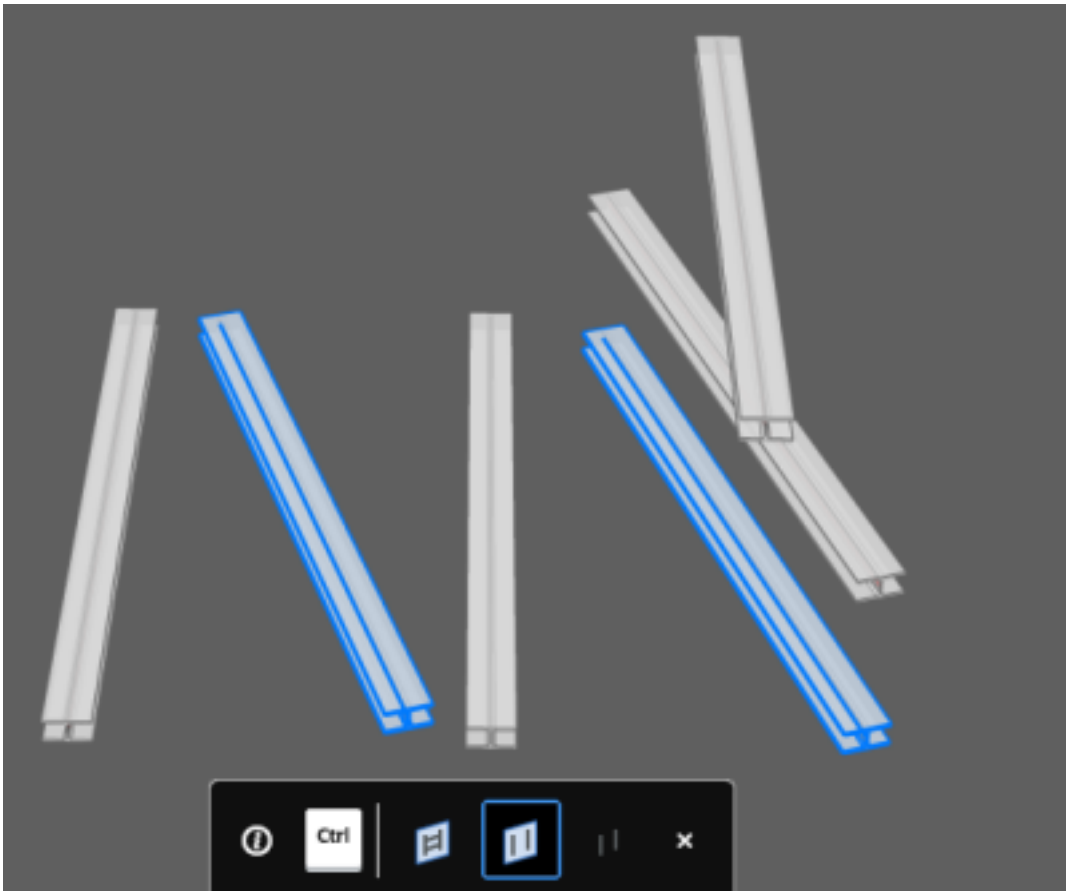


Figure 6.284

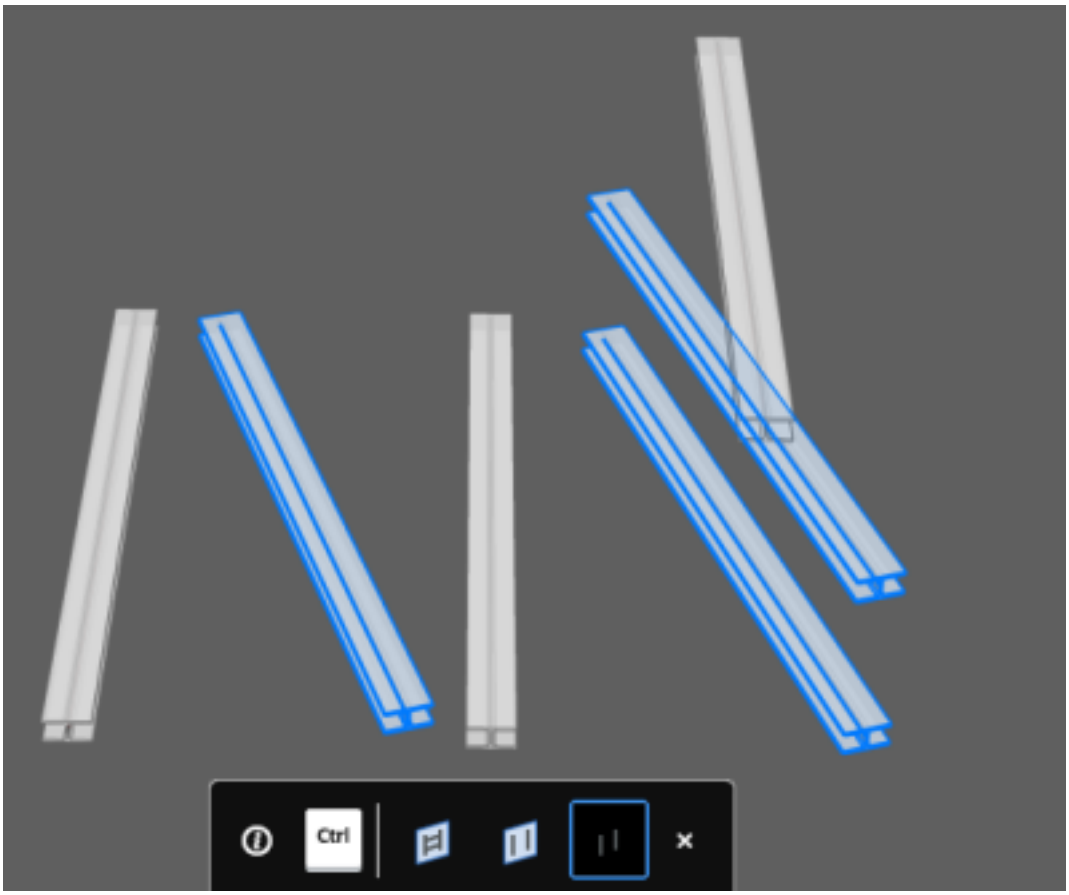


Figure 6.285



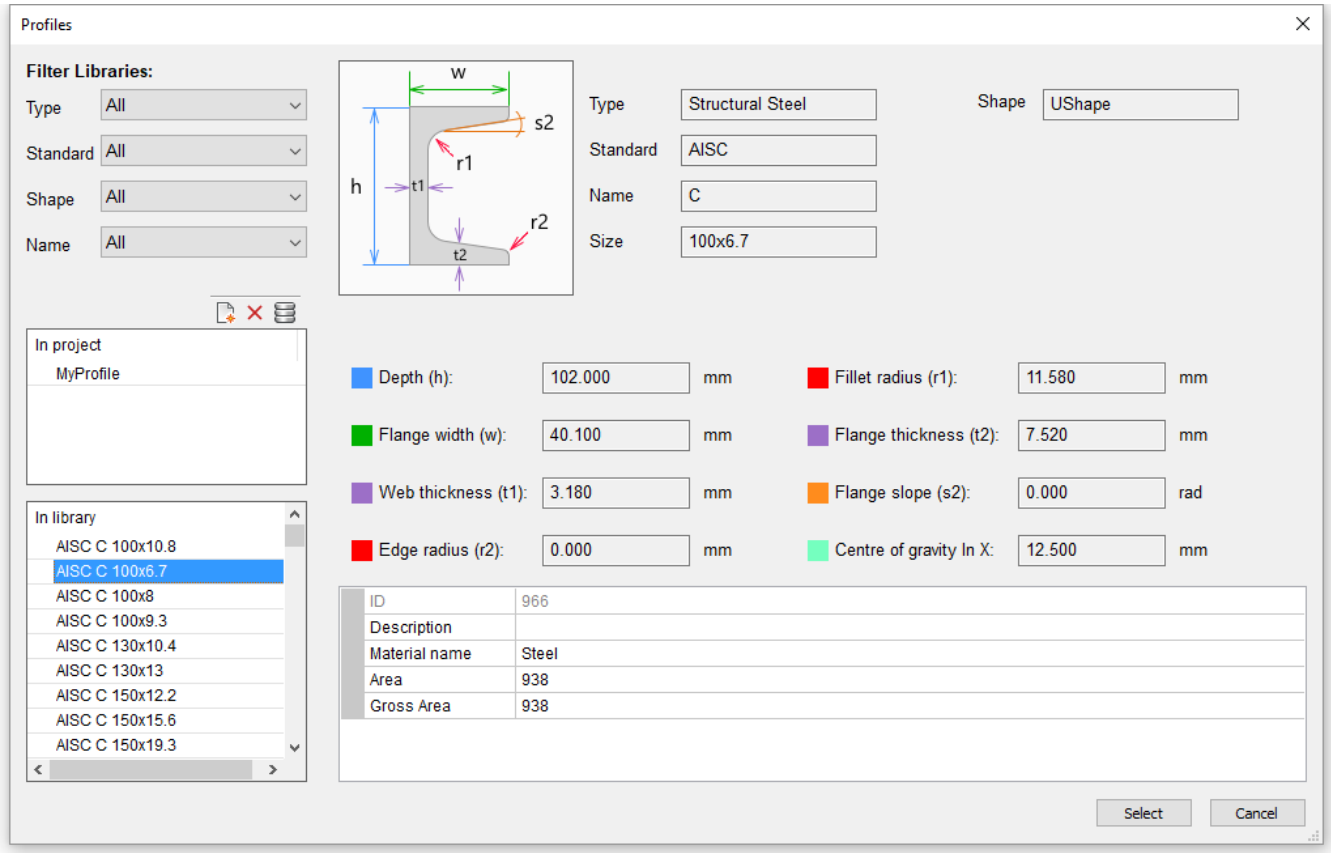
6.7.17 BimProfiles

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Opens the profiles library in the current project and the referenced library.

Accessing the Command

command bar: **bimprofiles** Bim Libraries panel: **Profiles Toolbar** | : **BimProfiles** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ProfilesDialog.png}) Select a profile in the In project or In library list to edit its properties. Drag and drop profiles to copy them from the Library to the Project and vice versa.

Figure 6.286



Command Options

	Description
Filters	Allows to filter the profiles list by **Type** , **Standard** , **Shape** and **Name** .
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/Profiles/Standard.png})	The **Type** options are: **All** , **Generic** , **Structural Concrete** , **Structural Steel** .
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/Profiles/Standard.png})	The **Standard** options are: **All** , **AISC** (Americal Institute of Steel Construction), **BS** (British Steel), **EURO** (European standard steel profiles), **User Defined.**
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/Profiles/Shape.png})	The **Shape** * options are: **All** , **Circle Hollow** , **I Shape** , **L Shape** , **Rectangle** , **Rectangle Hollow** , **T Shape** , **U Shape** *Depending on the selected **Standard** some **Shape** options are not available.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/Profiles/Shape.png})	The **Name** options depend on the selected **Type** , **Standard** and **Shape** option. E.g. available options for **Structural Steel / AISC / I Shape** are: HP, M, S, W.

Tools



Creates a new profile. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/

- **Type:** Select a type. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands
- **Name:** Type a name
- **Size:** Indicate the size of the profile.
- **Shape:** Do one of the following:
  - Select a shape. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comm Fill out the properties grid for the selected profile: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comm
  - Click the **Pick Profile in Model** button. Prompts you: Pick a profile - the **Profiles** dialog box temporarily closes to let you select a closed 2D entity or the boundary entities of an enclosed area. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comm Optionally, click the buttons at the bottom of the profile preview to rotate or mirror the profile.



Deletes the selected profile
Select a profile, then right click to display the context

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ProfilesContextMenu.png})

- **New Profile:** Creates a new profile.
- **Duplicate:** Creates a new profile as a duplicate of the selected profile.
- **Delete:** Deletes the selected profile.
- **Copy to Project:** Copies the selected profile to the Project library.

Procedures

- Applying profile to linear solids (2.4.2)

Related Commands

BimAddEccentricity (6.7.1) - allows to control the relative position of the axis of a linear solid. BimApplyProfile (6.7.2) - applies a profile to linear entities and linear solids. BimLinearSolid (6.7.15) - creates linear solids. BimRecalculateAxis (6.7.44) - recalculates the axis of linear building elements and repositions it back on the center line of a linear element. BLCompositions (6.7.38) -Opens the compositions library in the current project and the referenced library.

Figure 6.287



Figure 6.288



Figure 6.289



Figure 6.290



Figure 6.291

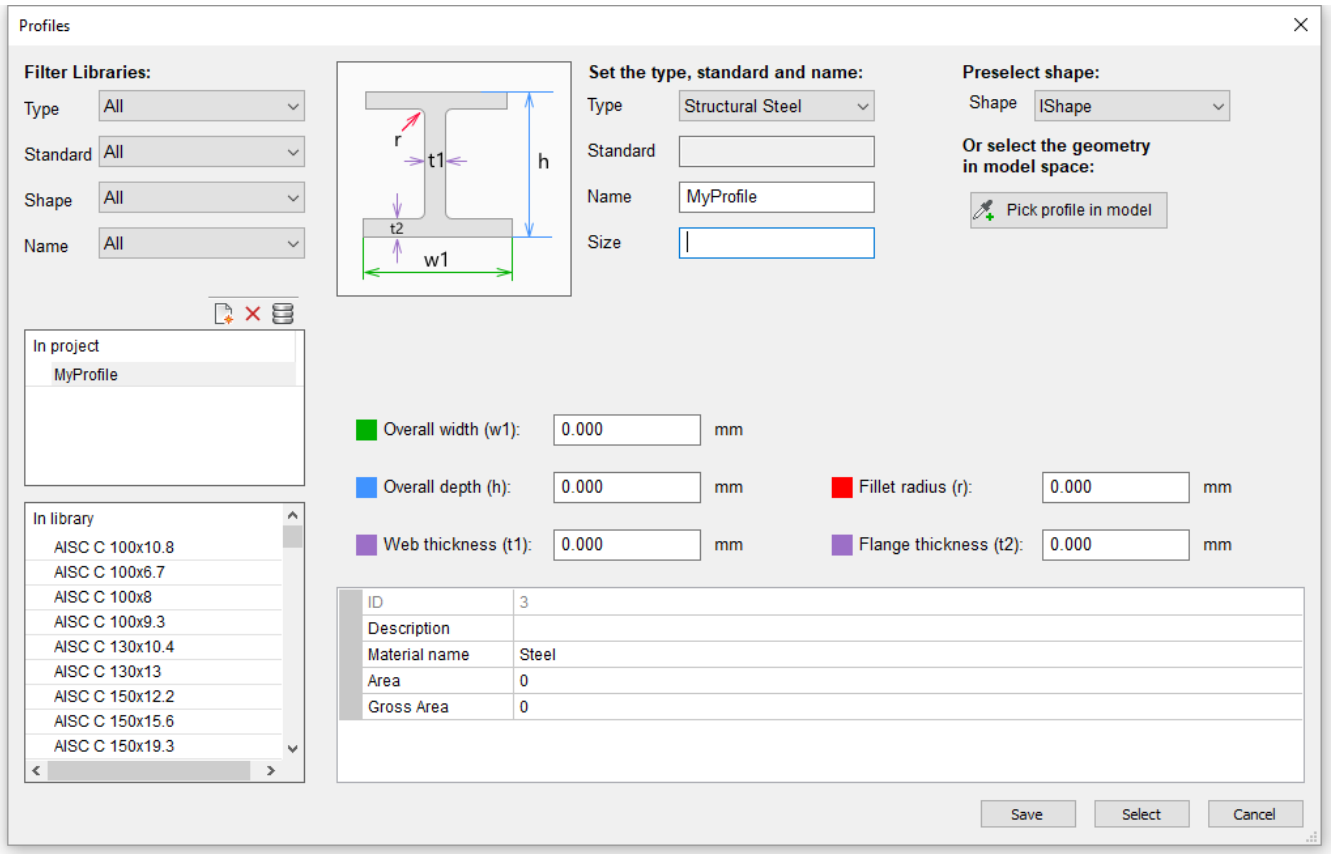


Figure 6.292

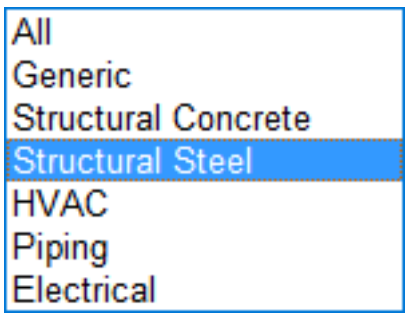


Figure 6.293

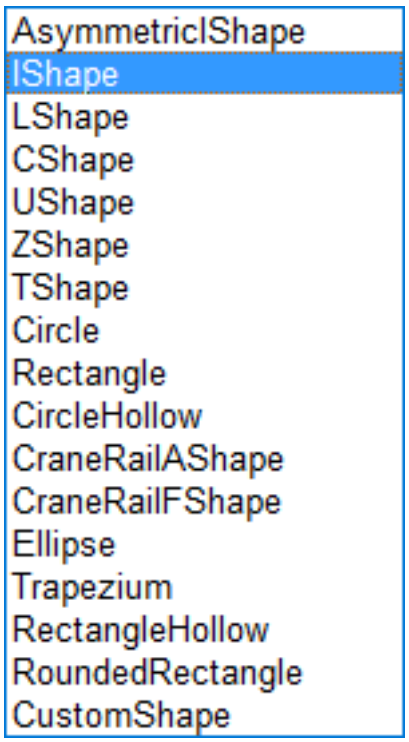


Figure 6.294

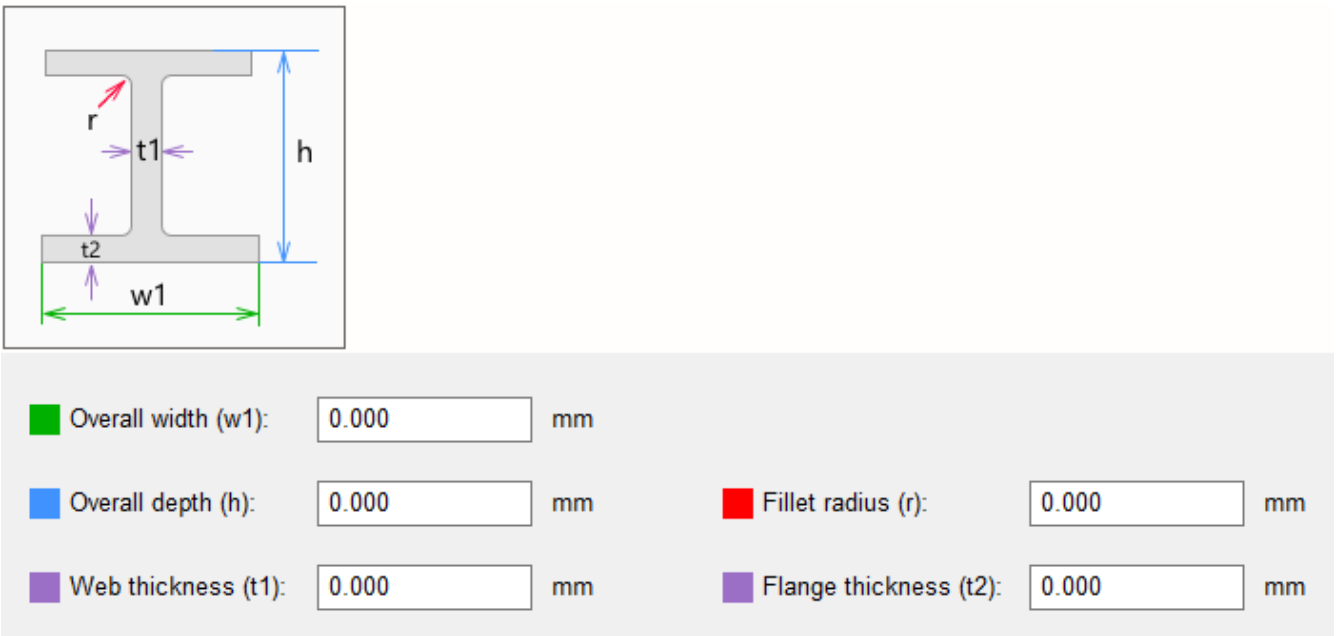


Figure 6.295

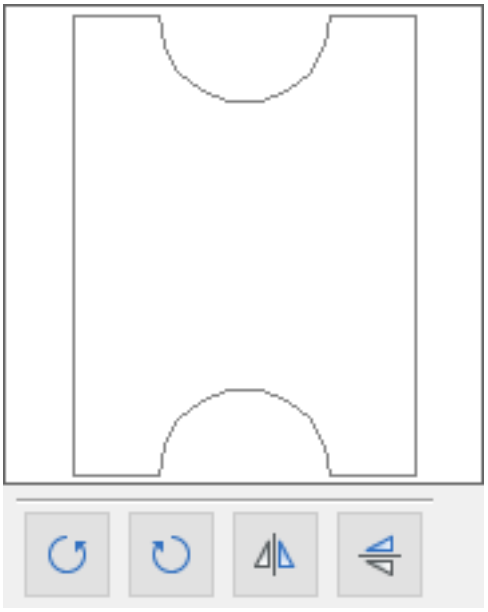


Figure 6.296

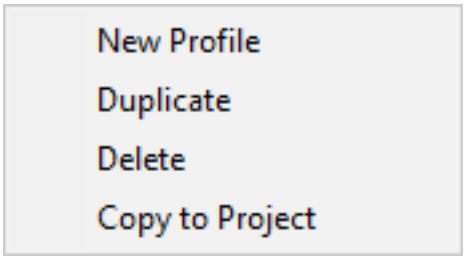


Figure 6.297



6.7.18 BimPatch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Allows to replace part of a generated section. The command is valid in model space only.

Accessing the Command

command bar: **bimpatch** : **BimPatch Prompts you in the command bar:** Specify start point of rectangle (in Model Space): **(Specify a point.)** Specify end point of rectangle: **(Specify a point or use the dynamic dimension fields to specify the dimensions of the rectangle).**

Note	If BricsCAD responds: 'Cannot apply patch: no intersection points found' set the GENERATEASSOCVIEWS system variable ON. Lookup the variable in the Settings (1.13.11) dialog under Dimensions or type GENERATEASSOCVIEWS in the command bar and choose ON.

A cut-out is generated as a nested block in the BIM section. BIM patch blocks are named Patch1, Patch2, .. The rectangular boundary of a BIM patch is placed on layer BIMPATCH_BOUNDARY. Double click the BIM section to launch RefEdit. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimPatchRefEditDia
Select the BIM Patch in the **Reference Name** tree, then click the **OK** button. The **Reference Edit** dialog box closes. Edit the BIM Patch. Use RefClose (6.26.11) to save the changes in the BIM Patch. When the corresponding BIM section is updated, BricsCAD will check if the generated section result within that rectangle is identical to the result which was generated at the time the BIM patch was created:

- If not, then the user edited BIM patch is preserved but the boundary is colored red.
- If it is, the user edited BIM patch is preserved.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimPatchRed.png}) Intersection points and endpoints of curves originating from 3d objects are used to find the position of a BIM patchrectangle. When objects are modified or moved in the 3d model in such way that the position of these intersection points change, but the geometry inside the BIM patchrectangle is not affected, then BricsCAD will relocate the user edited BIM patch.

Figure 6.298

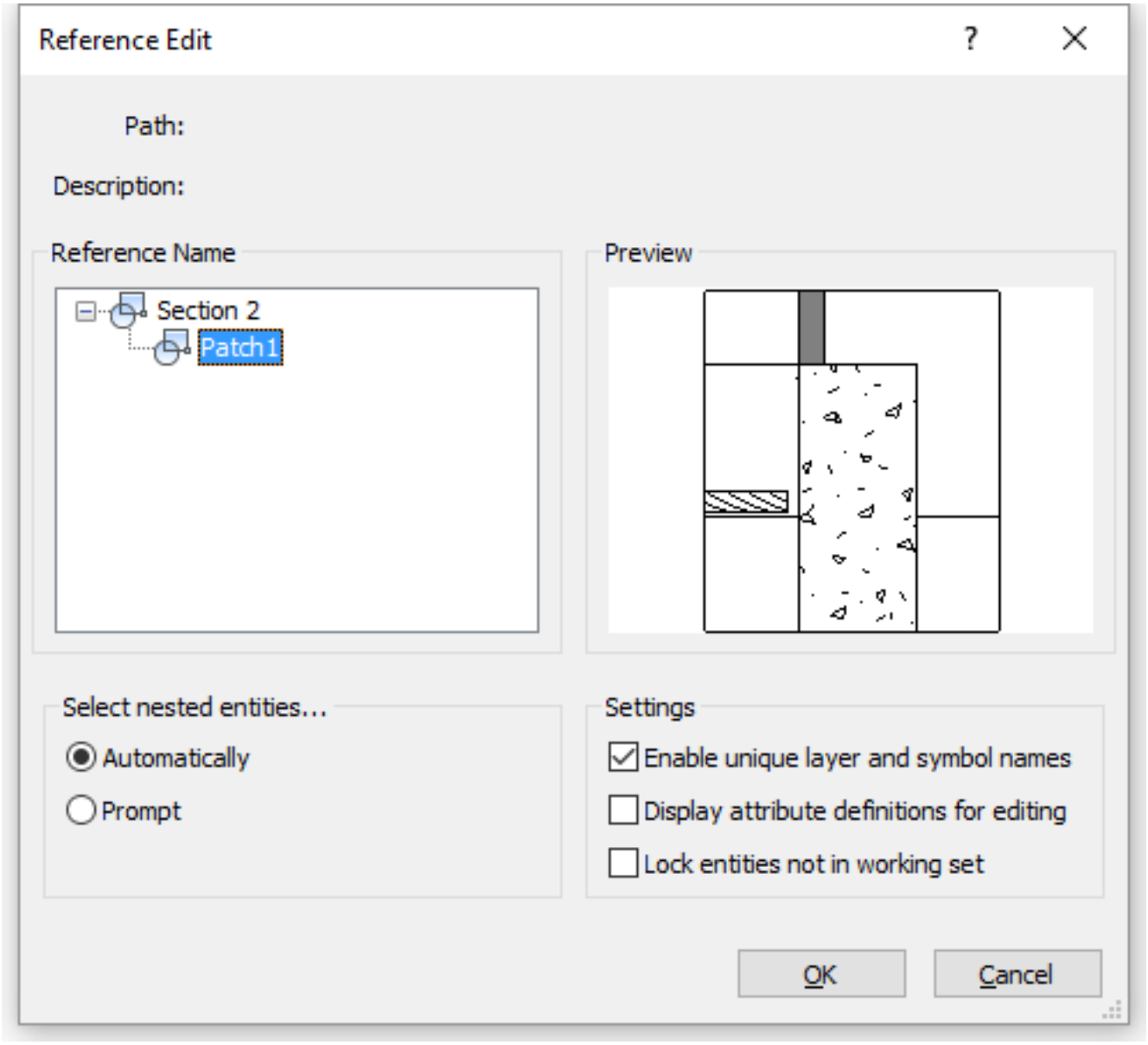
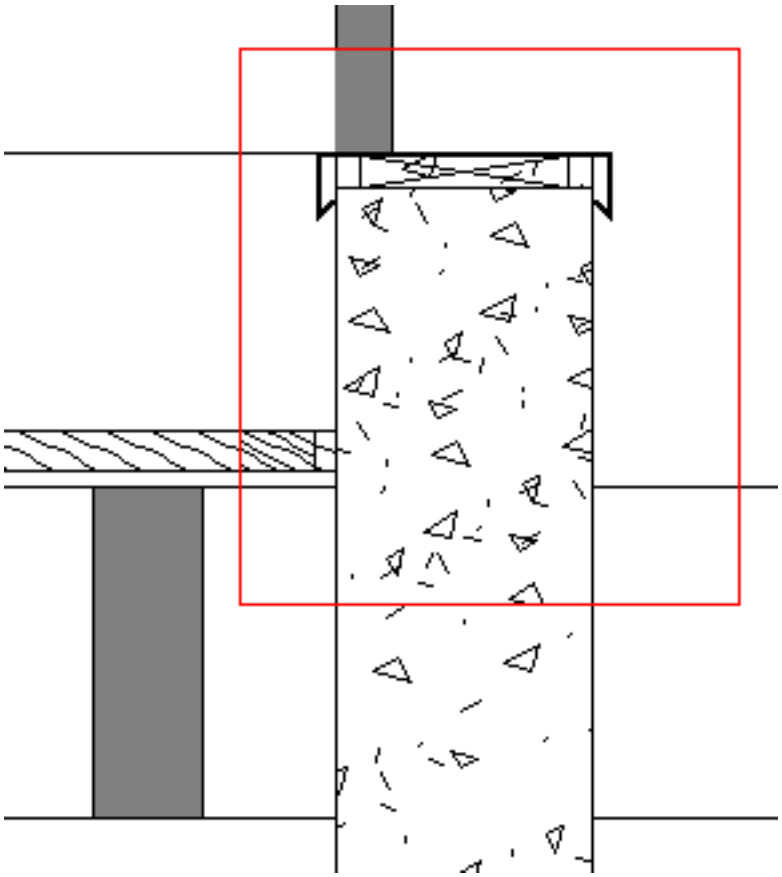

Reference Edit
Path:
Description:
Reference Name
Section 2
Patch1
Preview
Select nested entities...
Automatically
Prompt
Settings
Enable unique layer and symbol names
Display attribute definitions for editing
Lock entities not in working set
OK
Cancel

Figure 6.299



Command Options

No command options.

Related Commands

BimSection (6.7.27) - creates a BIM section entity. BimSectionOpen (6.7.28) - opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing. BimSectionUpdate (6.7.29) - exports or updates the result of a BIM section in a drawing. RefEdit (6.26.12) - dits block references and externally-referenced drawings through a dialog box.

Figure 6.300



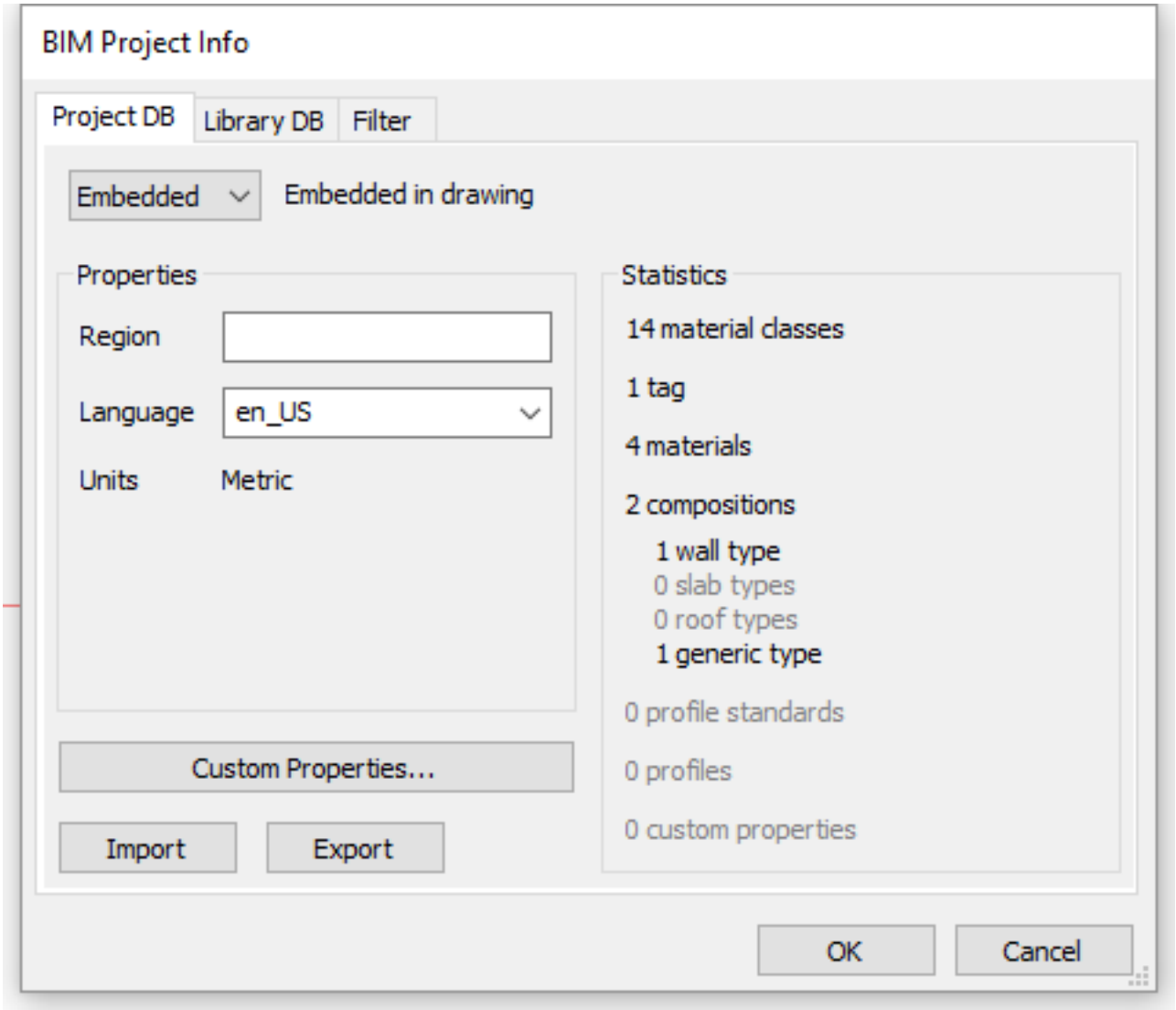
6.7.19 BimProjectInfo

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Opens the BIM Project Info dialog box.

Accessing the Command

 Command bar: **bimprojectinfo** : **BimProjectInfo** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsy

Figure 6.301


BIM Project Info
Project DB
Library DB
Filter
Embedded
Embedded in drawing
Properties
Region
Language
en_US
Units
Metric
Statistics
14 material classes
1 tag
4 materials
2 compositions
1 wall type
0 slab types
0 roof types
1 generic type
0 profile standards
0 profiles
0 custom properties
Custom Properties...
Import
Export
OK
Cancel

Procedures

To set the library database (2.6.5) To set the project database (2.6.5) To set the project filter (2.6.5)

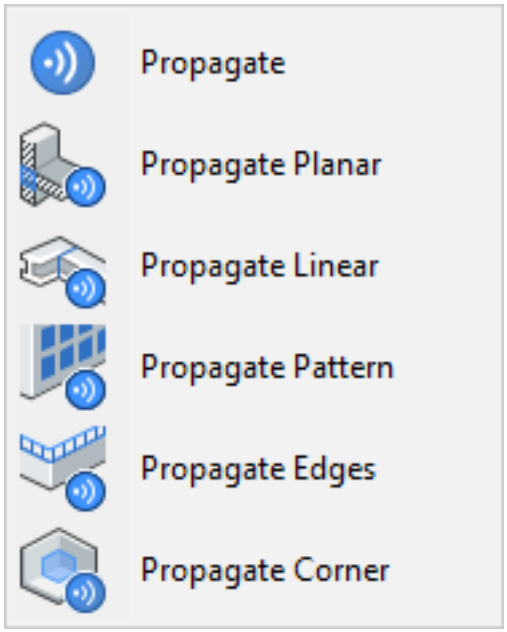
Related Commands

BimProfiles (6.7.17)- opens the profiles library in the current project and the referenced library. BLCompositions (6.7.38)- opens the compositions library in the current project and the referenced library. BLMaterials (6.7.39)- opens the physical materials library in the current project and the referenced library.

Figure 6.302



Figure 6.303



6.7.20 BimPropagate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Maps details (solids, holes, finishing geometry, ...) related to selected base solids to similar base solids in the model and optionally outlines the detail in a grid. Five variants of BIMPROPROGATE work on a limited set of situations:

- BIMPROPAGATEPLANAR (6.7.25) Propagation of connections between planar elements (walls/slabs/roofs).
- BIMPROPAGATELINEAR (6.7.23) Propagation of connections between linear elements (beams, columns, pipes, ducts and their connections to walls and slabs).
- BIMPROPAGATEPATTERN (6.7.24) Propagation of a single element on a flat surface to multiple locations and grids; can be used to propagate lights, light switches, windows, air diffusers, etc.
- BIMPROPAGATEEDGES (6.7.22) Propagation along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc.
- BIMPROPAGATECORNER (6.7.21) Propagates details connected to three planar base solids (i.e., corners). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/PropagateCommand

Accessing the Command

 Keyboard:**bimpropagate** Menu:**BIM | Propagate | Propagate** Toolbar:**BIM Propagate** Quad: **Model | Propagate** Ribbon Tab | Panel: **Model | Propagate : BimPropagate Promps you:** Select base solids: **(Select a solid (1).)** Entities in set: 1 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/PropagateSelect1b.png Select base solids: **(Select a solid (2).)** Entities in set: 2 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 Select base solids: **(Select more solids or press Enter.)** Select detail objects or detail sub entities (optional): **(Select detail objects (3)or press Enter.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM Propagate detail as an extrusion of this 2D section? [Yes/No/Inflate first] <Yes>: **(Press Enter or choose an option.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/PropagateDetail.png}) Searching for similar connections Creating the suggestions Showing 4 application details Resolving conflicts and extensions Duration start BIMPROPAGATE: 0.571000 Showing 3 visible suggestions of total 3. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtechniques/PropagateOriginalDetail.png}) Choose next action [Apply all/Cancel] <Apply all>: **(Press Enter to apply the selected suggestions.)** Applying all xx visible suggestions

Figure 6.304

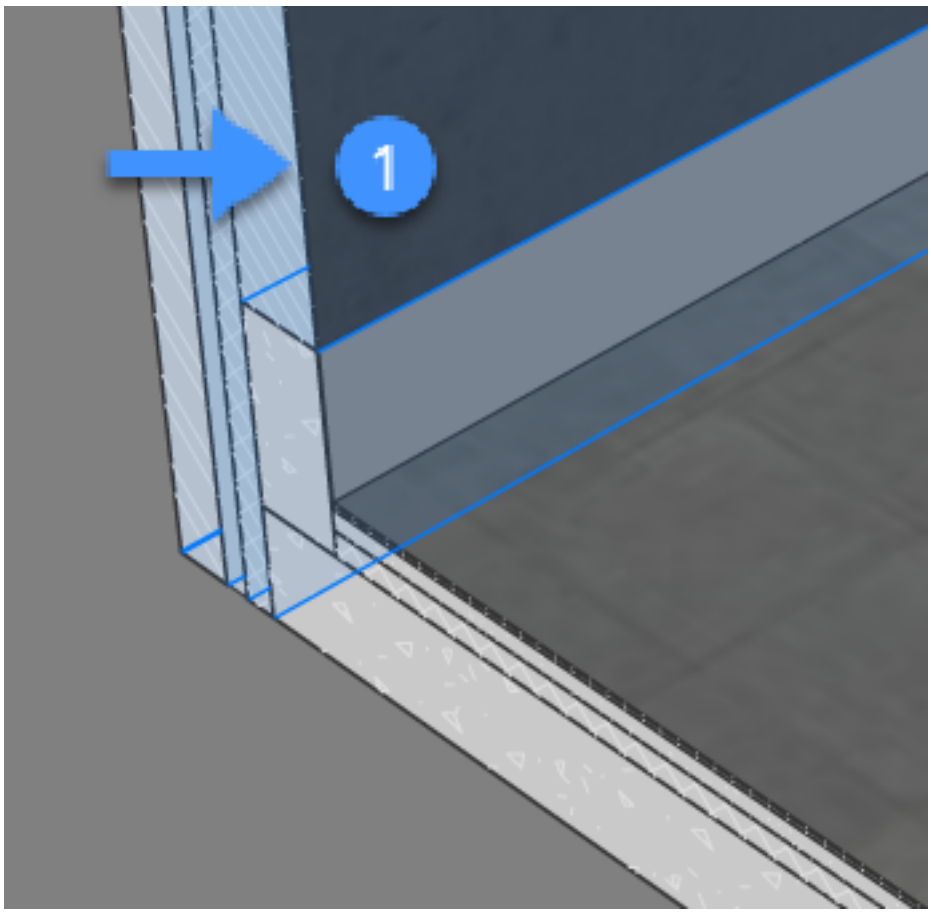


Figure 6.305

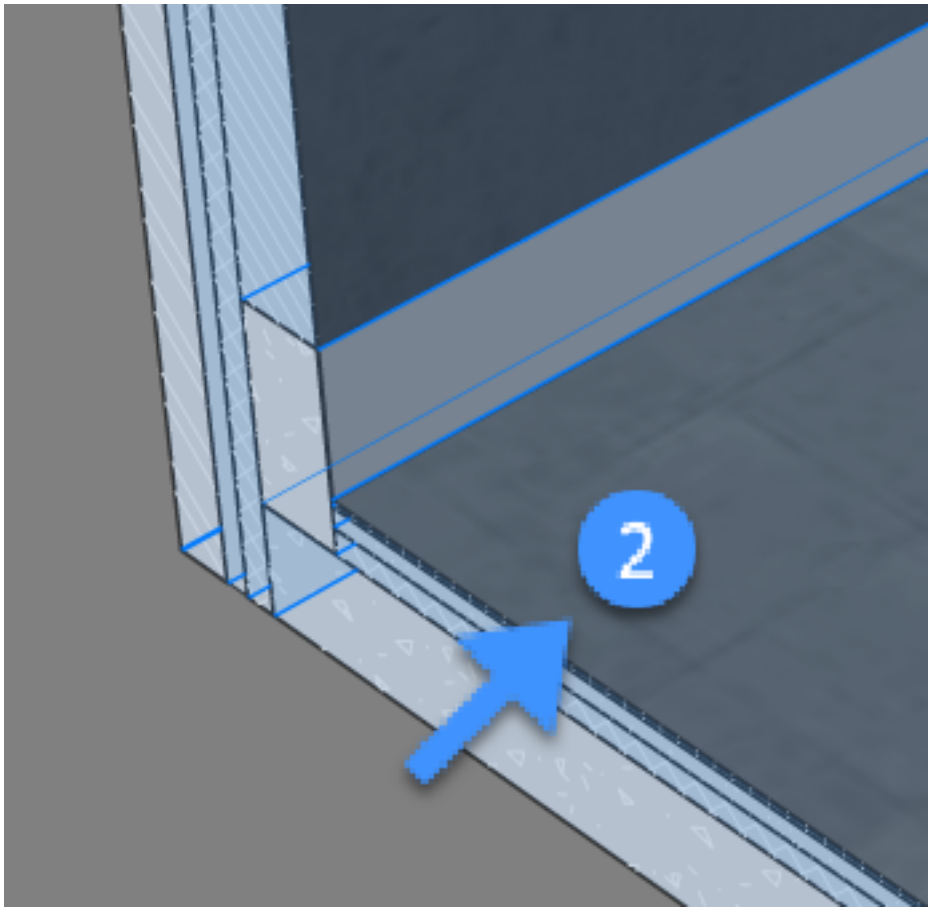


Figure 6.306

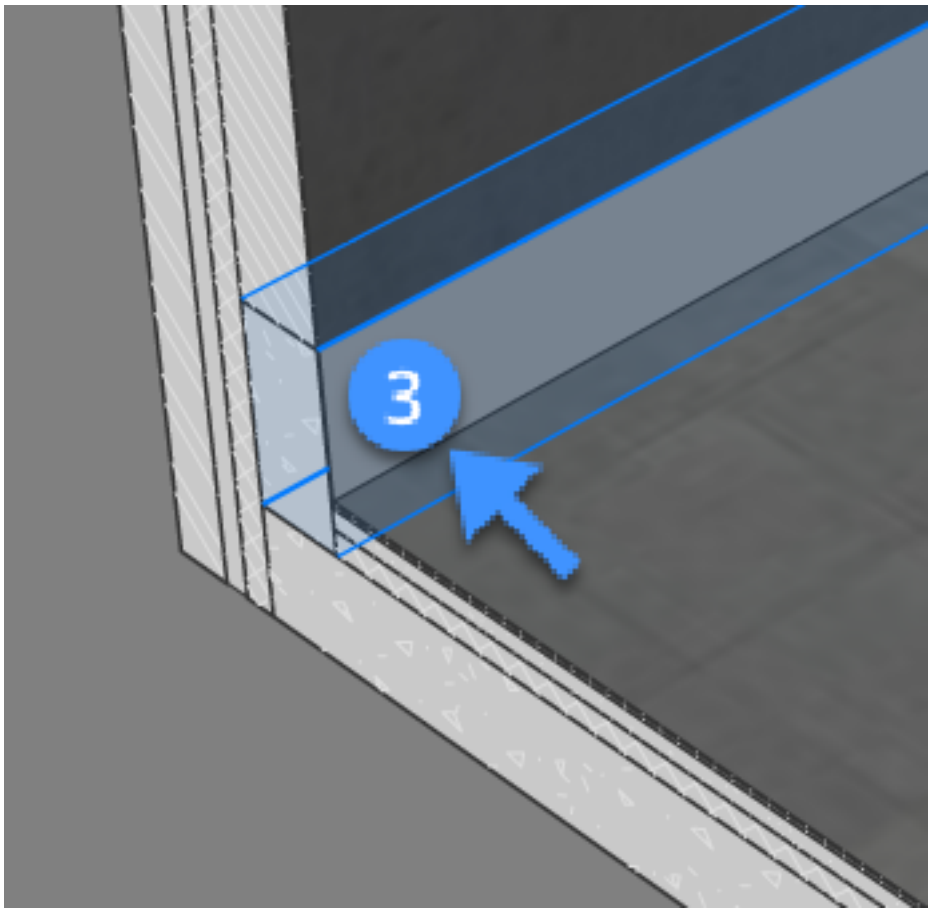


Figure 6.307

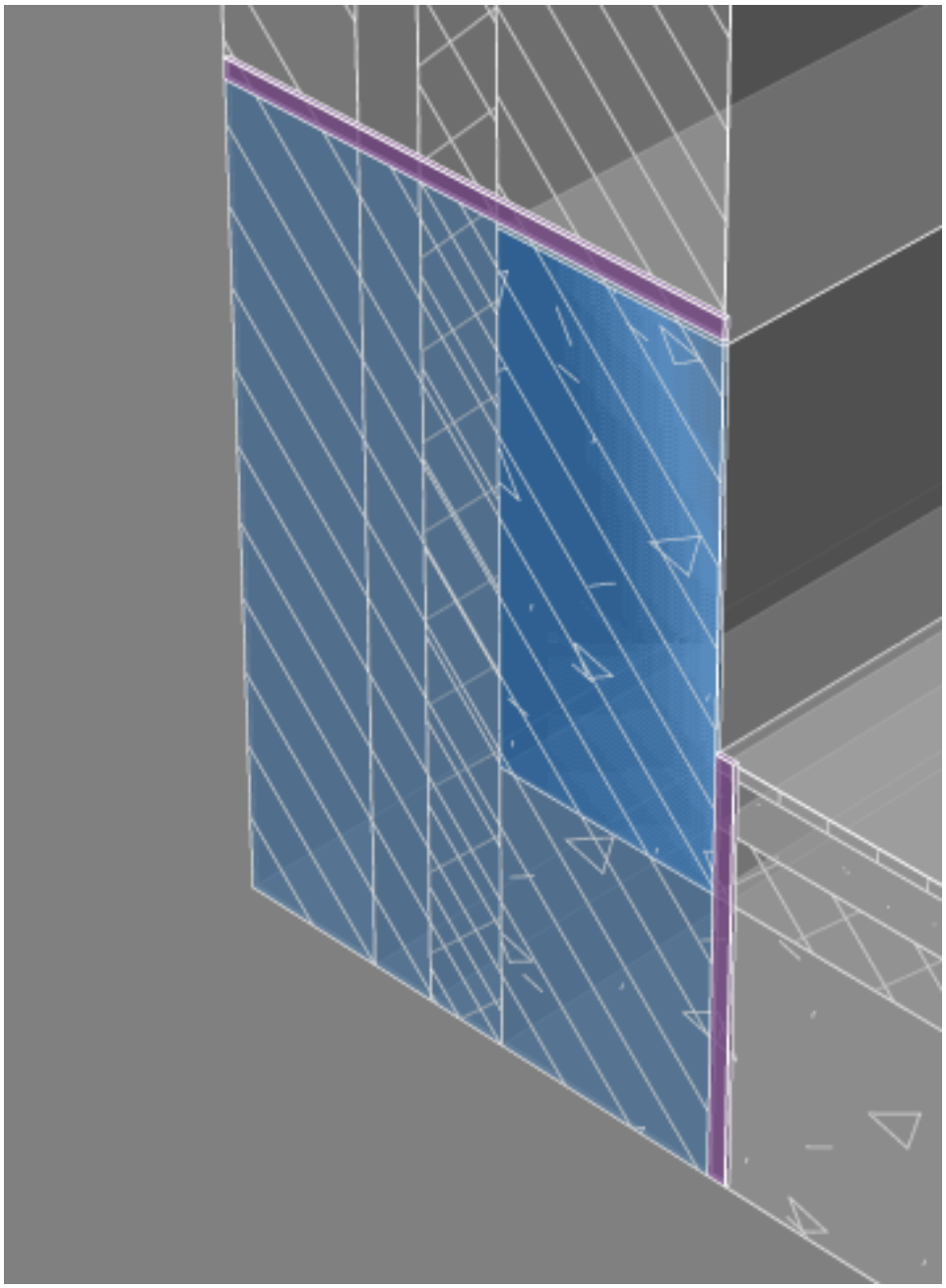
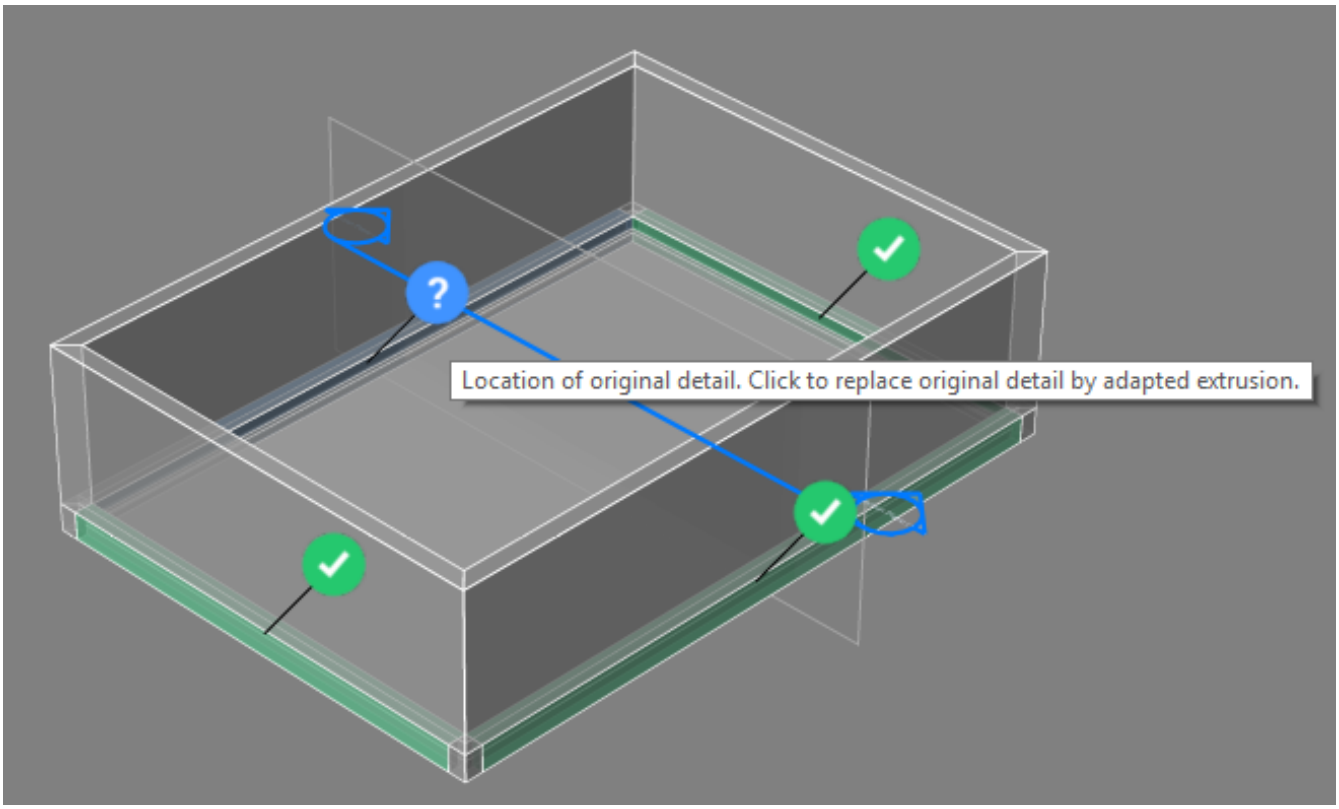


Figure 6.308


Location of original detail. Click to replace original detail by adapted extrusion.

Command Options

Option	Description
Apply All	 : Selected suggestion. Click to switch to not selected.  :The detail will be applied, but there are several alternatives. Hovering over the icon to cycle between alternatives  :The detail will be applied, but interferences were detected  : Not selected suggestion. Click to switch to selected.  : Location of the original detail. Hover over the icon for more information.Click to apply original detail by adapted extrusion. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BIM/Modelingtec Use window selection to select multiple widgets (): • Click the left mouse button then move the mouse to the right to select all widgets inside the selection window. • Click the left mouse button then move the mouse to the left to select all widgets inside or overlapping the selection window. • Keep the Shift-key down while window selecting to unselect widgets. Click one of the selected widgets to modify the state of the entire selection set.
Inflate first	Expands the sensitive zone (left) to include similar solids which are not connected yet (right) in the model (right).The edges of the initially detected detail are moved outward over a specified distance. Prompts you: **Enter inflation distance:(Type a value).** ===ERROR WITH DOWNLOADED IMAGE=== ===ERROR WITH DOWNLOADED IMAGE===

Procedures

The five flavours of BimPropagate (2.8.2) General procedure (2.8.2)

Related Commands

BimPropagateCorner (6.7.21) -Propagates details connected to three planar base solids (i.e., corners). BimPropagateEdges (6.7.22) -Propagates a detail along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc. BimPropagateLinear (6.7.23) -Propagates connections between linear elements (beams, columns, pipes, ducts and their connections to walls and slabs). BimPropagatePattern (6.7.24) -Propagates a single element on a flat surface to multiple locations and grids; can be used to propagate lights, light switches, windows, air diffusers, etc. BimPropagatePlanar (6.7.25) -Propagates connections between planar elements (walls/slabs/roofs).

Figure 6.309

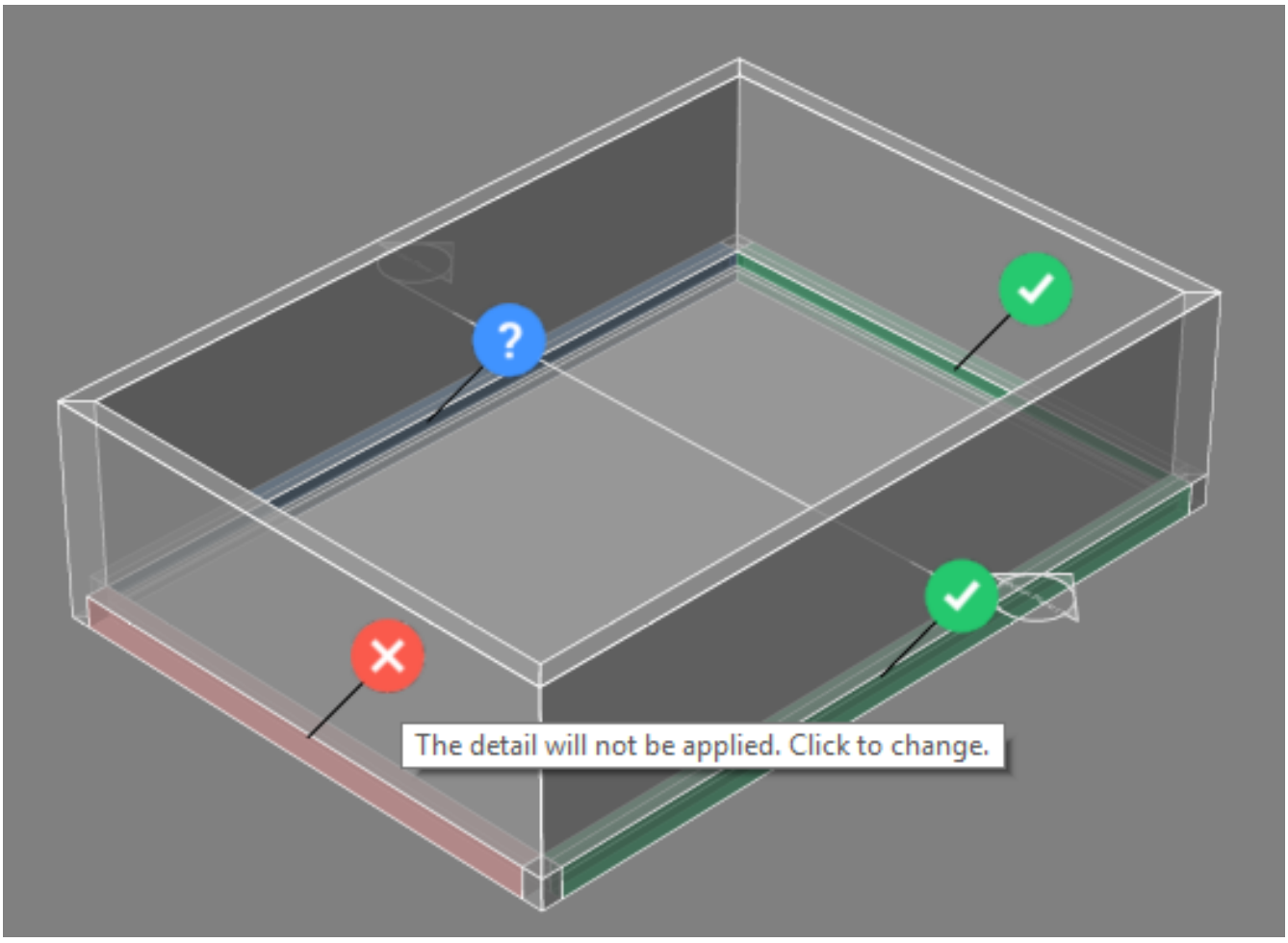


Figure 6.310



6.7.21 BimPropagateCorner

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Propagates details connected to three planar base solids (i.e. corners).

Tutorial:How to use Propagate Corner. Click here (2.7.7) to watch.

Accessing the Command

 Keyboard:**bimpropagatecorner** Menu:**BIM | Propagate | Propagate Corner** Toolbar:**BIM Propagate** Ribbon Tab | Panel: **Model | Propagate : BimPropagateCorner Prompts you:** Select at least three planar solids that form a 3D corner:**(Select the first solid.)** Entities in set: 1 Select at least three planar solids that form a 3D corner:**(Select the second solid.)** Entities in set: 2 Select at least three planar solids that form a 3D corner:**(Select the third solid.)** Entities in set: 3 Select at least three planar solids that form a 3D corner:**(Select more solids or press Enter.)** Select detail objects (solids, block refs or faces) to be included in the corner detail (optional):**(Choose an option or press Enter.)** Propagate this 3D detail? [Yes/No/Inflate first] <Yes>:**(Choose an option or press Enter.)** Searching for similar connections Creating the suggestions Showing 4 application details Resolving conflicts and extensions Duration start BIMPROPAGATE: 0.241000

Command Options

Option	Description
Select detail objects	Select additional solids, blocks or faces to be included in the corner detail.
Inflate first	Expands the sensitive zone of the detail.

Related Commands

BimPropagate (6.7.20)-Maps details (solids, holes, finishing geometry, ...) related to selected base solids to similar base solids in the model and optionally outlines the detail in a grid. BimPropagateEdges (6.7.22) -Propagates a detail along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc. BimPropagateLinear (6.7.23) -Propagates connections between linear elements (beams, columns, pipes, ducts and their connections to walls and slabs). BimPropagatePattern (6.7.24) -Propagates a single element on a flat surface to multiple locations and grids; can be used to propagate lights, light switches, windows, air diffusers, etc. BimPropagatePlanar (6.7.25) -Propagates connections between planar elements (walls/slabs/roofs).

Figure 6.311



6.7.22 BimPropagateEdges

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Propagates a detail along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc.

Tutorial: How to use Propagate Edges Click here (2.7.3) to watch

Accessing the Command


Keyboard:**bimpropagateedges** Menu:**BIM | Propagate | Propagate Edges** Toolbar:**BIM Propagate** Ribbon Tab | Panel: **Model | Propagate : BimPropagateEdges Prompts you:** Select planar solid to which edge detail is related:**(Select a planar solid (1).)** Entities in set: 1 Select planar solid to which edge detail is related:**(Select more planar solids or press Enter.)** Select detail objects to be included in the edge detail (solids or faces):**(Select the detail object(s) (2).)** Entities in set: 1 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/Prop
Select detail objects to be included in the edge detail (solids or faces):**(Select more detail objects or press Enter.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/PropagateEdgesDetail.png}) Propagate detail as an extrusion of this 2D section? [Yes/No/Inflate first] <Yes>: **(Press Enter or choose an option.)** Propagate to edges with similar orientation only? [Yes/No] <Yes>:**(Press Enter or choose an option.)** Searching for similar connections Creating the suggestions Showing 6 application details Resolving conflicts and extensions Duration start BIMPROPAGATE: 0.930000 Showing 6 visible suggestions of total 6. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/PropagateEdgesSuggestions.png}) Choose next action [Apply all/Cancel] <Apply all>:**(Press Enter to propagate the detail along all selected edges.)** Applying all 6 visible suggestions (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/PropagateEdgesResult.png})

Figure 6.312

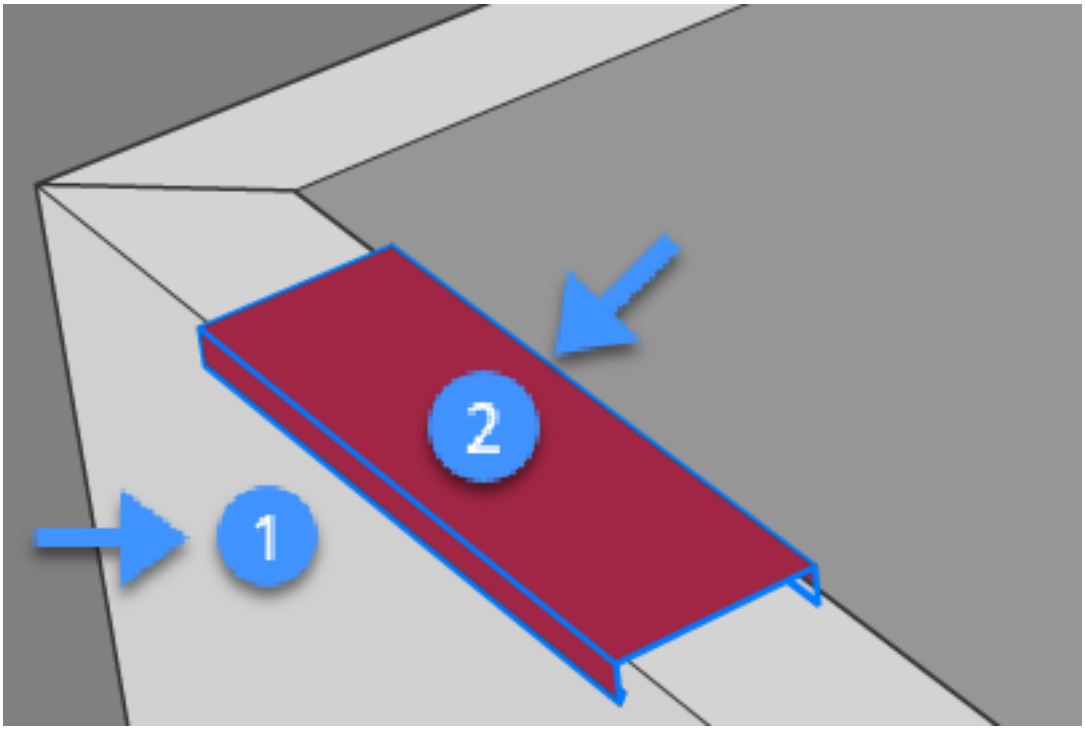


Figure 6.313

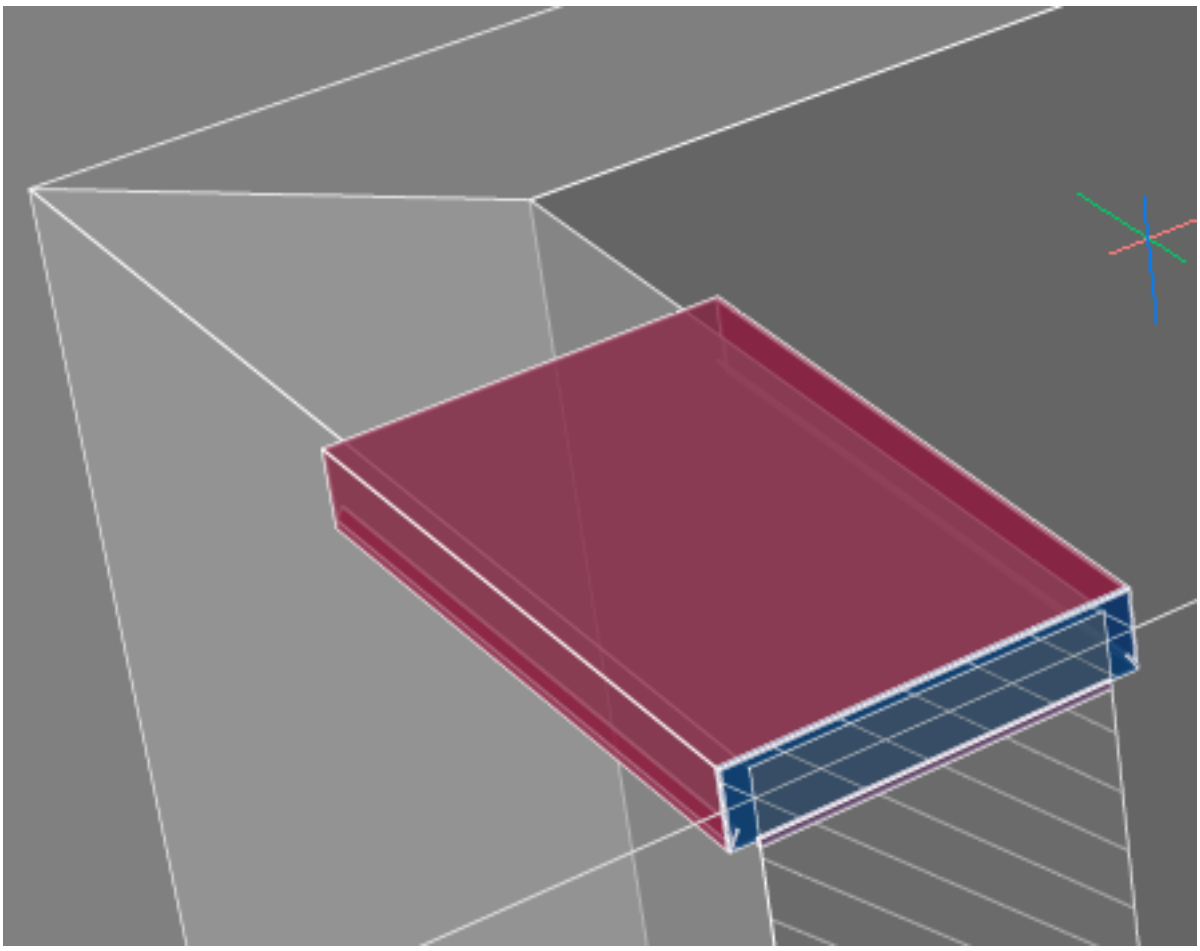


Figure 6.314

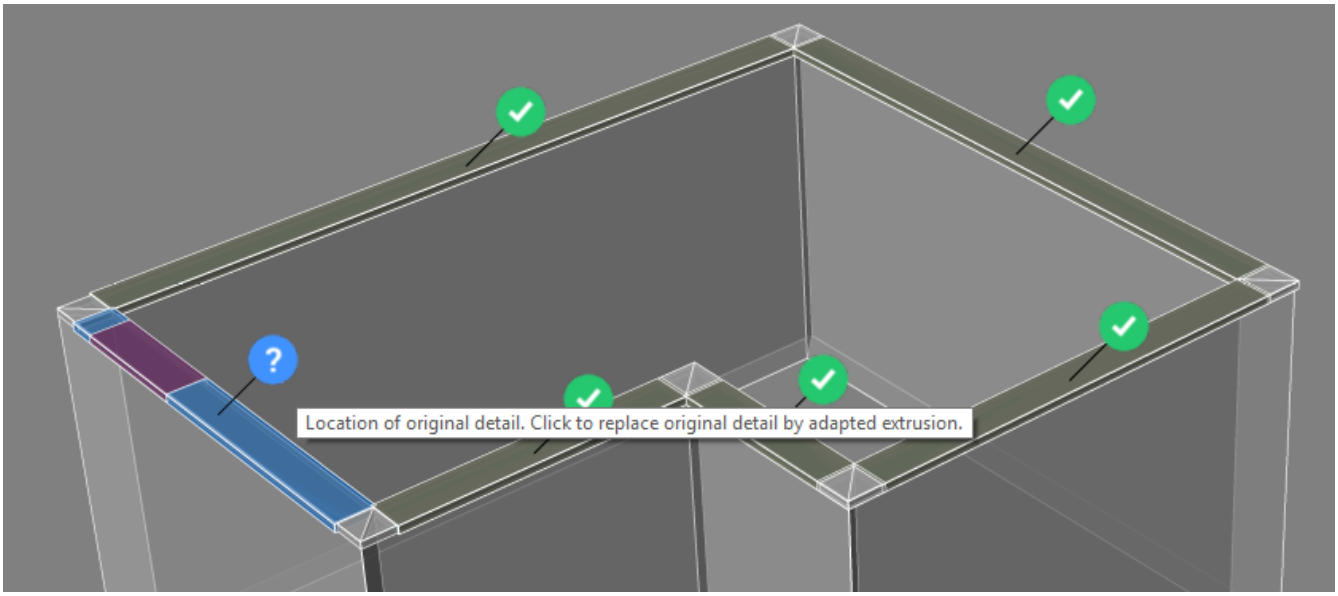
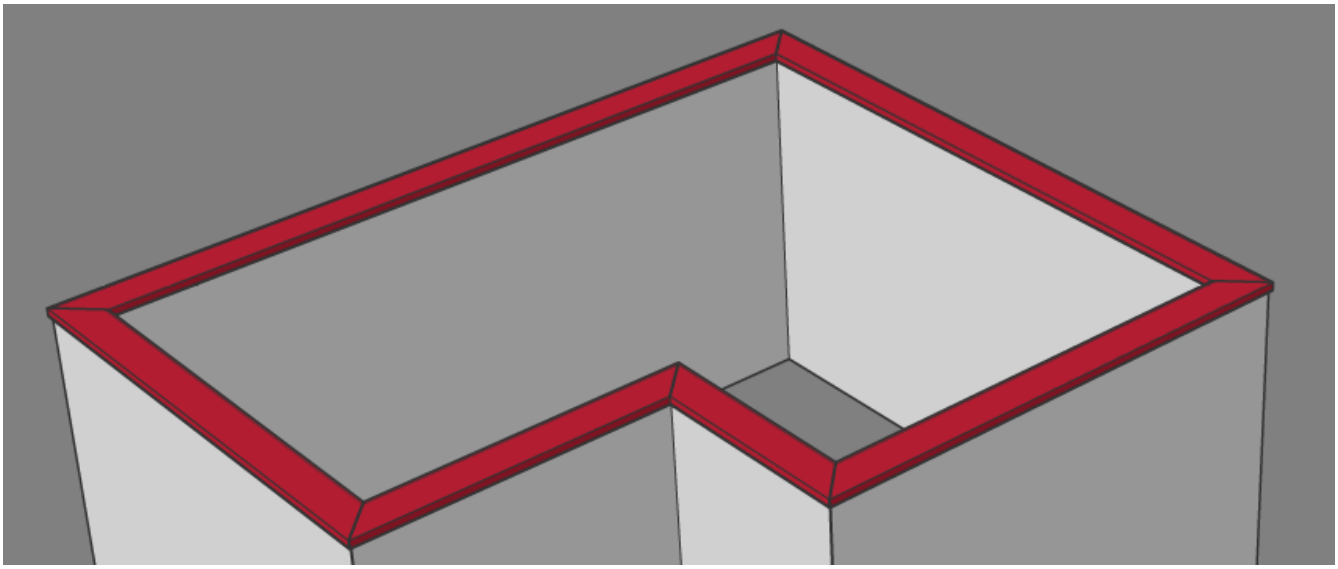


Figure 6.315



Command Options

Option	Description
Inflate first	Expands the sensitive zone of the detail.
Propagate to edges with similar orientation only	**Yes**: Only edges with similar orientation are selected. In the above example only the horizontal top face or the wall are selected. **No**: All minor faces of similar planar solids are selected. In the above example also the bottom face of the walls would be selected.
Apply All	 : Selected suggestion. Click to switch to not selected.  : Not selected suggestion. Click to switch to selected.  : Location of the original detail. Click to replace original detail by adapted extrusion.

Related Commands

BimPropagate (6.7.20)-Maps details (solids, holes, finishing geometry, ...) related to selected base solids to similar base solids in the model and optionally outlines the detail in a grid. BimPropagateCorner (6.7.21) -Propagates details connected to three planar base solids (i.e., corners). BimPropagateLinear (6.7.23) -Propagates connections between linear elements (beams, columns, pipes, ducts and their connections to walls and slabs). BimPropagatePattern (6.7.24) -Propagates a single element on a flat surface to multiple locations and grids; can be used to propagate lights, light switches, windows, air diffusers, etc. BimPropagatePlanar (6.7.25) -Propagates connections between planar elements (walls/slabs/roofs).

Figure 6.316



6.7.23 BimPropagateLinear

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Propagates connections between linear elements (beams, columns, pipes, ducts and their connections to walls and slabs).

Tutorial: How to use Propagate Linear Click here (2.7.4) to watch

Accessing the Command


Keyboard:**bimpropagatelinear** Menu:**BIM | Propagate | Propagate Linear** Toolbar:**BIM Propagate** Ribbon Tab | Panel: **Model | Propagate : BimPropagateLinear Prompts you:** Select linear or planar solids to connect. At least one should be linear: **(Select a solid.)** Entities in set: 1 Select linear or planar solids to connect. At least one should be linear: **(Select a solid or press Enter.)** Entities in set: 2 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V Select linear or planar solids to connect. At least one should be linear: **(Select a solid or press Enter.)** Select detail objects (solids, block refs or faces) to be included in the connection (optional): **(Optionally, select detail objects to be included in the propagate process.)** Entities in set: 11 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V Select detail objects (solids, block refs or faces) to be included in the connection (optional): **(Select more detail objects or press Enter.)** Propagate this 3D detail? [as Block/as Copy/No/Inflate first] <As block>: **(Press Enter or choose an option.)** ===ERROR WITH DOWNLOADED IMAGE=== Searching for similar connections Creating the suggestions Showing 18 application details Resolving conflicts and extensions Duration start BIMPROPAGATE: 1.662000 Showing 17 visible suggestions of total 17. Choose next action [Apply all/Cancel] <Apply all>: **(Press Enter to apply the connection to all selected suggestions .)** ===ERROR WITH DOWNLOADED IMAGE===

Figure 6.317

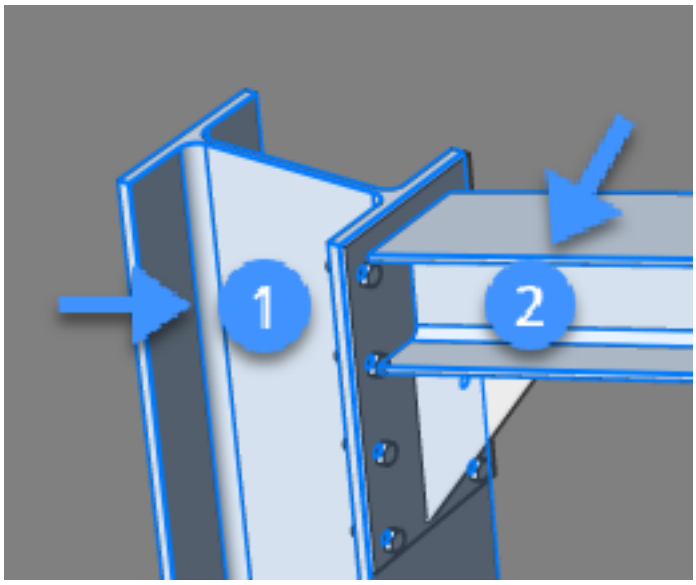
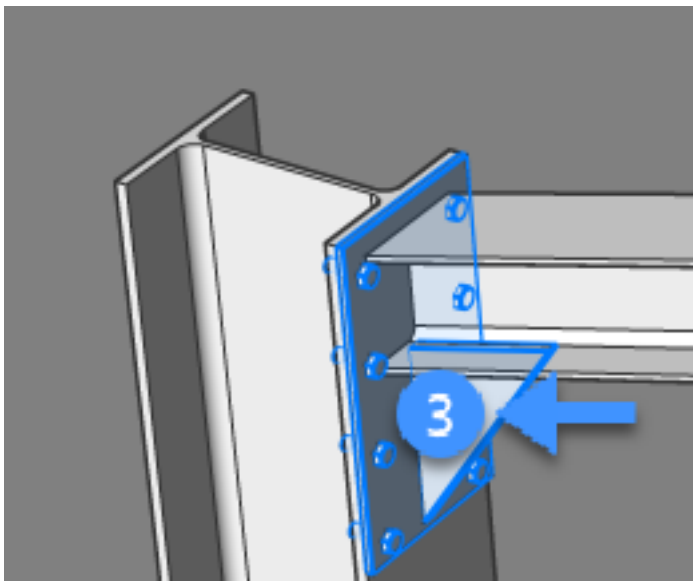


Figure 6.318



Command Options

Option	Description
As Block	Propagates the selected detail objects as a block.
As Copy	Propagates copies of the selected detail objects.
Inflate first	Expands the sensitive zone to include similar solids which are not connected yet in the model. Prompts you: **Enter inflation distance:(Type a value.)** ===ERROR WITH DOWNLOADED IMAGE===
Apply All	: Selected suggestion. Click to switch to not selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Propag A black triangel next to the Selected icon indicates alternatives are available. Hover over the icon to display a list of alternatives, then click an option. : Not selected suggestion. Click to switch to selected.

Related Commands

BimPropagate (6.7.20)-Maps details (solids, holes, finishing geometry, ...) related to selected base solids to similar base solids in the model and optionally outlines the detail in a grid. BimPropagateCorner (6.7.21) -Propagates details connected to three planar base solids (i.e., corners). BimPropagateEdges (6.7.22) -Propagates a detail along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc. BimPropagatePattern (6.7.24) -Propagates a single element on a flat surface to multiple locations and grids; can be used to propagate lights, light switches, windows, air diffusers, etc. BimPropagatePlanar (6.7.25) -Propagates connections between planar elements (walls/slabs/roofs).

Figure 6.319

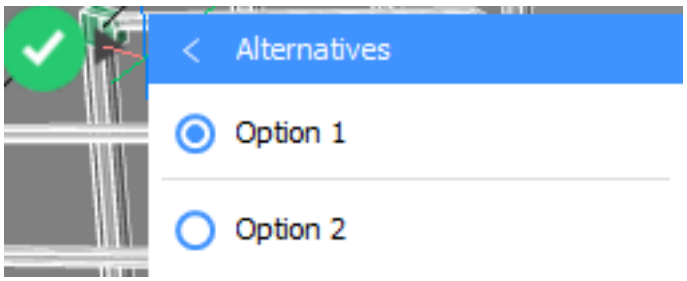


Figure 6.320



6.7.24 BimPropagatePattern

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Propagates a single element on a flat surface to multiple locations and grids; can be used to propagate lights, light switches, windows, air diffusers, etc.

Tutorial: How to use Propagate Pattern Click here (2.7.5) to watch

Accessing the Command

 Keyboard:**bimpropagatepattern** Menu:**BIM | Propagate | Propagate Pattern** Toolbar:**BIM Propagate** Ribbon Tab | Panel: **Model | Propagate : BimPropagatePattern You are prompted:** Select planar solid to which detail is related:**(Select a planar solid (1).)** Entities in set: 1 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Com Select planar solid to which detail is related:**(Select more planar solids or press Enter.)** Select detail objects to be propagated (solids or faces):**(Select detail objects (2).)** Entities in set: 5 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscon Select detail objects to be propagated (solids or faces):**(Select more detail objects or press Enter.)** Propagate this 3D detail? [Yes/No/Inflate first] <Yes>:**(Press Enter or choose an option.)** ===ERROR WITH DOWNLOADED IMAGE=== Searching for similar connections Creating the suggestions Showing 2 application details Resolving conflicts and extensions Duration start BimPropagate: 0.524000 Showing 1 visible suggestions of total 1. Choose next action [Apply all/Cancel] <Apply all>:**(Press Enter or choose an option.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/PropagatePatternOptions.png}) Applying all xx visible suggestions

Figure 6.321

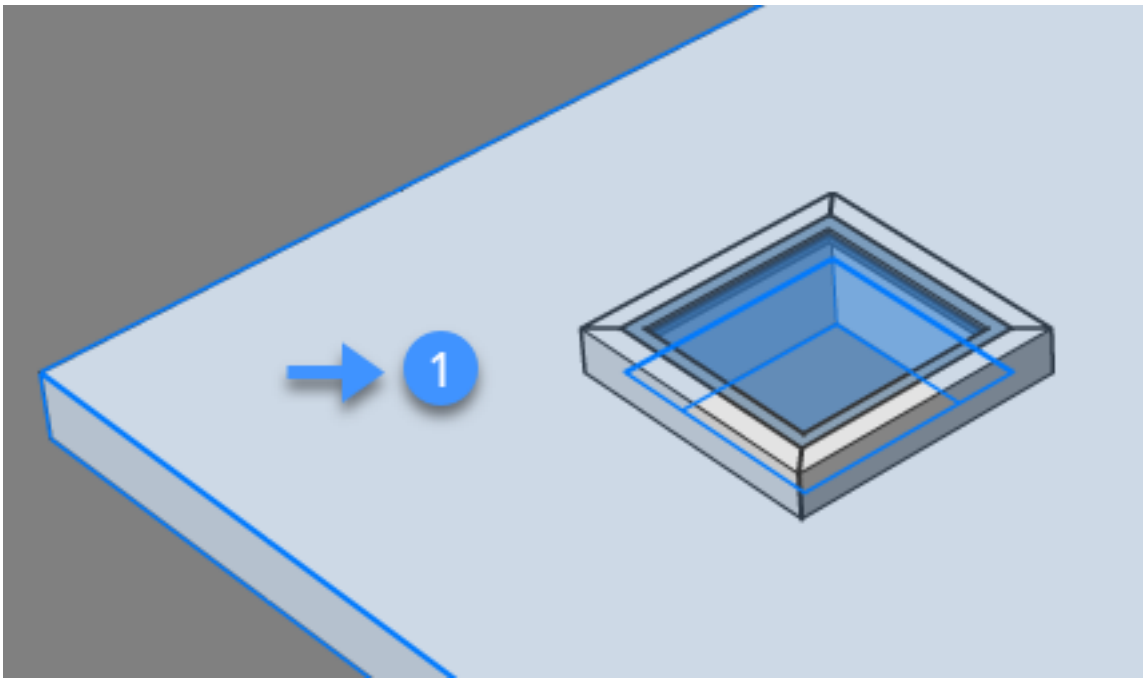


Figure 6.322

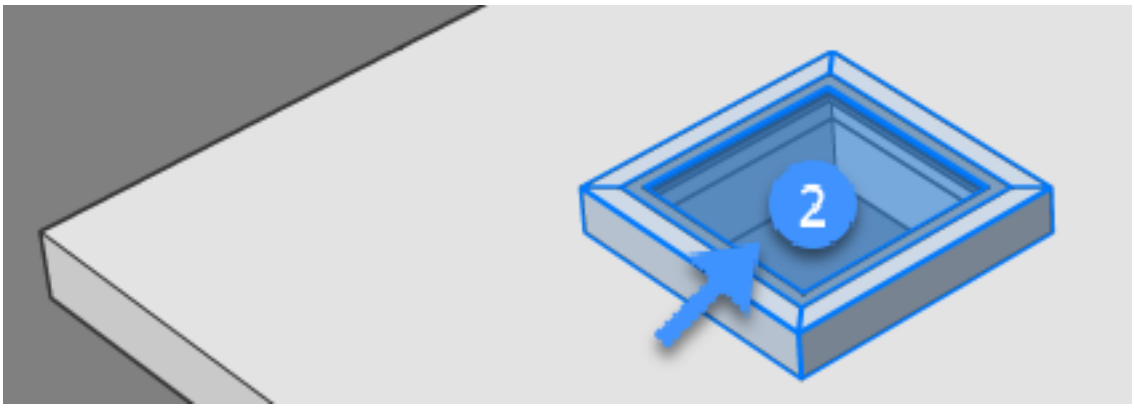


Figure 6.323

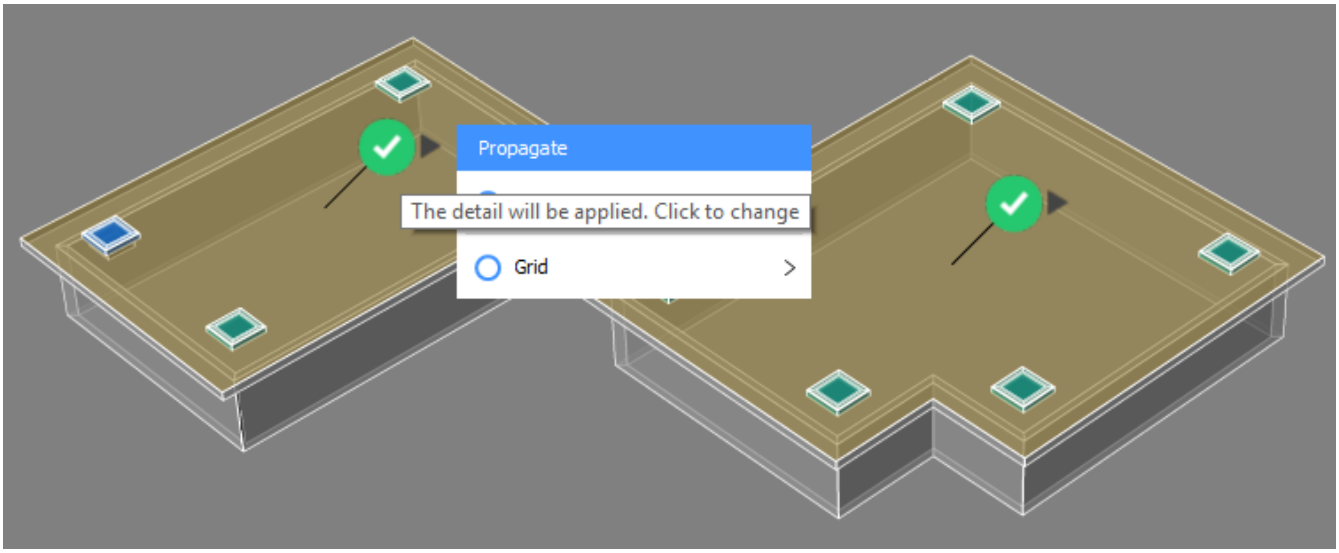
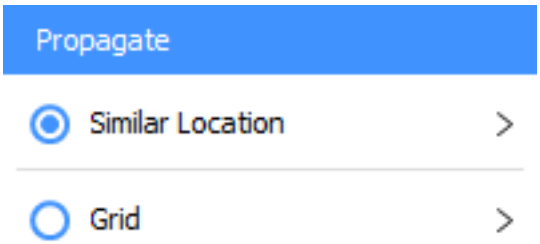


Figure 6.324



Command Options

Option	Description
Inflate First	Expands the sensitive zone of the detail.
Apply All	: Selected solid. Click to switch to not selected. : Not selected solid. Click to switch to selected.
Similar Location	Propagates the detail to similar locations on selected solids. Choose an option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ • **All**: Propagates the detail to all suggested locations on all selected solids. • **Explode**: Allows to keep/remove individual locations on a selected solid.
Grid	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ Propagates a grid of the detail on selected solids: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ • **Add/Remove Column/Grid**: Allows to define the number of columns and rows. • **Select Bottom/Top Row**: Switches the position of a single row or keeps the row or the original detail only. • **Select Left/Right Column**: Switches the position of a single column or keeps the column of the original detail only. • **Explode**: Explodes the grid; allows to keep/remove individual members of the grid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Propag

Related Commands

BimPropagate (6.7.20)-Maps details (solids, holes, finishing geometry, ...) related to selected base solids to similar base solids in the model and optionally outlines the detail in a grid. BimPropagateCorner (6.7.21) -Propagates details connected to three planar base solids (i.e., corners). BimPropagateEdges (6.7.22) -Propagates a detail along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc. BimPropagateLinear (6.7.23) -Propagates connections between linear elements (beams, columns, pipes, ducts and their connections to walls and slabs). BimPropagatePlanar (6.7.25) -Propagates connections between planar elements (walls/slabs/roofs).

Figure 6.325

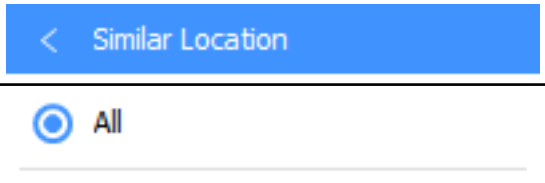


Figure 6.326

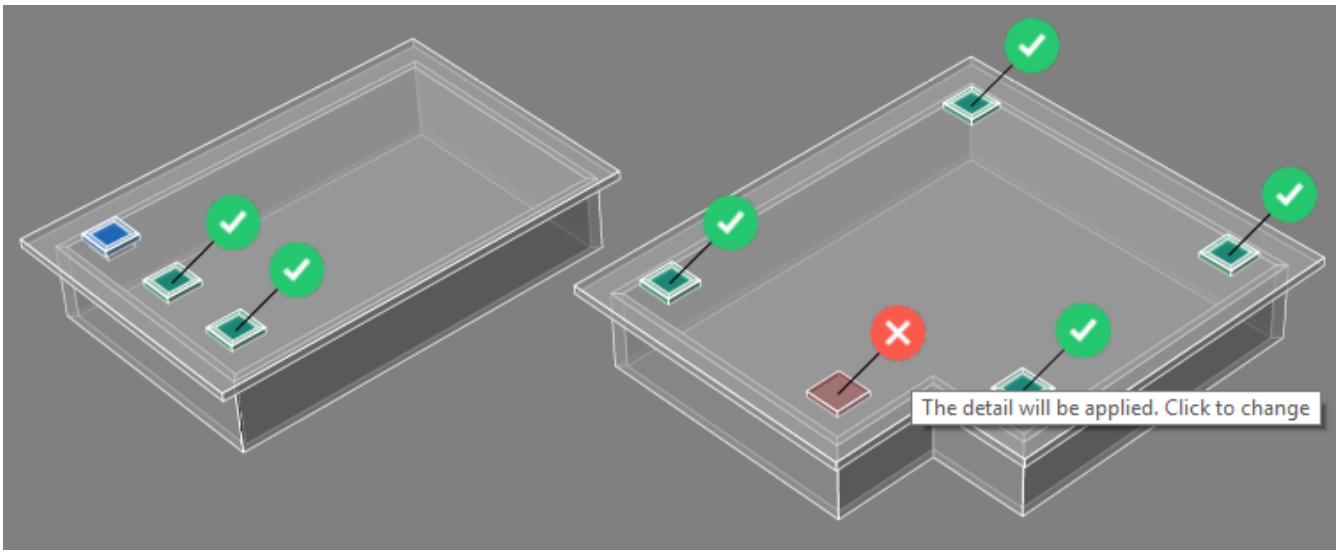


Figure 6.327

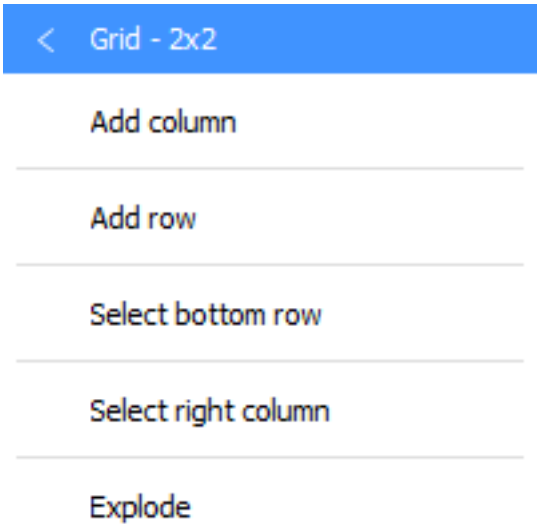


Figure 6.328

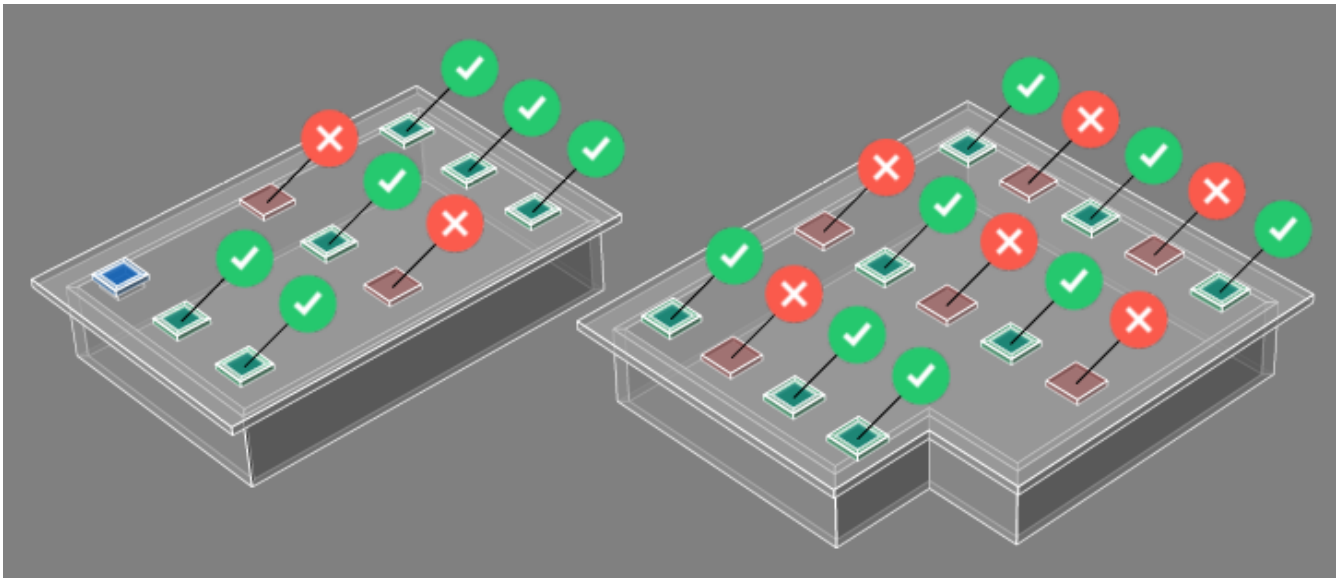


Figure 6.329



6.7.25 BimPropagatePlanar

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Propagates connections between planar elements (walls/slabs/roofs).

Tutorial: How to use Propagate Planer Click here (2.7.6) to watch

Accessing the Command


Keyboard:**bimpropagateplanar** Menu:**BIM | Propagate | Propagate Planar** Toolbar:**BIM Propagate** Ribbon Tab | Panel: **Model | Propagate : BimPropagatePlanar** See BimPropagate (6.7.20).

Related Commands

BimPropagate (6.7.20)-Maps details (solids, holes, finishing geometry, ...) related to selected base solids to similar base solids in the model and optionally outlines the detail in a grid. BimPropagateCorner (6.7.21) -Propagates details connected to three planar base solids (i.e., corners). BimPropagateEdges (6.7.22) -Propagates a detail along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc. BimPropagateLinear (6.7.23) -Propagates connections between linear elements (beams, columns, pipes, ducts and their connections to walls and slabs). BimPropagatePattern (6.7.24) -Propagates a single element on a flat surface to multiple locations and grids; can be used to propagate lights, light switches, windows, air diffusers, etc.

Figure 6.330



6.7.26 BimRoom

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Creates BIM Room entities.

Accessing the Command

 Keyboard:**bimroom** Menu:**BIM | Room** Toolbar:**BIM Rooms** Ribbon Tab | Panel: **Data | Rooms** : **BimRoom**
Prompts you in the command bar: Pick a point inside the room or <select>: **(Pick a point inside a space enclosed space or choose an option.)** A label is created at the geometric center of the room. Use the BimTag (6.7.33) command to add the room label to a plan section.

Command Options

Option	Description

Pick a point

Creates a BimRoom entity from an enclosed space. The boundary entities must be Bim classified (6.7.5). To divide a large space in different rooms, divide the space using linear entities, which you then classify as Walls. The **Room Bounding** property of the enclosing entities must be ON. The **Center Line** property controls the creation of the room boundary: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/

- **OFF**: The inner edges of the surrounding solids define the room boundary (left).
- **ON**: The center linesof the surrounding solids define the room boundary (middle).
- **Extrapolate**: If set for room bounding columns, the columns are treated as a point (right).

Prompts you: **Pick a point inside the room or <select>:(Pick a point inside the room area).** Some info displays near the cursor: Room



found: Click to create the room.
Room not found:

- No enclosed area found: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands
- Bounding entities are not correctly classified: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands
- Room already exists: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands

Dynamic UCS defines the bottom plane of the room. In this plane, an area enclosed by room bounding entities is found and a room mark is placed. The room mark is a block consisting of a hatch and attributes for name, number and area. A room stamp is created in the center of the room. You can control the layout of the room stamp by editing _RoomCallout.dwg in the **Support\Bim\Sections** subfolder of the Roamable Root folder (1.13.2). By default this is: **C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\Vx**

Select

Creates a static BimRoom entity from an existing entity or a detected boundary Prompts you: **Select 3d solid, region, polyline, or boundary**:**(Select an entity or click inside an enclosed area*.)** * The **Enable Boundary Detection** option ( ) of the SELECTIONMODES (1.13.11) system variable must be selected.

Room Properties	A Room corresponds to an IfcSpace, and has all IFC properties, apart from the BIM properties. Room properties can be edited in the Properties Panel (6.24.49): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ • **Building**: Click the field, then select a building (see BimSpatialLocations (6.7.31)). • **Story**: Click the field, then select a story (see BimSpatialLocations (6.7.31)). • **Wall Finish**: Shows the material of the surrounding walls if a composition is applied. When the room is defined by more the 4 solids, the Wall Finish properties do not display. • **Room representation**: The options are – **None**: Room stamp only. – **Hatch**: Room stamp and a solid hatch. – **Solid**: 3D solid. The height of the solid is copied from the surrounding wall solids. • **Interior Elevations**: Controls the display of **Interior Elevation** sections. If interior elevations are not created yet for the room the property is disabled. (see BimSection (6.7.27)). or using the EAttEdit (6.13.1) command: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/

Related Commands

Bimify (6.7.13) - analyzes the model and runs automatic classification and spatial location assignment for the entire model. Creates new buildings and/or stories and rooms if necessary. BimUpdateRoom (6.7.34) - updates the geometry of a Room entity. BimTag (6.7.33) - creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model.

Figure 6.331

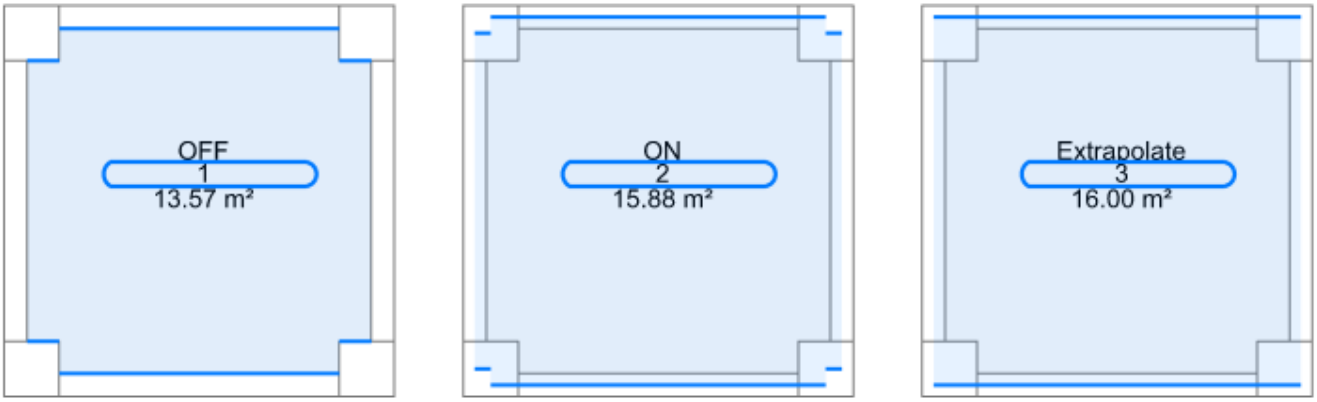


Figure 6.332

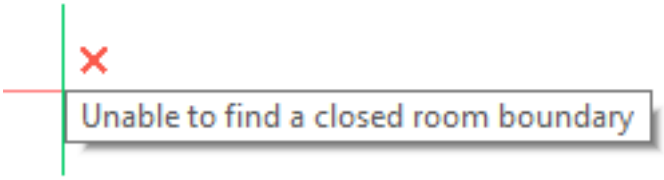


Figure 6.333

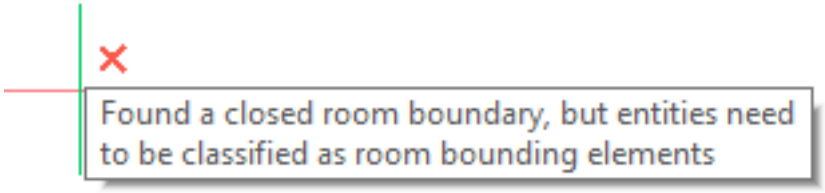


Figure 6.334

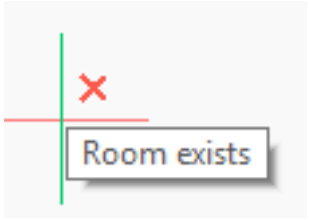


Figure 6.335

Attributes	
Room Name	Office
Room Number	1.003
Room Area	11.19 m²
BIM	
Type	Room
Name	Office
Description	
Building	Sky Office & Housing
Story	Ground Floor
GUID	3bXiwHzobCcgLVkAk4zMCd
Number	1.003
Department	
Wall north finish	Facing Bricks, Hand-formed
Wall east finish	Gypsum Board
Wall west finish	Gypsum Board
Wall south finish	Gypsum Board
Room representation	Solid
Interior Elevations	Off

Figure 6.336

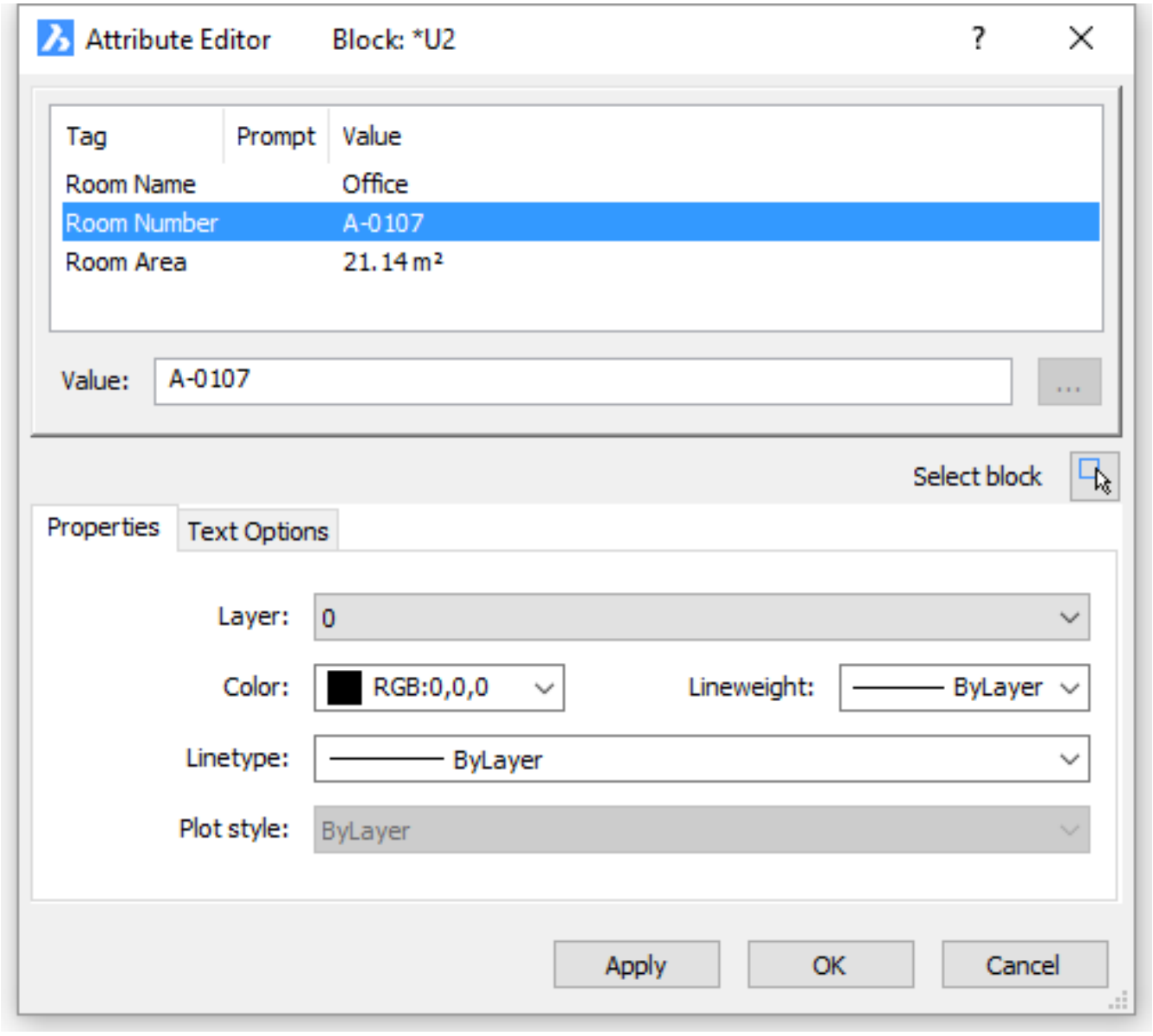

Attribute Editor
Block: *U2
Tag
Prompt
Value
Room Name
Office
Room Number
A-0107
Room Area
21.14 m²
Value:
A-0107
Select block
Properties
Text Options
Layer:
0
Color:
RGB:0,0,0
Lineweight:
ByLayer
Linetype:
ByLayer
Plot style:
ByLayer
Apply
OK
Cancel

Figure 6.337



6.7.27 BimSection

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Creates a BIM section entity.

Accessing the Command

 command bar: **bimsection** menu bar: **BIM | Define Section** toolbar: **BIM |** **or** (Detail Section) : **BimSection** The Section Tracker is attached to the cursor. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Cor The section plane (1) lies in the XY-plane of the current coordinate system (WCS or UCS). The view direction (2) is in the negative Z-direction of the current coordinate system. If Dynamic UCS (UCSDETECT (1.13.11)) is ON, the section plane aligns to the face of a 3D solid under the cursor. **Prompts you in the command bar:** Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]: **(Specify a point.)** The section plane displays dynamically parallel to UCS XY-plane, through the cursor position. Specify distance: **(Type a distance* or specify a point.)** * It is recommended to have Dynamic Dimensions (1.15.9) (DYN) active, which allows to type the distance in the dynamic entry field. The section plane is defined through the point or at the specified offset from the first point. The section line is parallel to the X-axis of the UCS or dynamic UCS and through the point at the specified offset from the first point. Only the section line and the callouts of a BIM Section entity display. When highlighted or selected the section plane, section boundary and/or section volume display. The **Clip Display** property of the BIM Section entity is enabled by default. Double click the section line or a callout of a section to toggle **Clip Display** on/off. Multiple sections can have **Clip Display** on simultaneously. The **Clip Display** property can be saved in a model view (See the View (6.31.1) command). BIM section entities are created on the current layer. It is recommended to create a dedicated layer for BIM sections. The **BIM_Sections** layer is created automatically along the first BIM section entity. Horizontal sections get the **Plan** section type, other sections get the **Section** section type, unless no geometry is cut by the section plane, then the section type is **Elevation** . The section type can be edited in the **Section Type** field on the **Properties Bar** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimSectionType.png})

NOTE	The GENERATEASSOCVIEWS (1.13.11) system variable controls whether VIEWSECTION produces 2D drawings which are persistently associated with the source 3D model. If GENERATEASSOCIVIEWS is ON, associative dimensions are updated automatically when the 3D model is modified and BIMSECTIONUPDATE (6.7.29) is executed.

Figure 6.338

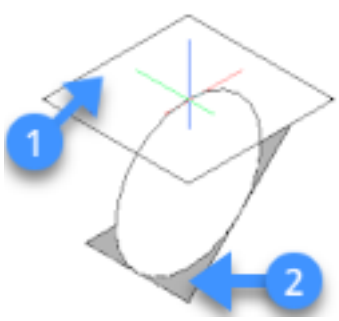
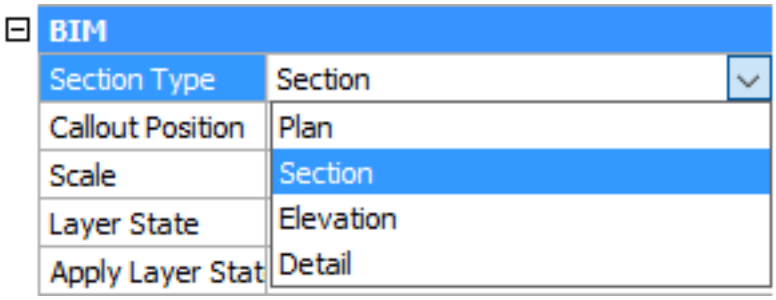


Figure 6.339



Command Options

Option	Description
Scale	Sets the **Scale** property of the viewport in the drawing file created by the BIMSECTIONUPDATE (6.7.29) command. The default **Scale** is saved through the **SectionScale** user preference. Default = 0.02. Prompts you: **Specify a scale for the viewport with the section result (0.000001 - 1000000.000000) <0.02>: (Do one of the following:)** • Press Enter to accept the default scale. The default scale is saved through the SectionScale (1.13.11) user preference. • Type a value, then press Enter. The value is saved to the SectionScale (1.13.11) user preference.
Detail	Creates a **Detail** section type. Prompts you: **Select first corner of the volume box base rectangle or [Based on an existing section/Scale]: (Specify a point, or choose an option.) Select the oppersite corner of volume box base rectangle: (Do one of the following)**: • Specify a point. • Use the dynamic dimension fields to define the base rectangle. Hit the TAB key to switch between the dynamic dimension fields. **Select the height of volume box: (Do one of the following)**: • Specify a point. • Type a value in the dynamic dimension field.

Based on an existing section	Creates a **Detail** section type based on an existing section. The base rectangle of the detail section will be parallel to the section plane of the selected section. Prompts you: **Select an existing section plane as base - (Select a Bim section). Select first corner of the volume box base rectangle or [Based on an existing section/Scale]** - **(Specify a point, or choose an option.) Select the oppersite corner of volume box base rectangle: (Do one of the following)**: • Specify a point. • Use the dynamic dimension fields to define the base rectangle. Hit the TAB key to switch between the dynamic dimension fields. **Select the height of volume box: (Do one of the following)**: • Specify a point. • Type a value in the dynamic dimension field.
Interior	Creates interior elevations and a floor plan of a room. Prompts you: **Select BIM Rooms to generate Interior Elevations** - **(Select a room)**. Entities in set: 1 **Select BIM Rooms to generate Interior Elevations** - **(Select more rooms or press Enter.)** Interior Elevations created for 1 BIM Room(s). The **BIM / Interior Elevations** property of the selected room(s) is set to ON. Use BimSectionUpdate (6.7.29) to update the interior elevations.
Reflected ceiling	Creates the plan of a ceiling projected onto the section plane. A reflected ceiling plan shows the size and location of light fixtures and other structures on the ceiling. Prompts you: **Select a point to place section or [Detail/Interior/Scale/Reflected ceiling] - (Specify a point).** The section plane displays dynamically parallel to UCS XY-plane, through the cursor position. **Specify distance**- **(Type a distance* or specify a point)**. The **Clip display** property of a reflected ceiling section iq**Off** by default.

Procedures

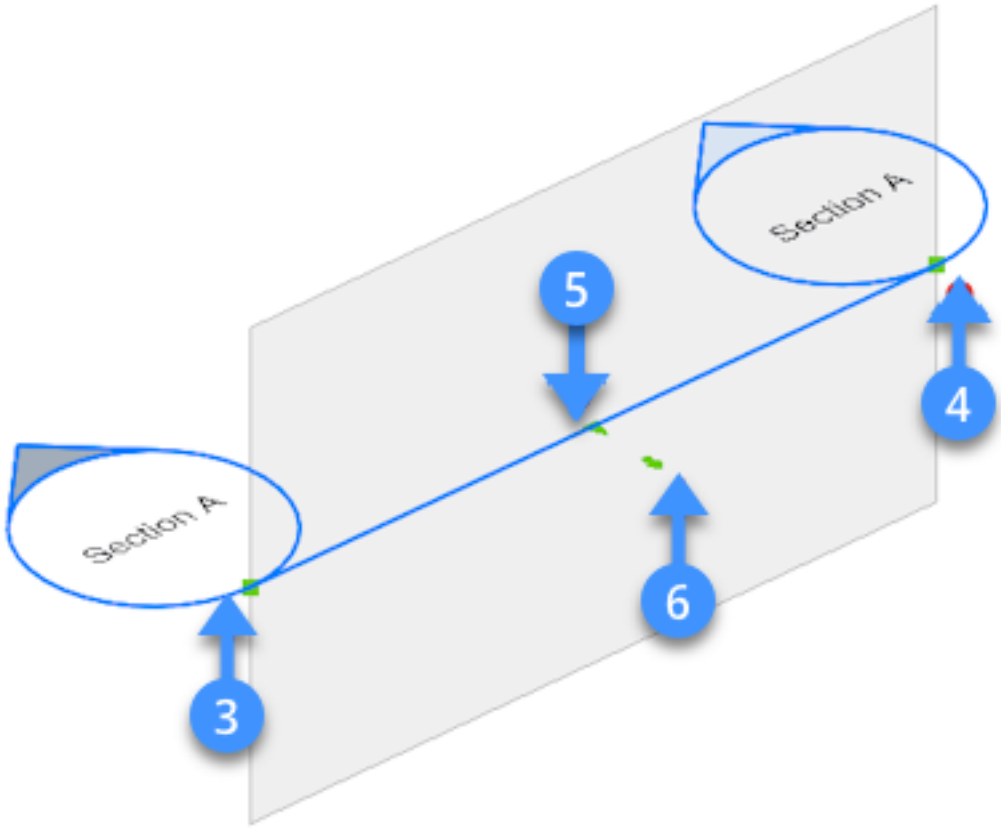
- General procedure to create a BIM section entity (1.18.4)
- Defining the BIM Section entity properties (1.18.4)
- To define a plan section (1.18.4)
- To define a cross section (1.18.4)
- To define a detail section (1.18.4)
- To create a 2D isometric drawing (1.18.4)
- To define an elevation (1.18.4)
- Turning a section plane into a BIM section entity (1.18.4)
- Creating a jogged BIM section entity (1.18.4)

Grips Editing

BIM Section entities can be edited through grips. Select the entity. A section entity in **Plane** state has 3 grips. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimSectionGripsEditingPlanea.png})

- 3: Start grip: allows to move the section and the position of the **Start** callout..
- 4: End grip: allows to modify the orientation of the section and the position of the **End** callout.

Figure 6.340



- 5: Center Grip: allows to move the section and the position of the **Mid** callout.
- 6: Click the arrow to flip the view direction.

In **Boundary** state, section entities have five more grips. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1

- 7: Midpoint grip: allows to move the section entity.
- 8: Corner grip: allows to deform the section boundary/volume.
- 9: Midpoint grip: allows to stretch the section boundary/volume.
- 10: Corner grip: allows to deform the section boundary/volume.
- 11: Midpoint grip: allows to stretch the section boundary/volume.

In **Volume** state, section entities have two more grips, compared to the boundary state.. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimSectionGripsEditingVolumea.png})

- 12: Allows to move the top plane of the section entity.
- 13: Allows to move the bottom plane of the section entity.

Figure 6.341

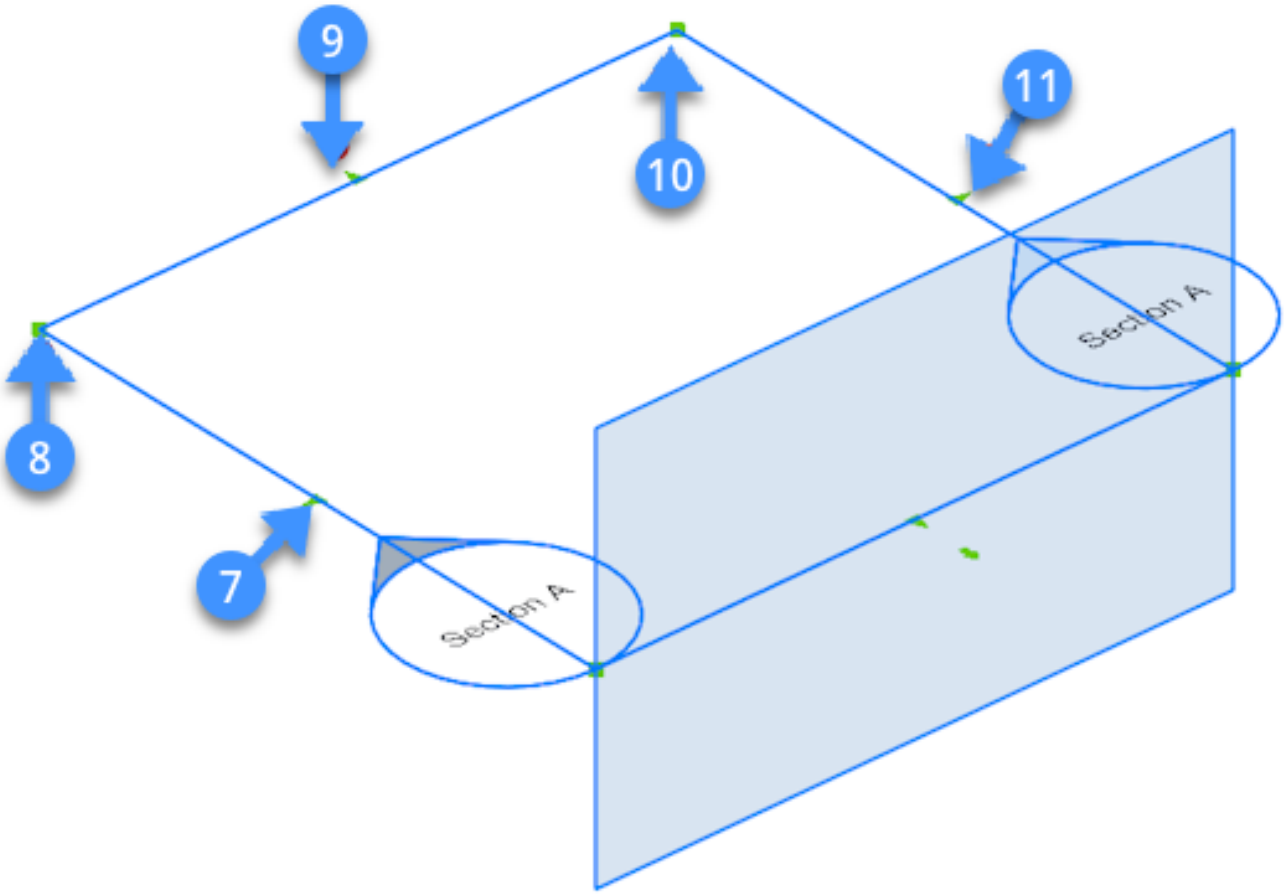
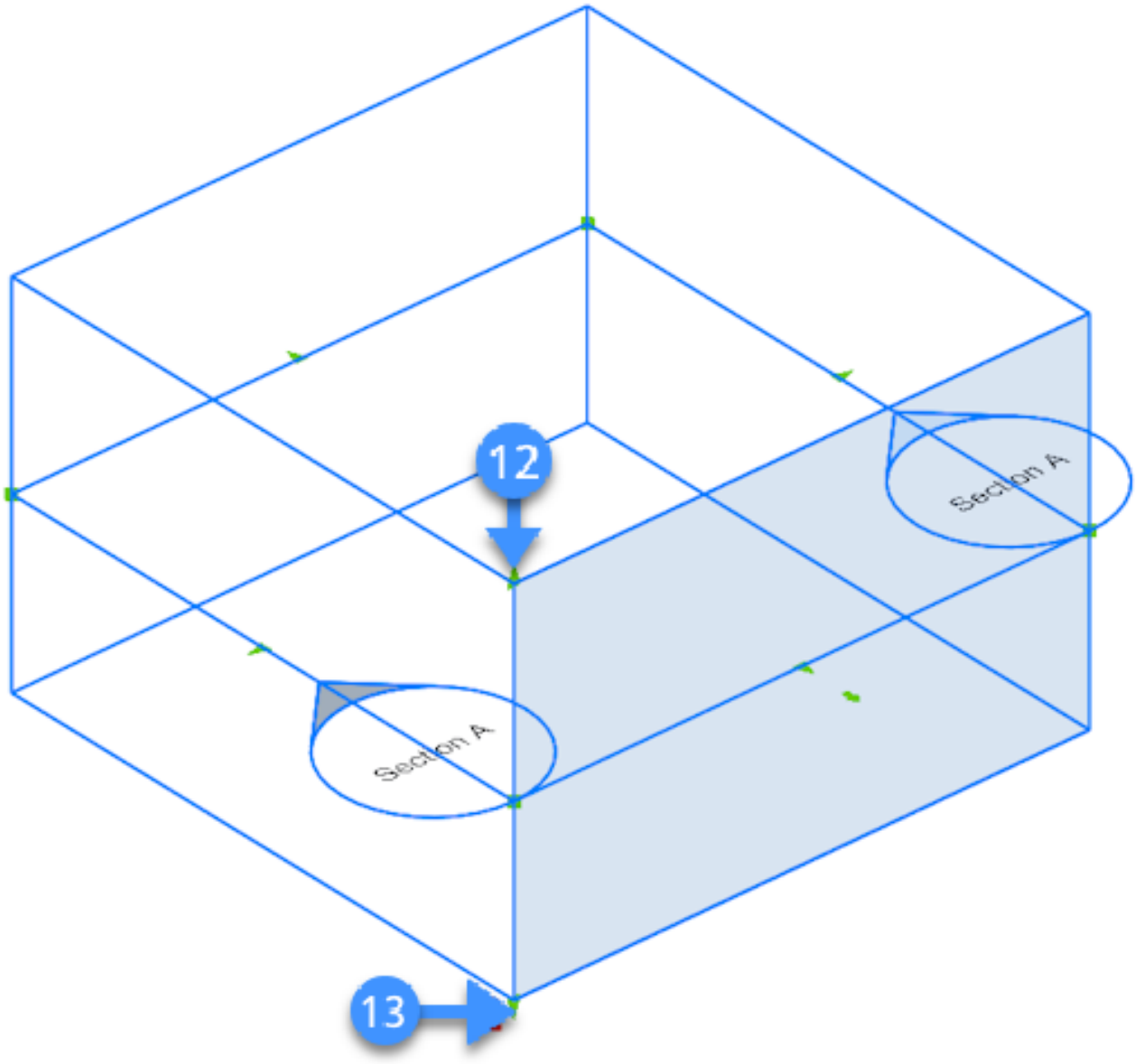


Figure 6.342



Related Commands

BimPatch (6.7.18) - allows to replace part of a generated section. BimSectionUpdate (6.7.29) - exports or updates the result of a BIM section in a drawing. ClipDisplay (6.9.24) - toggles the **Clip Display** property of a section plane or a BIM section entity. Section (6.27.11) - creates section planes of 3D solids; the results are region entities. SectionPlane (6.27.12) - creates a section entity that creates sections of 3D solids. SectionPlaneSettings (6.27.13) - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog.

Figure 6.343



6.7.28 BimSectionOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing.

Accessing the Command

 command bar: **bimsectionopen** menu bar: **BIM | Open Section Model** toolbar: **BIM |** Quad: **BIM |** : **BimSectionOpen**

To open the 3D BIM model related to a BIM section drawing **Prompts you in the command bar:** Select BIM section entity or BIM section results: **(Either select a section block in Model Space or a viewport in a Paper Space layout.)** Entities in set: 1 Select BIM section entity or BIM section results: **(Press Enter to open the 3D model.)**

To open the drawing file(s) related to a BIM section entity **Prompts you in the command bar:** Select BIM section entity or BIM section results: **(Select a BIM section entity.)** Entities in set: 1 Select BIM section entity or BIM section results: **(Select more BIM section entities, or press Enter.)**

Command Options

No options.

Procedures

- Opening a generated drawing (2.12.4)

Related Commands

BimPatch (6.7.18) - allows to replace part of a generated section. BimSection (6.7.27) - creates a BIM section entity. BimSectionUpdate (6.7.29) - exports or updates the result of a BIM section in a drawing.

Figure 6.344



6.7.29 BimSectionUpdate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Exports or updates the result of a BIM section in a drawing.

Accessing the Command

 command bar: **bimsectionupdate** menu bar: **BIM Update Section** toolbar: **BIM** |  : **BimSectionUpdate**

To create or update BIM sections when in a BIM 3D model Open the BIM 3D model. Launch the BimSectionUpdate command. **Prompts you in the command bar:** Select BIM section entity to generate or BIM section result to update: **(Select a BIM section entity or a BIM room*.)** Entities in set:1 Select BIM section entity to generate or BIM section result to update: **(Select more BIM section entities or BIM rooms, or press Enter.)** Bricsys reports: xx section(s) were generated C:\BricsCAD\Villa\Section Plane (1) ... C:\BricsCAD\Villa\Section Plane (xx) *BIM rooms can be selected if the interior elevations have been created by the Interior (6.7.27) option of the BimSection (6.7.27) command.

To update a BIM section drawing Open the BIM section drawing. Launch the BimSectionUpdate command. **Prompts you in the command bar:** Select BIM section entity to generate or BIM section result to update: **(Either select a section in Model Space or a viewport in a Paper Space layout.)** Entities in set:1 Select BIM section entity to generate or BIM section result to update: **(Either select a section in Model Space or a viewport in a Paper Space layout; or Press Enter.)**

Command Options

No Options

Procedures

- To Update a BIM Section (2.12.4)

Related Commands

BimPatch (6.7.18) - allows to replace part of a generated section. BimSection (6.7.27) - creates a BIM section entity. SectionPlane (6.27.12) - creates a section entity that creates sections of 3D solids. SectionPlaneSettings (6.27.13) - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog. SectionPlaneToBlock (6.27.14) - saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block. BimTag (6.7.33) -Creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model.

Figure 6.345



6.7.30 BimStructuralConnect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Connects structural profile solids such as beams and columns.

Accessing the Command

 Menu:**BIM | Bim Connect | Structural Connect** Toolbar:**Bim Connect** Quad Tab:**Model** Ribbon Tab | Panel: Keyboard:**bimstructuralconnect : BimStructuralConnect Prompts you:** Select solids to connect:**(Select one or more structural profile solids*.)** Entities in set: 1 Select solids to connect: **(Select more structural profile solids* or press Enter.)** *The axes of the selected solids must be coplanar. When 2 solids are selected: Select base solid [Switch] <Accept>:**(Press Enter to accept the connection or choose an option.)** Apply connection or switch to [Smooth/Planar] <Accept>:**(Press Enter to accept the connection or choose an option.)** When multiple solids are selected: Apply connection or switch to [Smooth/Planar] <Accept>:**(Press Enter to accept the connection or choose an option.)**

Command Options

Switch	Switches between a mitered connection and 2 butted connections. Prompts you: **Select base solid [Switch] <Accept>: (Type s then press Enter or tap the Ctrl-key)**. The connection between the solids changes. Press Enter to accept the selected connection**.** If the HOTKEYASSISTANT = ON, the the Hotkey Assistant indicates the currently selected L-connection type. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/HotKey
Smooth/Planar	Switches between a smooth ( ) or planar ( ) connection. Prompts you: **Apply connection or switch to [Smooth/Planar] <Accept>:** • Tap the Ctrl-key to switch between smooth and planar connection types. • Press Enter to accept the selected connection. If the HOTKEYASSISTANT = ON, the the Hotkey Assistant indicates the currently selected connection type. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Structu (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Structu Connection types: Smooth (left) or Planar (right)

Procedures

- Making a structural connection to linear solids. (2.4.2)

Related Commands

BimDrag (6.7.8) - moves 3D solids perpendicular to a selected face by dragging that face. Optionally BimFlowConnect (6.7.11) - creates a connection between flow segments, such as pipes or HVAC ducts. preserves connections to other solids. LConnect (6.20.19) - creates an L-connection between solids. TConnect (6.29.10) - connects solid faces to other solids.

Figure 6.346



Figure 6.347

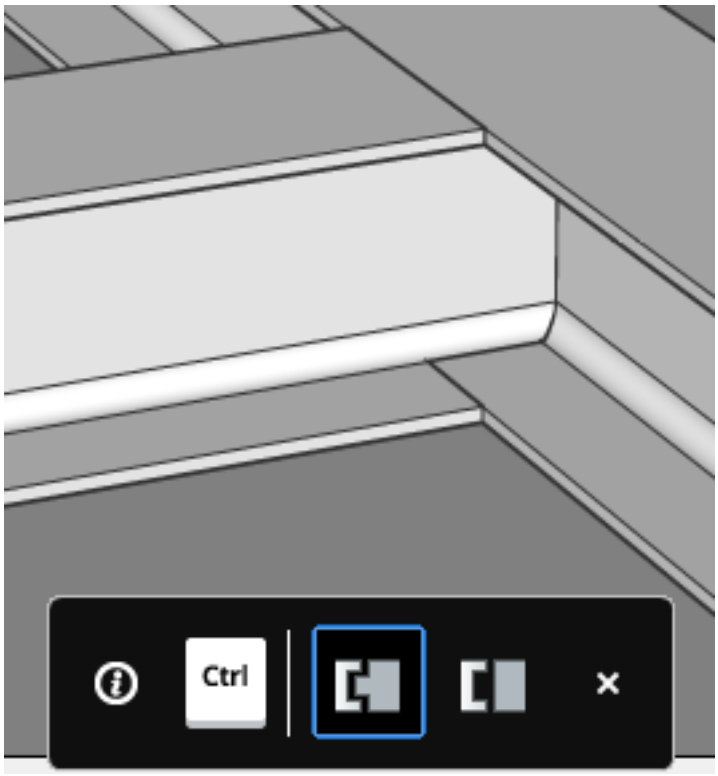


Figure 6.348

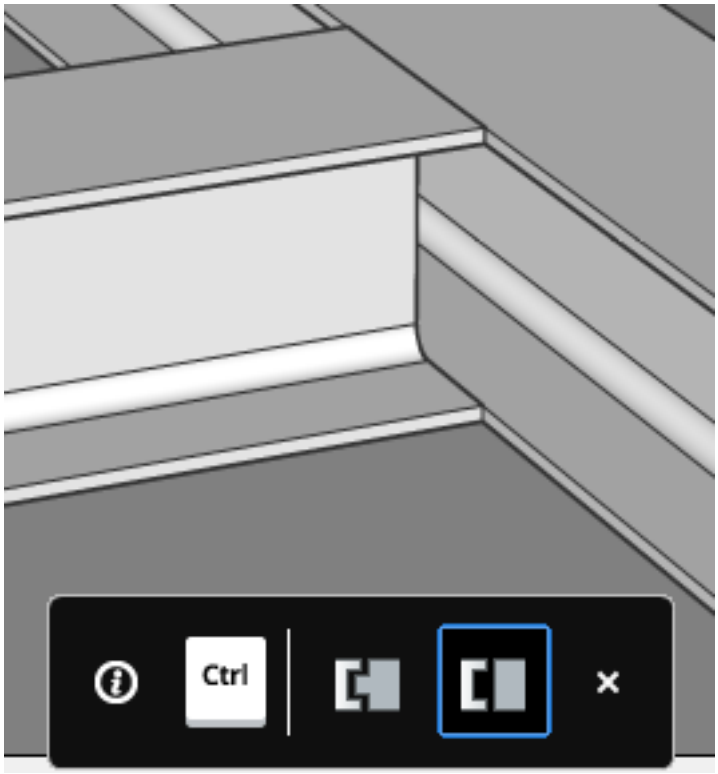


Figure 6.349



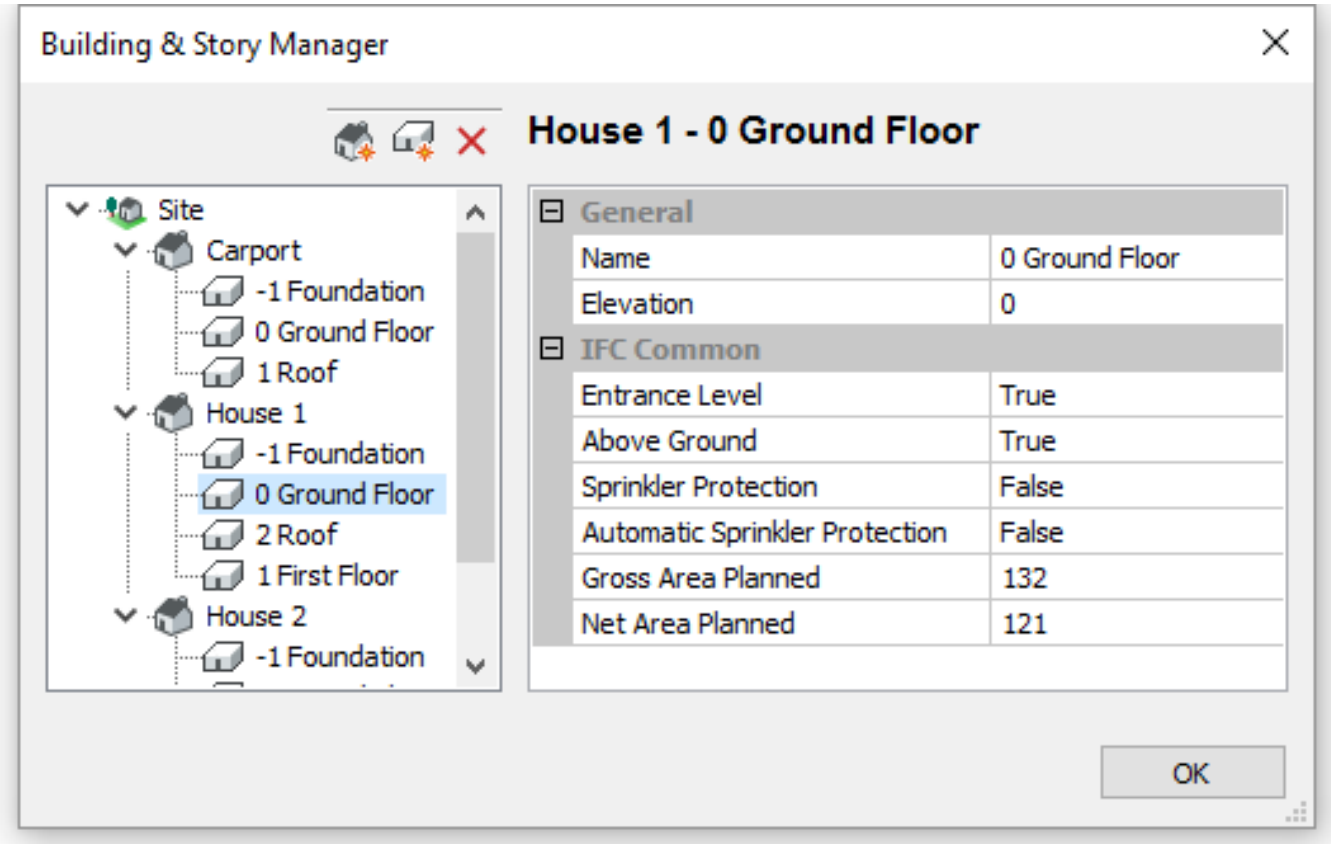
6.7.31 BimSpatialLocations

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Opens the Buildings & Stories manager. This explorer allows to create and edit site, buildings and stories.

Accessing the Command

command bar: **bimspatiallocations** menu bar: **BIM | Spatial Locations** toolbar: **BIM |** : **BimSpatialLocations** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimSpatialLocat
Click the **Close** (X) button or the **OK** button to close the dialog box.

Figure 6.350



Command Options

Icon	Option	Description
	New Building	Adds a new building to the model. Fill out the properties grid: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	New Story	Adds a new story in the selected building. Fill out the properties grid: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
	Delete	Deletes the selected building or story. An alert box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 Click the **Yes** button to delete the selected item.

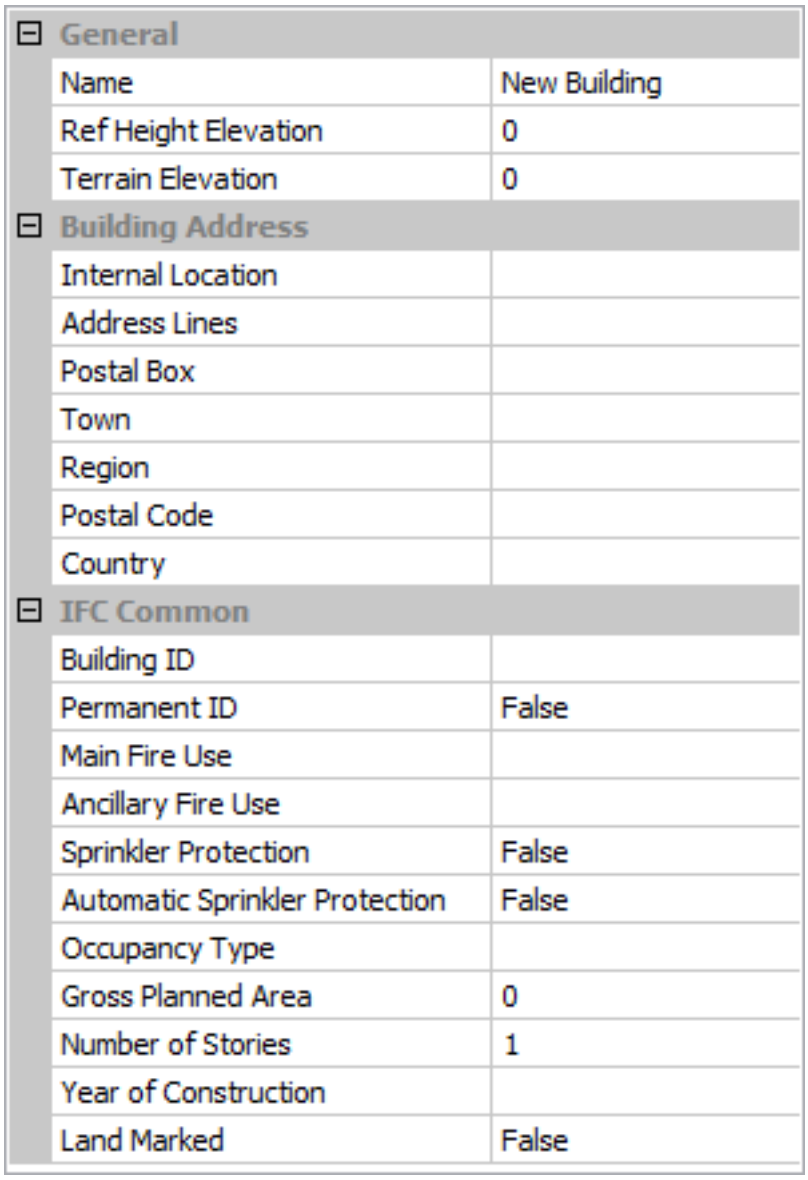
Procedures

- Assigning a building and story to a building element (2.6.11)
- Viewing the building elements in the Structure browser (2.6.11)

Related Commands

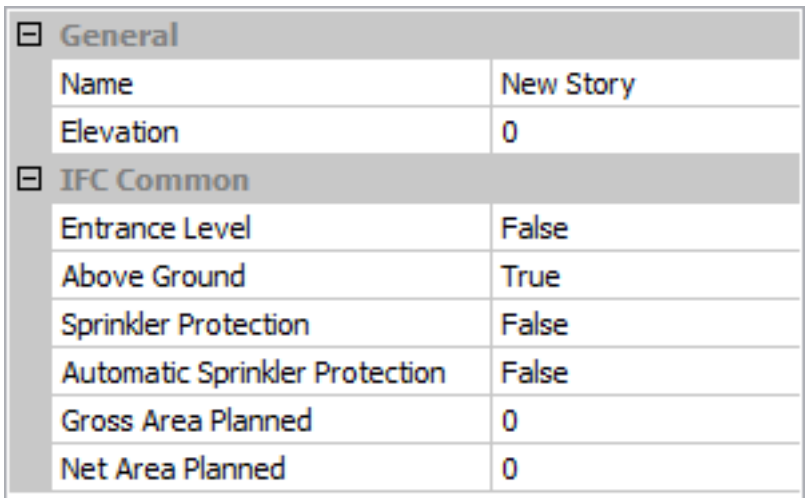
BimAttachSpatialLocation (6.7.4) - allows to attach or remove a spatial location to a selection of entities. BimClassify (6.7.5) - classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier). Bimify (6.7.13) - analyzes the model and runs automatic classification and spatial location assignment for the entire model. BimRoom (6.7.26) - creates Room entities.

Figure 6.351



General	
Name	New Building
Ref Height Elevation	0
Terrain Elevation	0
Building Address	
Internal Location	
Address Lines	
Postal Box	
Town	
Region	
Postal Code	
Country	
IFC Common	
Building ID	
Permanent ID	False
Main Fire Use	
Ancillary Fire Use	
Sprinkler Protection	False
Automatic Sprinkler Protection	False
Occupancy Type	
Gross Planned Area	0
Number of Stories	1
Year of Construction	
Land Marked	False

Figure 6.352



General	
Name	New Story
Elevation	0
IFC Common	
Entrance Level	False
Above Ground	True
Sprinkler Protection	False
Automatic Sprinkler Protection	False
Gross Area Planned	0
Net Area Planned	0

Figure 6.353

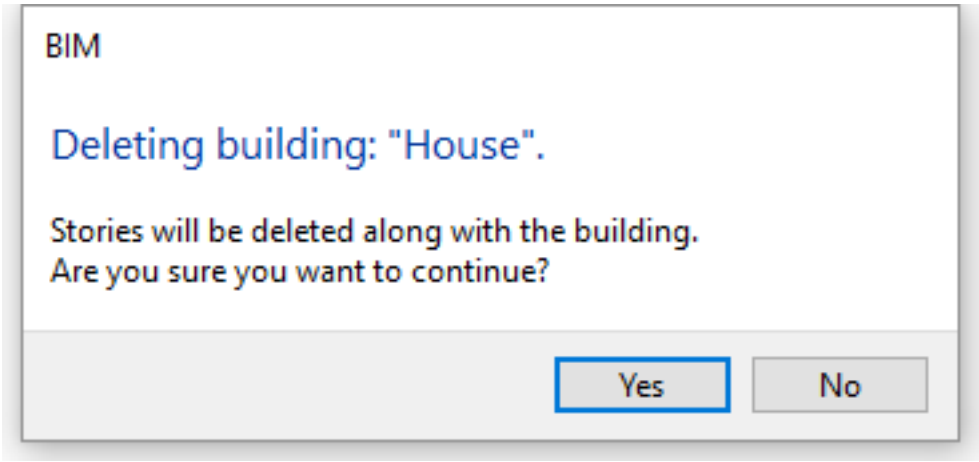

BIM
Deleting building: "House".
Stories will be deleted along with the building.
Are you sure you want to continue?
Yes
No

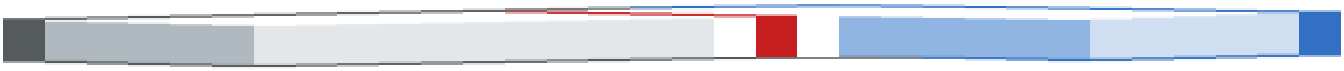
Figure 6.354



6.7.32 BimSplit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Automatically splits segmented solids into separated solids and allows to split a solid using a selection of cutting faces.

Accessing the Command



command bar: **bimsplit** menu bar: **Bim | Split** toolbar: **Bim** : **BimSplit Prompts you in the command bar:** Select solid for auto splitting or cutting faces for manual splitting: **(Select a 3D solid* or one or more faces**.)** Entities in set: 1 Select solid for auto splitting or cutting faces for manual splitting: **(Select more solids or faces or press Enter.)** Entity was split in 6 parts (*) To select a solid, make sure that the value of SelectionPreview is 2 or 3; hover over the solid, then hit the TAB key until the whole solid highlights and click. (**) To select a face, place the cursor over a face and click. Faces highlight one by one, regardless of the value of SelectionPreview or SelectionModes.

Note	Inserts, such as windows and doors, which existed in the split solid are removed.

Command Options

Select solid:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Command	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/
Solid selected ...	... split into 6 separate solids

Select cutting faces:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Command	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/
Two cutting faces selected ...	... split into 4 separate solids

Procedures

To split a solid automatically (2.4.16) To split a solid using cutting faces (2.4.16)

Related Commands

BimConnect (6.20.19) - creates an L-connection between minor faces of two solids. Union (6.30.6) - performs Boolean union operations on 3D solids and 2D regions; adds one to another to form a single entity. Slice (6.27.36) - cuts a 3D solid with a plane into two parts. SolidEdit (6.27.39) - edits 3D solids and 2D regions.

Figure 6.355

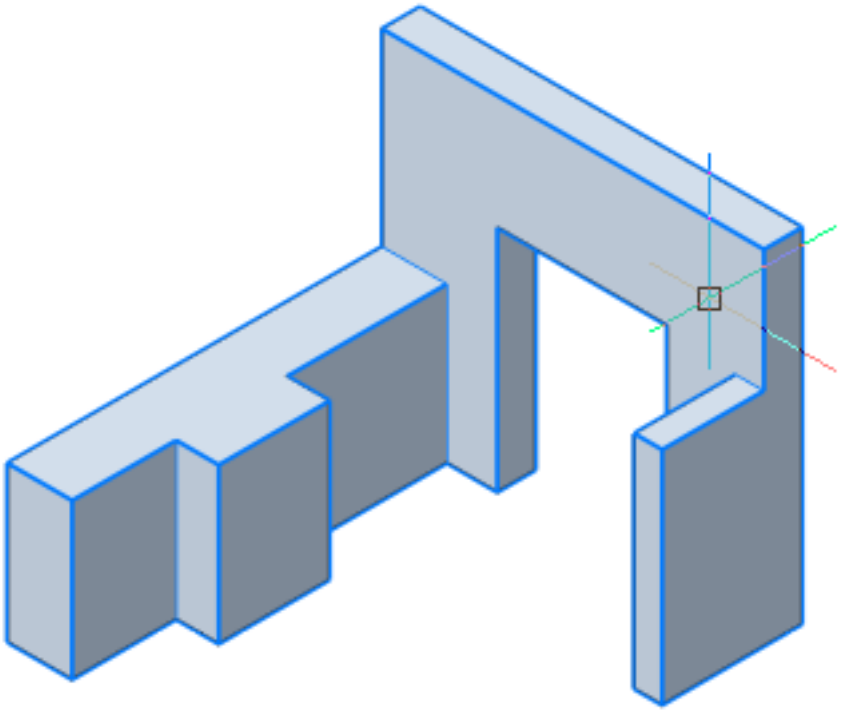


Figure 6.356

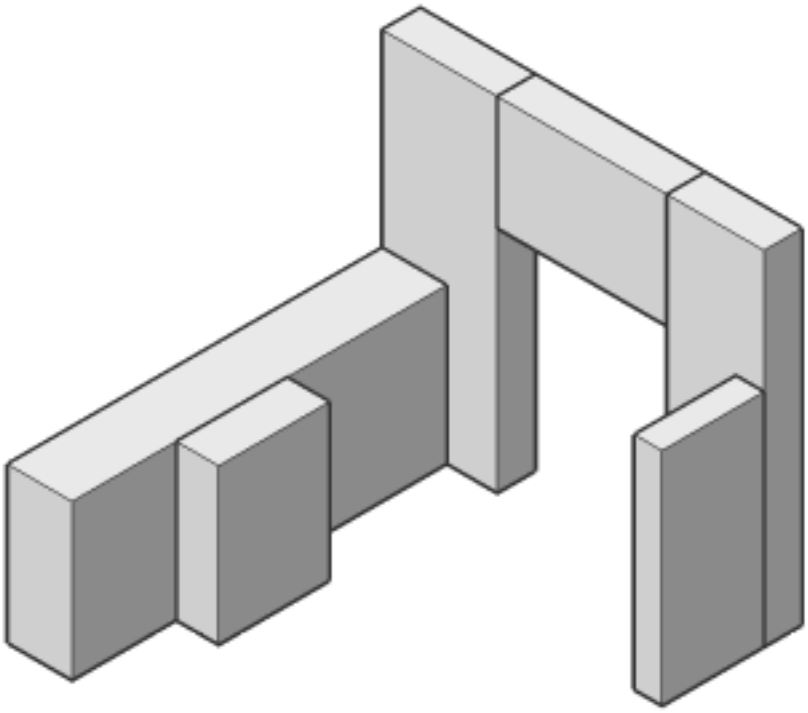


Figure 6.357

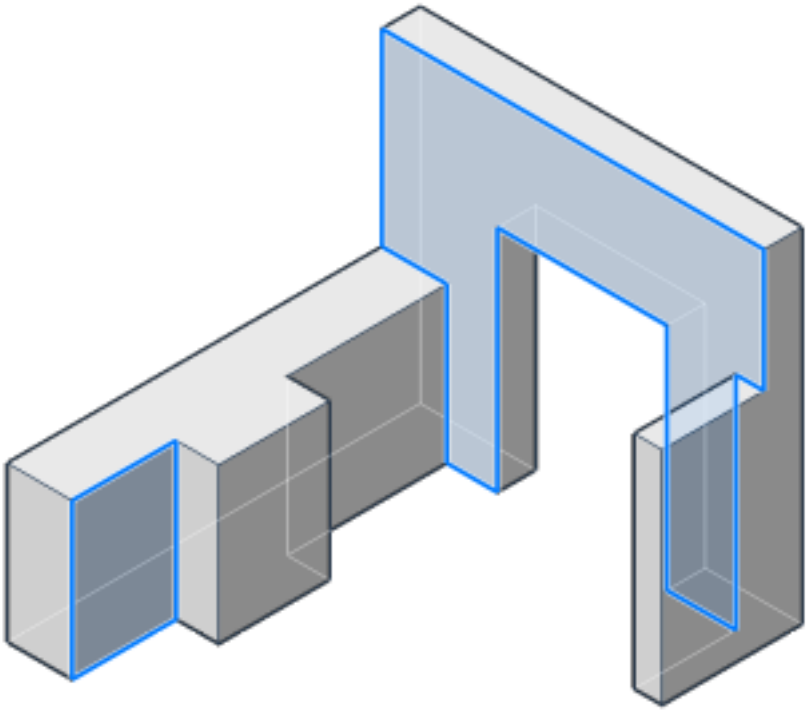


Figure 6.358

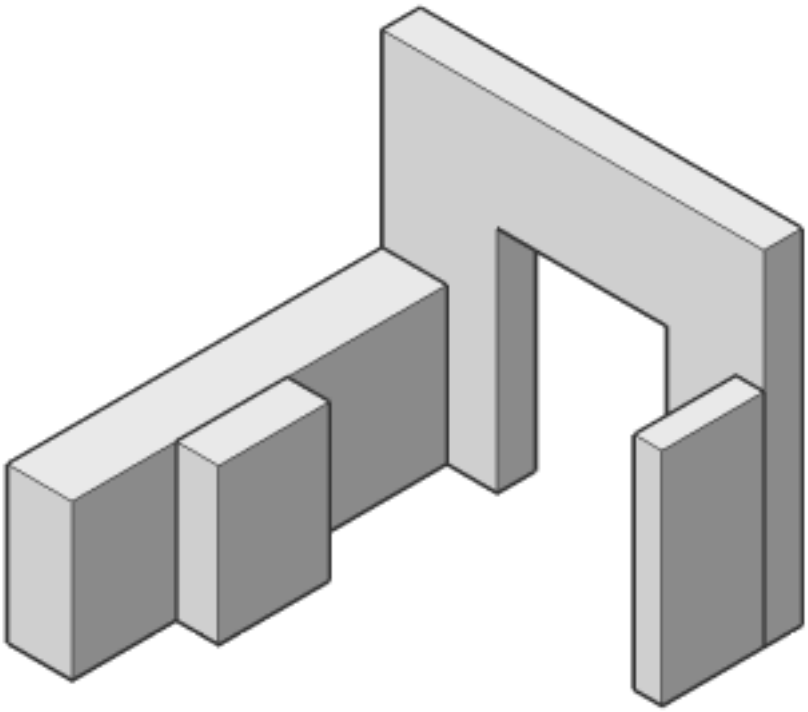


Figure 6.359



6.7.33 BimTag

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model. The command can be used on section viewports only. The GENERATEASSOCVIEWS (1.13.11) system variable must be ON when generating or updating BIM tags.

Accessing the Command

A M command bar: **bimtag** ribbon: **Annotate | Tag** : **BimTag Prompts you in the command bar:** Select BIM section viewport [Manual]: **(Click the border of a Bim Section viewport.)** Tags are created automatically for all sectioned building entities in the selected viewport. The tags are created as multileader (6.21.20) entities using multileader styles in **_Section Tag.dwg**. The content of such multileader style is a block with attributes. Each attribute refers to a property of a BIM entity type. BIM entity types are linked to a multileader style in**_TagTypeToStyle.csv**. All source files sit in the **Bim/Sections** folder of the support folder (6.27.58), e.g.: **C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V18x64\en_US\Support\Bim\Sections.** If tags already exist in the selected viewport, **prompts you:** Tags already have been assigned to the section. Choose an update option [Update existing/update existing and add New/Regenerate all] <Regenerate all>: **(Press Enter to (re)generate all tags or choose an option.)**

Command Options

Option	Description
M Manual	This option creates tags on background entities in sections and elevations. Prompts you: **Pick point on a sectioned BIM element [Auto/Default mleader styles]** - **(Click a bim element). Place the tag** - **(Pick a point to place the tag). Pick point on a sectioned BIM element [Auto/Default mleader styles]** - **(Click a bim element or press Enter to stop).**
A Auto	Tags are created automatically for all sectioned building entities in the selected viewport.
Update Existing	Updates the existing tags in the selected viewport. The position of relocated tag blocs is preserved.
Update Existing and Add New	Updates the existing tags in the selected viewport and adds new tags.
Regenerate All	Updates all tags in the selected viewport. Repositioned tag blocks are moved to their default location.

Related Commands

BmBalloon (6.8.1) - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout. BimProperties (6.7.41) - creates and manages property definitions in a dialog box. Organizes properties in property sets. BimRoom (6.7.26) - creates BIM Room entities. BimSectionUpdate (6.7.29) - exports or updates the result of a BIM section in a drawing. MLeader (6.21.20) - creates multileader entities using the current multileader style. MLeaderEdit (6.21.23) - adds leader lines to and removes leader lines from a multileader entity.

Figure 6.360



6.7.34 BimUpdateRoom

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Updates the geometry of a Room entity.

Accessing the Command


keyboard: **bimupdateroom** menu bar: **BIM | Update Room** toolbar: **BIM Rooms** Quad TAB: **Model** Ribbon Tab | Panel:**Data | Rooms** : **BimUpdateRoom Prompts you in the command bar:** Select Room: **(Select a Room entity.)** Entities in set: 1 Select Room: **(Select more Room entities or press Enter.)** Updated room <RoomName> ...

Command Options

No command options.

Related Commands

BimRoom (6.7.26) - creates Room entities.

Figure 6.361



6.7.35 BimUpdateThickness

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Re-applies the overall thickness of a composition to the solid.

Accessing the Command

command bar: **bimupdatethickness** menu bar: **BIM | Update Thickness** toolbar: **BIM |** : **BimUpdateThickness** **Prompts you in the command bar:** Select entities to update thickness: **(Select an entity.)** Entities in set: 1 Select entities to update thickness: **(Select an entity or press Enter.)** Done on 1 entity(ies). Thickness of x element(s) has been updated. Skipped: y element(s).

Command Options

No options.

Related Commands

BimAttachComposition (6.7.3) - attaches a BIM composition to solids. Bimflip (6.7.10) - flips the starting face from which the layers of a composition are set out.

Figure 6.362



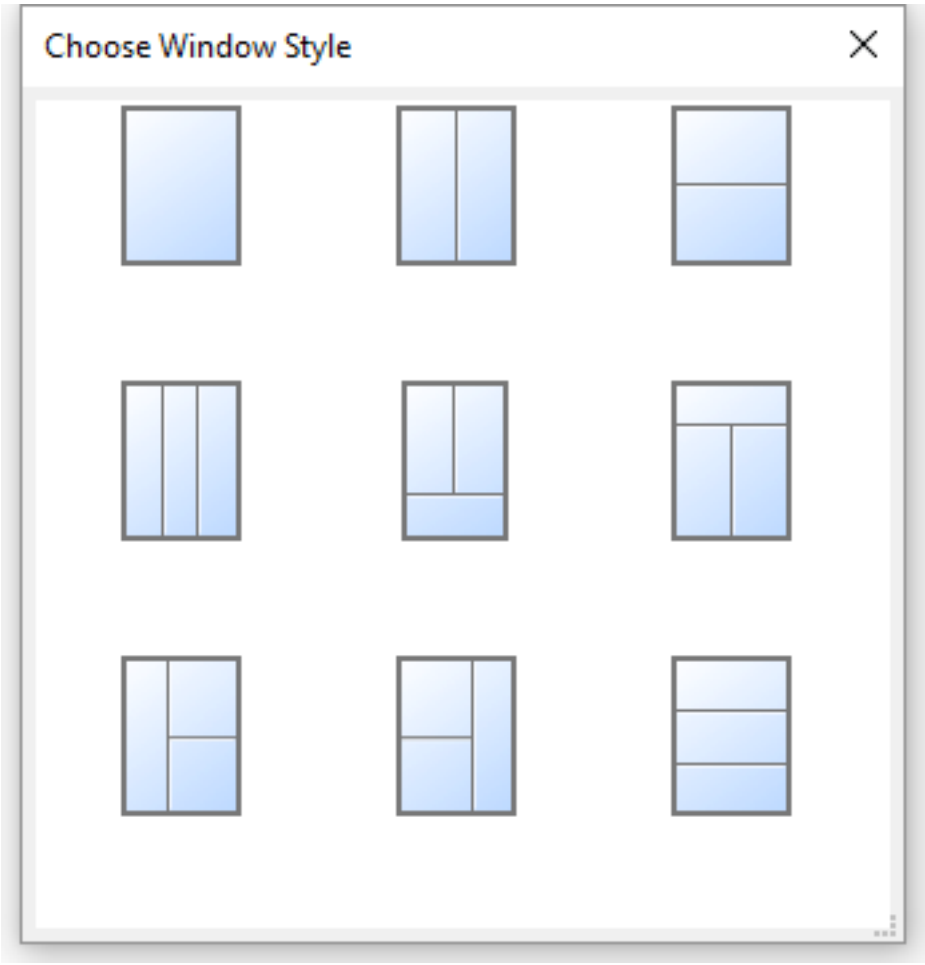
6.7.36 BimWindowCreate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Creates a fully parametric window or opening based on a closed contour.

Accessing the Command

 command bar: **bimwindowcreate** menu bar: **Bim | Create Window** toolbar: **Bim | Create Window** ribbon: **Model | Component| Create Window** Quadcursor menu:**Model | Create Window** : **BimWindowCreate Prompts you in the command bar:** Select profile (boundary, closed polyline or grid) to create [wIndow/Opening] <Window>: (**Select a closed entity or boundary and pressEnter, or type 'O' to activate the opening option).** A Dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ChooseWindowStyleDialog.png}) Pick a window style. The window(s) is (are) created using the selected style. A series of parameters is created, which can be edited in the Parameters section ofthe Properties (6.24.49) panel.Rectangular windows and openings have a W (Width) and H (Height) parameter.

Figure 6.363



Command Options

Option	Description
Select Profile	Select the source geometry for the window or opening. The options are: • **Boundary**: Uses a detected boundary. **Tutorial: How to detect boundaries in BricsCAD BIM** Click here to watch (2.3.6) • **Closed Polyline**: Uses an existing closed polyline. • **Grid**: Uses a BIM Grid entity. See the BimGrid (6.7.12)and the Panelize (6.7.40)commands.
Window 	Creates a BIM Window entity. An opening in thesolid on which the profile entity lies is created automatically.
Opening 	Creates an opening in the solid on which the profile entity lies.The opening is created by a solid on the BIM_SUBTRACT layer. This layer is frozen by default. Thaw the layer to select the opening.

Procedures

To create a window using a closed 2D entity (2.4.3) To create a window using a boundary (2.4.3) To create a window in an existing opening (2.4.3)

Related Commands

BimCurtainWall (6.7.7) - creates a curtain wall from the face of a 3D Solid as a block. BimInsert (6.7.14) - inserts a window or door in a solid. Inserts a copy of a window or door in a solid. BimGrid (6.7.12) - creates 2-dimensional rectangular and radial grids. BimWindowUpdate (6.7.37) - updates the opening made by a window or door in a solid, in case the definition changed in such way that the opening was not correctly updated automatically. Panelize (6.7.40) - creates a grid on the face of a 3D Solid as a block.

Figure 6.364



6.7.37 BimWindowUpdate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Updates the opening made by a window or door in a solid, in case the definition changed in such way that the opening was not correctly updated automatically.

Accessing the Command

 command bar: **bimwindowupdate** menu bar: **BIM| Update Window** toolbar: **BIM | Update Window** ribbon: M**odel | Component| Update Window** Quadcursor menu:**Model | Update Window** : **BimWindowUpdate Prompts you in the command bar:** Select windows to update: **(Select a window or door.)** Entities in set: 1 Select windows to update: **(Select another window or door or right click.)**

Command Options

No options.

Related Commands

BimInsert (6.7.14) - inserts a window or door in a solid. Inserts a copy of a window or door in a solid. BimWindowCreate (6.7.36) - creates a fully parametric window based on a closed contour.

Figure 6.365



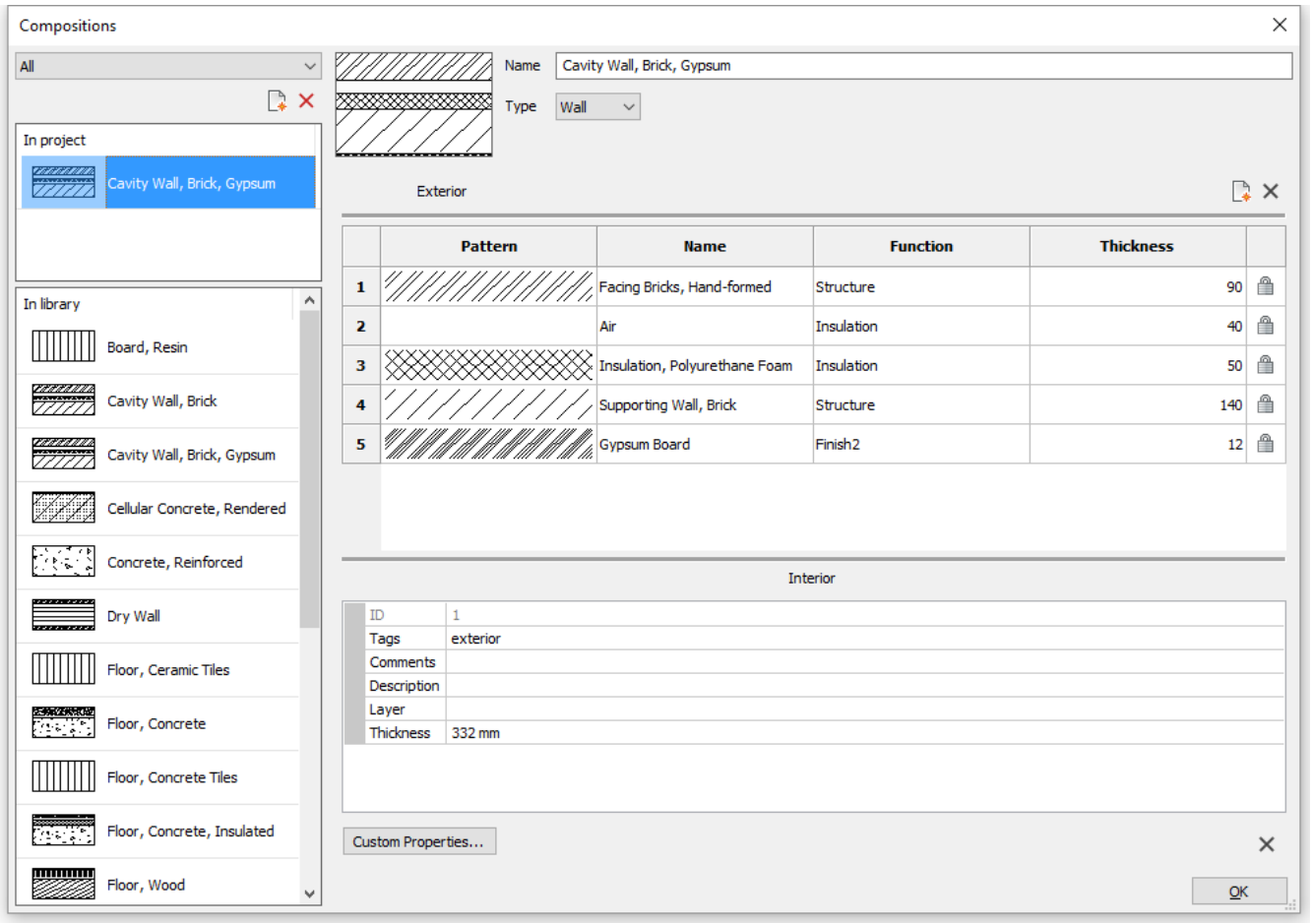
6.7.38 BLCompositions

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Opens the compositions library in the current project and the referenced library.

Accessing the Command


command bar: **blcompositions** The dialog box also displays when you double click a composition in either the **In Project** or the **In Library** list on the BIM Compositions panel. (1.12.7) : **BLCompositions** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/CompositionsDialog.png})

Figure 6.366



Procedures

- About the composition thickness (2.6.2)
- Adding compositions (2.6.2)
- Editing a composition (2.6.2)
- To copy compositions between the Project and Library databases (2.6.2)
- Deleting compositions (2.6.2)
- Attaching a composition to solids (2.6.2)
- Displaying the composition plies (2.6.2)
- To remove a composition from a solid (2.6.2)

Related Commands

BimAttachComposition (6.7.3) - attaches a BIM composition to solids. BimProfiles (6.7.17) - opens the profiles library in the current project and the referenced library. BimProjectInfo (6.7.19) - opens the BIM Project Info dialog box. BLMaterials (6.7.39) - opens the physical materials library in the current project and the referenced library.

Figure 6.367



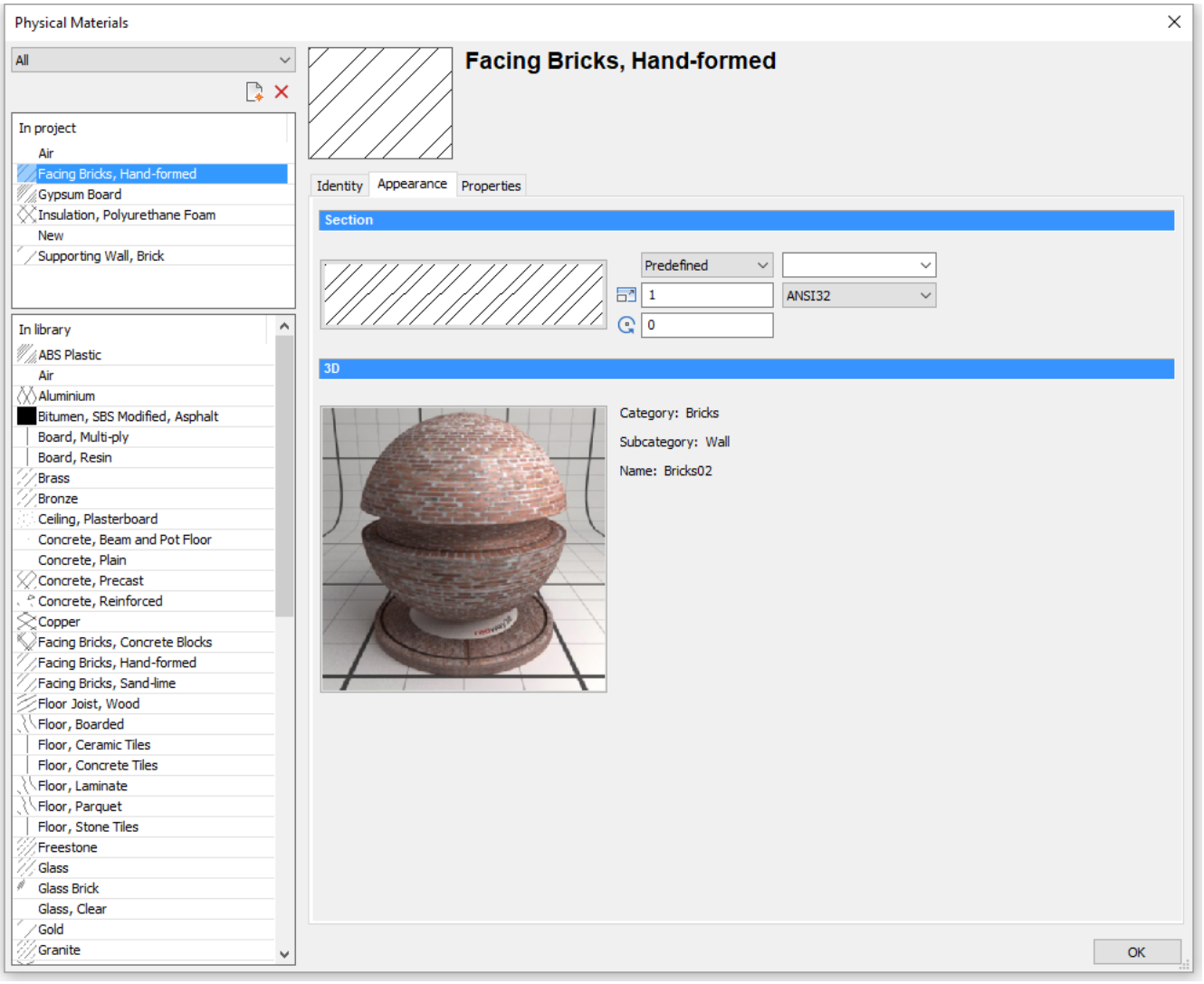
6.7.39 BLMaterials

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Opens the Physical Materials library in the current project and the referenced library.

Accessing the Command

 command bar: **blmaterials** : **BLMaterials** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h

Figure 6.368


Physical Materials
All
In project
Air
Facing Bricks, Hand-formed
Gypsum Board
Insulation, Polyurethane Foam
New
Supporting Wall, Brick
In library
ABS Plastic
Air
Aluminium
Bitumen, SBS Modified, Asphalt
Board, Multi-ply
Board, Resin
Brass
Bronze
Ceiling, Plasterboard
Concrete, Beam and Pot Floor
Concrete, Plain
Concrete, Precast
Concrete, Reinforced
Copper
Facing Bricks, Concrete Blocks
Facing Bricks, Hand-formed
Facing Bricks, Sand-lime
Floor Joist, Wood
Floor, Boarded
Floor, Ceramic Tiles
Floor, Concrete Tiles
Floor, Laminate
Floor, Parquet
Floor, Stone Tiles
Freestone
Glass
Glass Brick
Glass, Clear
Gold
Granite
Facing Bricks, Hand-formed
Identity
Appearance
Properties
Section
Predefined
1
ANSI32
0
3D
Category: Bricks
Subcategory: Wall
Name: Bricks02
OK

Procedures

- Adding physical materials (2.6.10)
- Editing a physical material (2.6.10)
- To copy materials between the Project and Library databases (2.6.10)
- Deleting physical materials (2.6.10)

Related Commands

BimProfiles (6.7.17) - opens the profiles library in the current project and the referenced library. BimProjectInfo (6.7.19) - opens the BIM Project Info dialog box. BLCompositions (6.7.38) - opens the compositions library in the current project and the referenced library.

Figure 6.369



6.7.40 Panelize

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Creates a grid on the face of a 3D Solid as a block.

Accessing the Command

 Keyboard: **panelize** Menu: **BIM | panelize** Toolbar: **BIM Curtain Wall** Quad Tab: **Model** Ribbon Tab | Panel: **Model | Curtain Wall : Panelize Prompts you:** Select face or set [Result]: **(Select the face of a solid)** Create grid or [Length U panels/Length V panels/Number U panels/Number V panels]: **(Press Enter or choose an option.)** ===ERROR WITH DOWNLOADED IMAGE===

Command Options

Option	Description
Length U panels	Sets the length of the panels in U direction (height). Prompts you: **U grid length <550>:(Type a value or press Enter).**
Length V panels	Sets the length of the panels in V direction (width). Prompts you: **V grid length <3000>:(Type a value or press Enter).**
Number U panels	Number of panels in U direction (rows). Prompts you: **U grid size <5>:(Type a number or press Enter).**
Number V panels	Number of panels in V direction (columns). Prompts you: **V grid size <3>:(Type a number or press Enter).**

Related Commands

BimCurtainWall (6.7.7) -Creates a curtain wall from the face of a 3D Solid as a block.

Figure 6.370



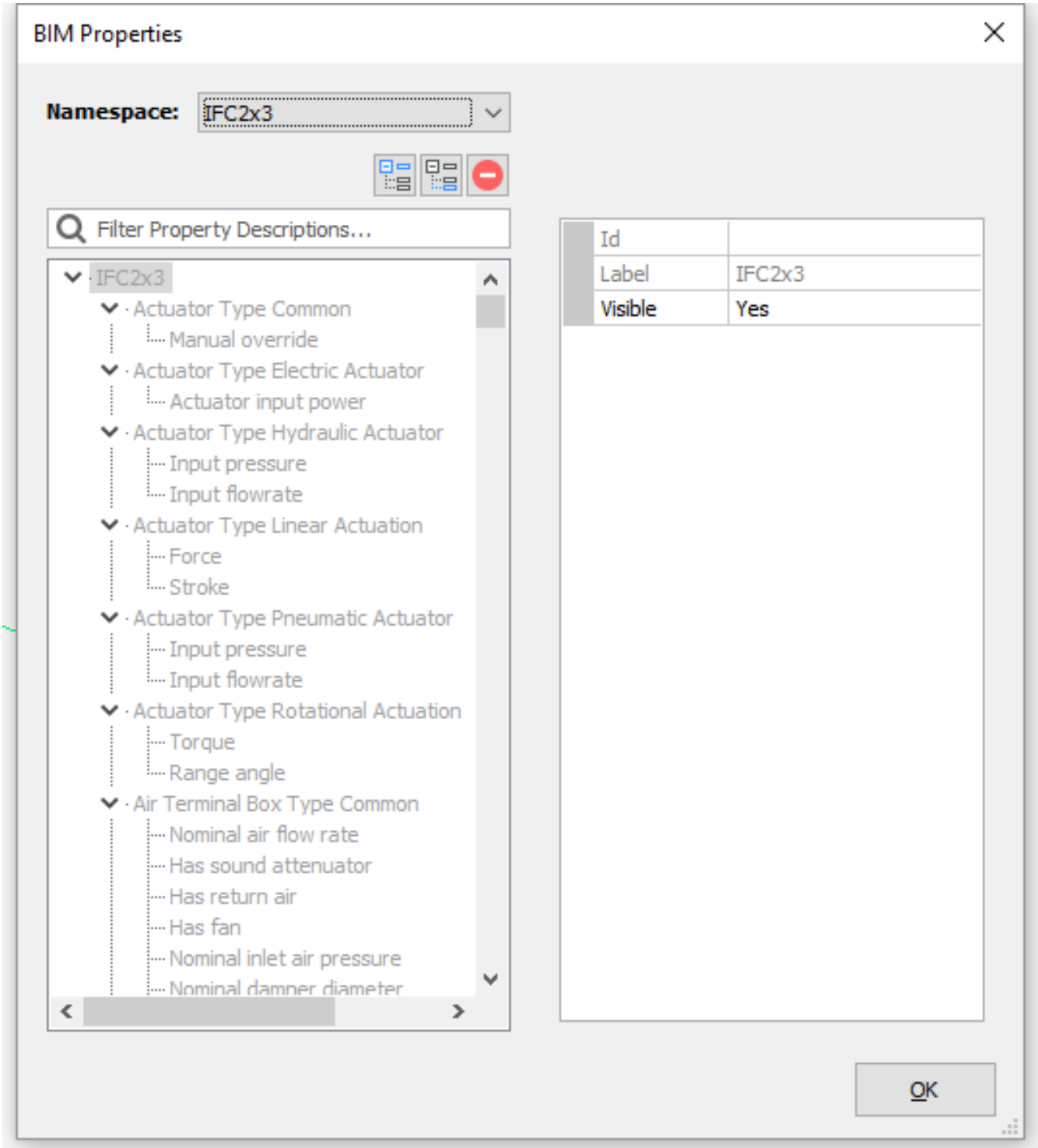
6.7.41 BimProperties

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Creates and manages property definitions in a dialog box. Organizes properties in property sets. BIM properties are included in IFC export (6.13.15).

Accessing the Command


Keyboard: **bimproperties** Toolbar:**BIM Databases** Quad Tab:**BIM** Ribbon Tab | Panel: **Data | Database** : **BimProperties** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/Bi
The BIM properties are saved in the ***.bsyslib** project database. See Project Info (2.6.5) in the User Guide.

Figure 6.371


BIM Properties
Namespace:
IFC2x3
Filter Property Descriptions...
IFC2x3
Actuator Type Common
Manual override
Actuator Type Electric Actuator
Actuator input power
Actuator Type Hydraulic Actuator
Input pressure
Input flowrate
Actuator Type Linear Actuation
Force
Stroke
Actuator Type Pneumatic Actuator
Input pressure
Input flowrate
Actuator Type Rotational Actuation
Torque
Range angle
Air Terminal Box Type Common
Nominal air flow rate
Has sound attenuator
Has return air
Has fan
Nominal inlet air pressure
Nominal damper diameter
Id
Label
IFC2x3
Visible
Yes
OK

Command Options

Option	Description
Namespace	Properties exist in a namespace. Two namespaces are available: IFC2x3 : IFC properties are read-only. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ • Id : This field is empty and cannot be edited. • Label : This field cannot be edited. • Visible : Defines whether the IFC2x3 properties are visible in the Properties panel (1.12.4). User : (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ • Id : Name of the userspace. This field is not editable. • Label : The label of the namespace is used in the properties tree. If the Label field is empty, the Id is used instead. • Visible : Defines whether the namespace properties are visible in the Properties panel (1.12.4).
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/BimProperties.png})	Allows to filter the property tree. Type a search string in the **Id** or the **Label** contains the search string display. The properties tree is filtered as you type.
IFC2x3 namespace	The IFC2x3 properties cannot be edited. You can only control the visibility of the various property sets. To control the visibility of an IFC property set: • Select IFC2x3 namespace. • Select the property set. • Set the **Visible** property of the property set to **Yes** .
User namespace	

 Add Property Set	Allows to add a property set in the current namespace. Click the button to add a new property set. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ Do the following: • Type a name in the **Id** field. • Type a name in the **Label** field. The **Label** name appears in the properties tree. If the **Label** field is empty, the **Id** name displays. • Set the **Per Instance** property (not implemented yet). • Set the **Visible** property. Defines whether the properties in the property set are visible in the Properties panel (1.12.4). • Select the building elements the property set applies to. All entities which are classified this way get the property assigned automatically. Available categories and building elements are the same as in the BimClassify (6.7.5) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands • Expand a category, then check building elements. • Click a category checkbox to select all building elements in that category • Click the **All** button to select all building elements in all categories. • Click the **None** button to uncheck all building elements in all categories
 Add Property	Allows to add a property to the currently selected property set. Select a property set in the properties tree, then click the button to add a new property. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ Do the following: • Type a name in the **Id** field. • Type a name in the **Label** field. The **Label** name appears in the properties tree. If the **Label** field is empty, the **Id** name displays. • (optional) Type a description in the **Description** field. • Select the property Type. The options are: **Boolean** , **Integer** , **Real** and **String** .
 Delete	Deletes the selected Property or Property set.

Related Commands

BimExport (6.7.9) - saves a BIM model as an IFC file for use with an IFC-certified applications. BimClassify (6.7.5) - classifies an entity as a Building Element and get a name and an internal GUID (= globally unique identifier). BimTag (6.7.33) - creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model.

Figure 6.372

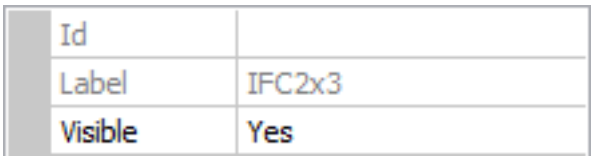


Figure 6.373

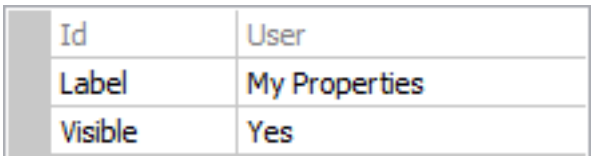


Figure 6.374



Figure 6.375

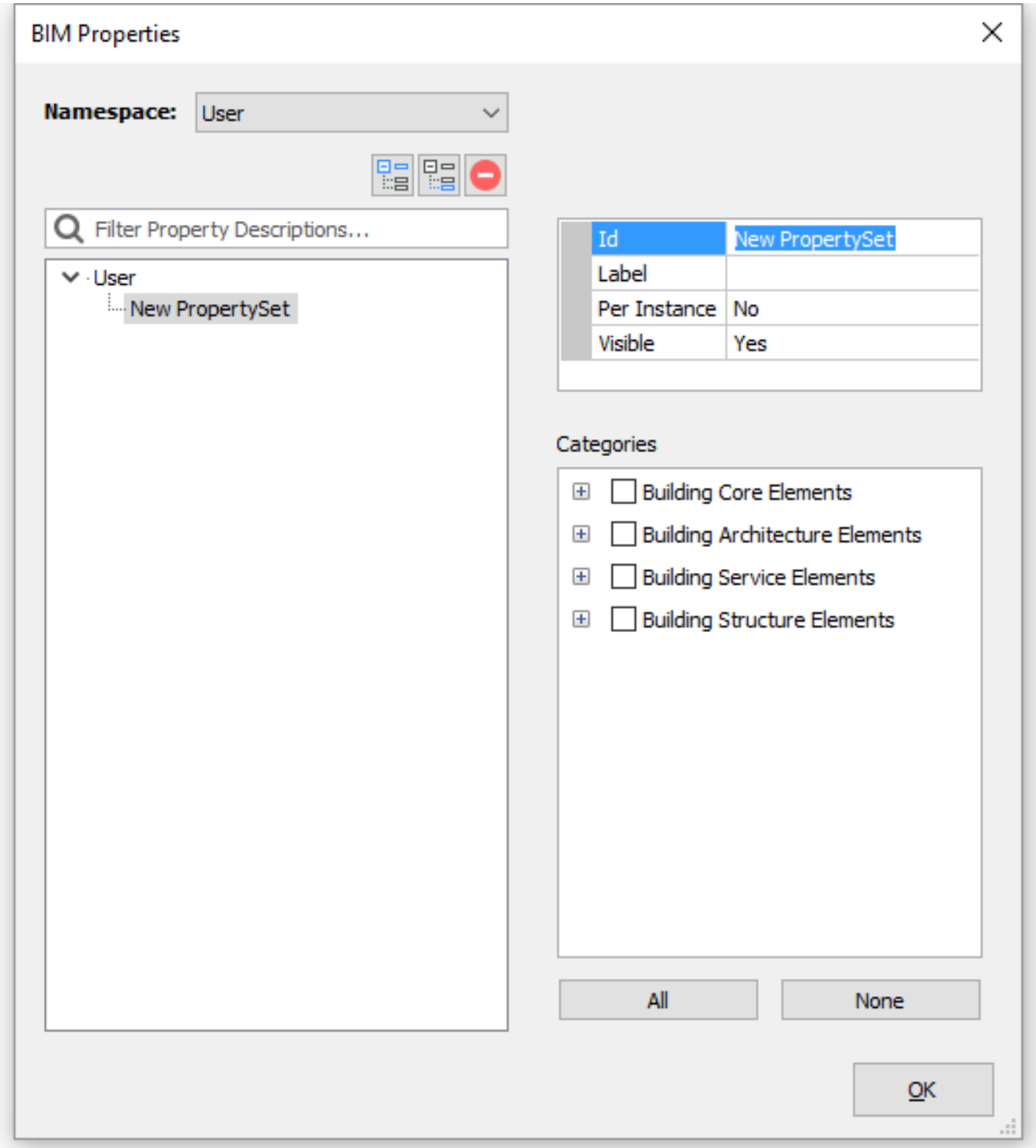

BIM Properties
Namespace:
User
Filter Property Descriptions...
User
New PropertySet
Id
New PropertySet
Label
Per Instance
No
Visible
Yes
Categories
Building Core Elements
Building Architecture Elements
Building Service Elements
Building Structure Elements
All
None
OK

Figure 6.376

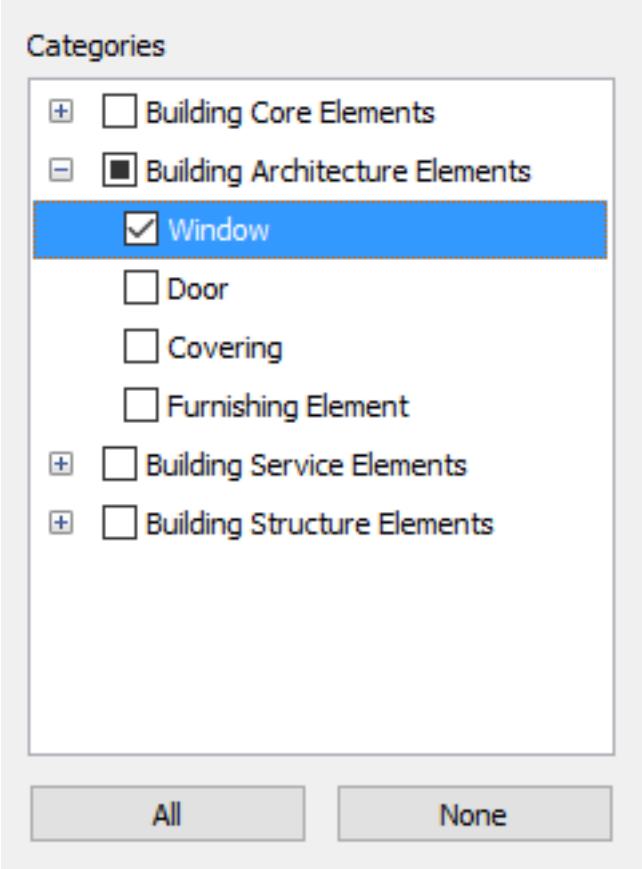


Figure 6.377

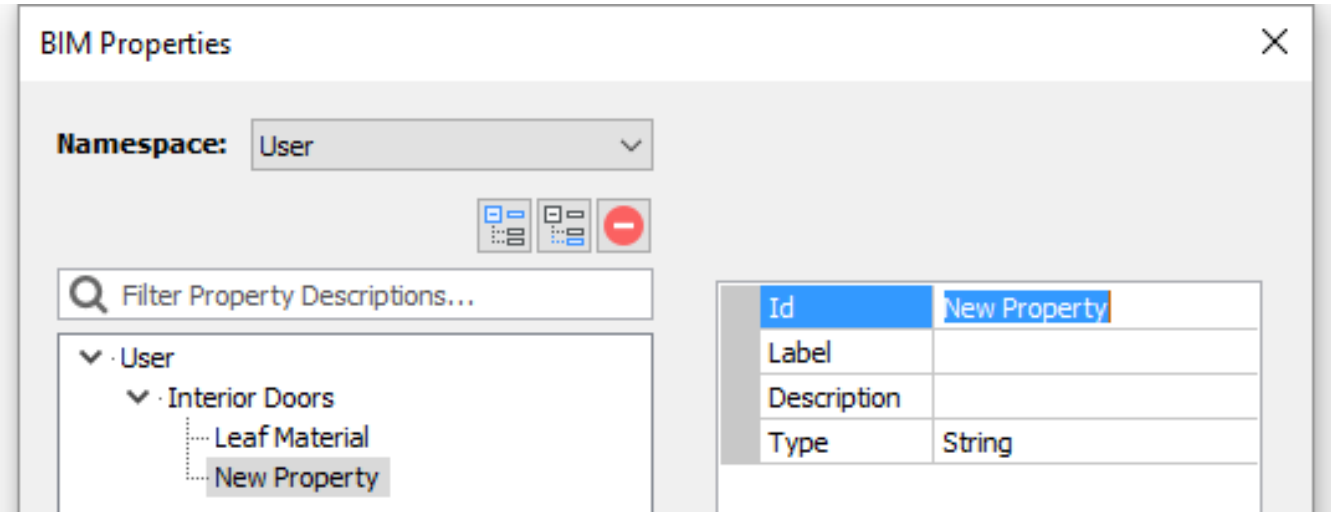


Figure 6.378



6.7.42 BimQuickDraw

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-bim.png}) Use QuickDraw to quickly create and edit rectangular and L-shaped rooms and buildings.

Tutorial: How to use Quickdraw Click here (2.3.23) to watch

Accessing the Command


Keyboard: **bimquickdraw** Menu: **BIM|Quickdraw** Toolbar: **BIM** Quad Tab: **Model** Ribbon Tab | Panel:**Model|Create 3D :BimQuickdraw** The layout of the QuickDraw cursor represents the room being created: Left to right: placed freely, aligned with one existing walls, aligned with two existing walls. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/QuickDrawCursor2.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/QuickDrawCursor3.png}) **Prompts you:** Current settings: Room Height: 3000, Wall Width: 250, Slab Thickness: 250 Select first point or [room Height/wall Width/slab Thickness/Auto adopt sizes]: **(Select a point to define the first corner of the room or choose an option.)** Select second point: **(Select the a point to define the opposite corner of the room.)**

Figure 6.379

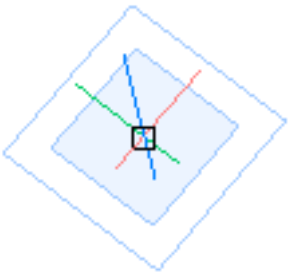


Figure 6.380

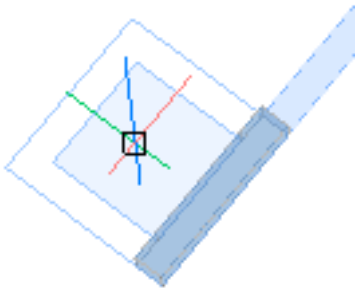
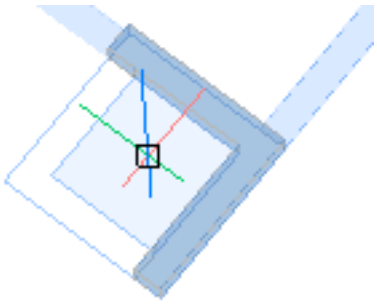


Figure 6.381



Command Options

Option	Description
Room Height	Sets the height of the room. The height is saved to the **Default Room Height** system variableand will be used until set differently. Default room height = 3000 mm The height of the exterior walls = Room Height + Slab Thickness. The height of interior walls = Room Height.
Wall Width	Sets the width of the walls being created. The width is saved to the **Default Wall Width** system variable and will be used until set differently. Default wall width = 250 mm.
Slab Thickness	Sets the slab thickness of the room. The thickness is saved to the **Default Slab Thickness** system variableand will be used until set differently. Default slab thickness = 250 mm.

Select first point	Defines the first corner of a room. The layout of the QuickDraw cursor indicates the alignment with existing walls and distances from existing solids. Depending on the layout of the QuickDraw cursor: • Default cursor: Click a point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands • Aligned with one exiting wall; distance from one existing solid: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands – Type a value in the distance field. – Click a point to accept the alignment • Aligned with two existing walls; distances from two existing solids. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands – Type a value in the distance field, then press the TAB key and type value in the second distance field. Optionally press the TAB key to adjust the value in the first distance field. Press Enter to accept the distances. – Click to accept the alignment.
Select second point	Defines the opposite corner of a room. When creating the exterior walls of a building, do one of the following: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ • Click a point. • Type a value in the distance field, then press the TAB key and type value in the second distance field. Optionally press the TAB key to adjust the value in the first distance field. Press **Enter** to accept the distances. When creating rooms: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ • Click a point. • Type a value in the distance field, then press the TAB key and type value in the next distance field. Optionally press the TAB key again to type a value in the next distance field, thus overriding the value in the first distance field. If necessary, keep pressing the TAB key to adjust the distances, then press Enter to accept.
Auto adopt sizes	Allows to copy the wall width and room height from an existing wall, thus overriding the value of the**Default Room Height**and **Default Wall Width**system variables. Prompts you: **Auto adopt sizes On/OFF <On>** - **(choose an option.) Note:** The value of Auto adopt sizes is reset to ON when you close and restart BimQuickDraw.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/QuickDrawAddStory.png}) Add Story	Hover the cursor over the icon ( ) to see the options: • Full copy of the top floor. • Copies the slab and the outer walls of the top floor. • Finish the building with flat roof. Click an icon to add a new story or to finish the building with a flat roof.

Procedures

- Creating an L - shaped house (**??**)
- Creating a Story and a Roof
- Setting the Quickdraw system variables

Related Commands

PolySolid (6.24.44) - Creates wall-like or pitched roof 3D solids. BimConnect (6.20.19) - Creates an L-connection between minor faces of two solids. BimRoom (6.7.26) - Creates BIM Room entities.

Figure 6.382

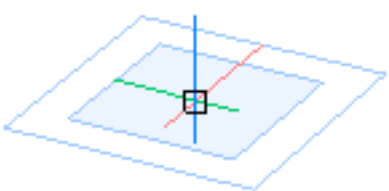


Figure 6.383

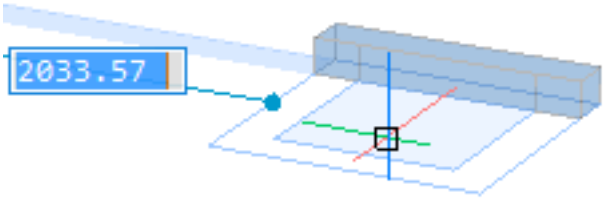


Figure 6.384

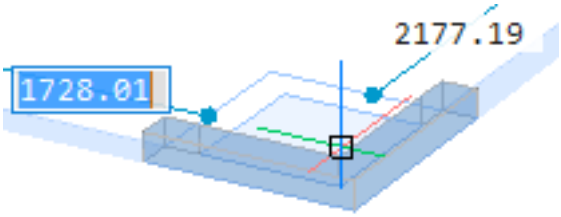


Figure 6.385

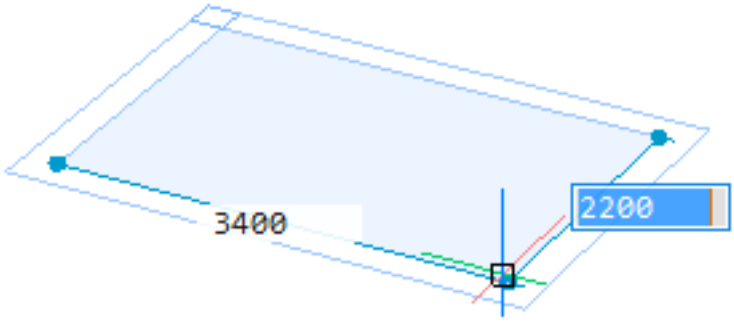


Figure 6.386

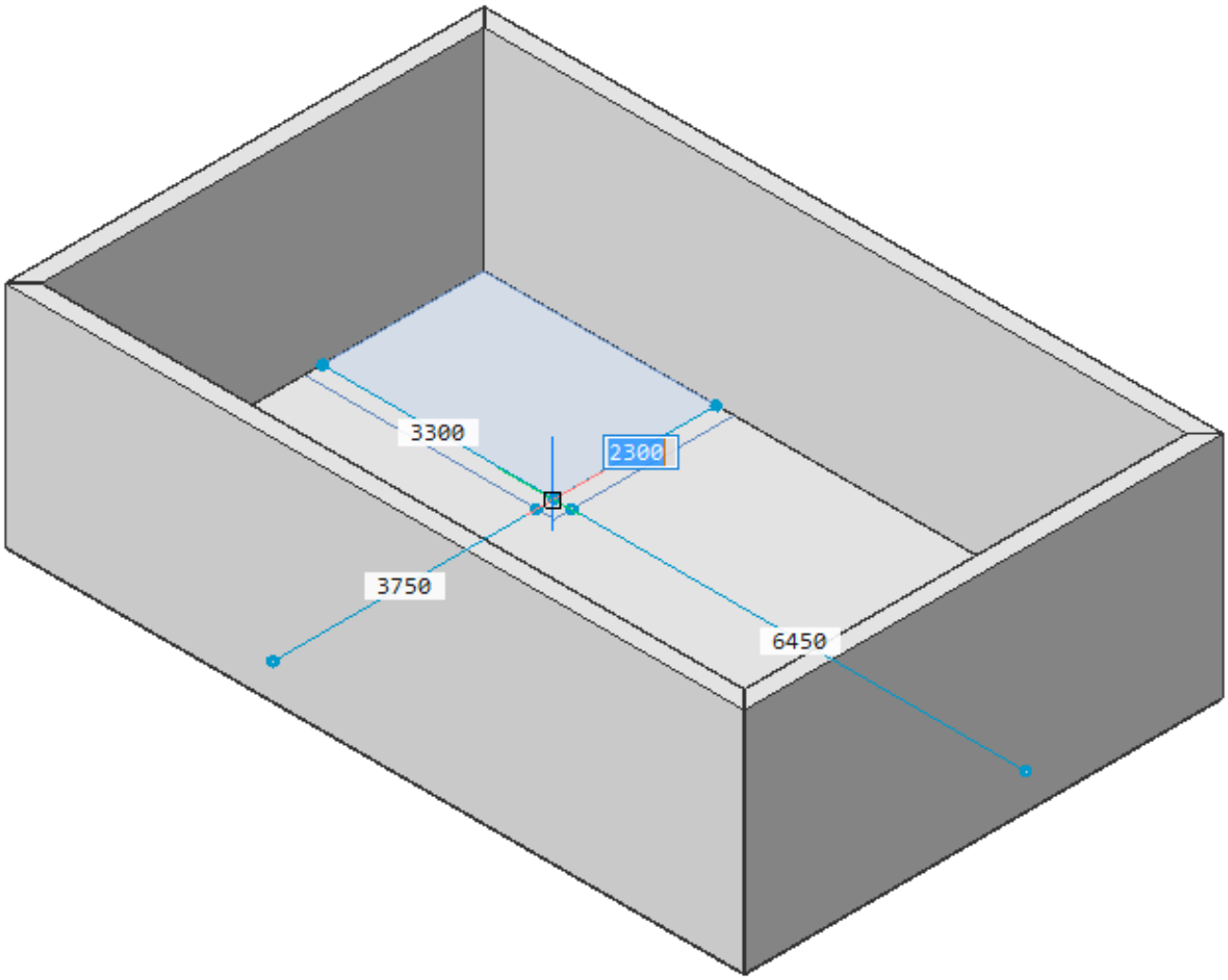


Figure 6.387

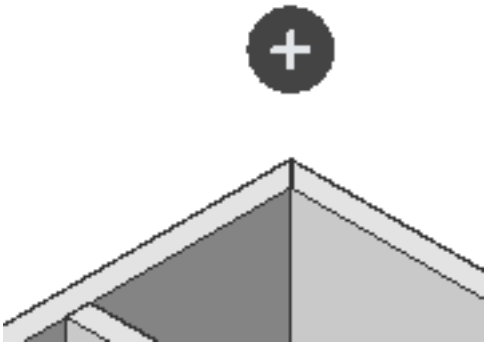


Figure 6.388



Figure 6.389



6.7.43 BimSetReferenceFace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Defines a reference and an opposing face to control the ply layout of a composition. If one of the plies of the composition has a variable thickness, the reference and opposing face can be non-parallel. In this case the fixed thickness plies are set out starting from the reference face, and the remainder of the solid is filled by the variable ply. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/SetReferenceFaceNonParallel.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/SetReferenceFaceNonParallel2.png

Figure 6.390



Accessing the Command

Keyboard:**bimsetreferenceface : BimSetReferenceFace Prompts you:** Select reference face: **(Select the reference face of the solid.)**

- If the selected face is parallel to the opposing face: Opposing face(s) are detected [Select manually] <Accept>: **(Press Enter to accept the parallel face or typ S to manaully select the opposing face manually.)**
- If the selected face is not parallel to the opposing face: Select opposing face: **(Select a face.)**

Reference face was set. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/SetReferen (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/SetReferenceFace2.png}) Solids without composition Multiply composition assigned Reference faces are either wrong or not found (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/SetReferenceFace3b.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/SetReferenceFace4.png}) Defining reference face (1) and opposing face (2) Attaching a composition (6.7.3) Reference faces highlight (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/SetReferenceFace5.png}) Result

Command Options

Option	Description
Select manually	Allows to select the opposite face manually. Prompts you: **Select opposing face: (Select a face.)**

Related Commands

BimAttachComposition (6.7.3) - attaches a BIM composition to solids. BimFlip (6.7.10) -flips the starting face from which the layers of a composition are set out.

Figure 6.391

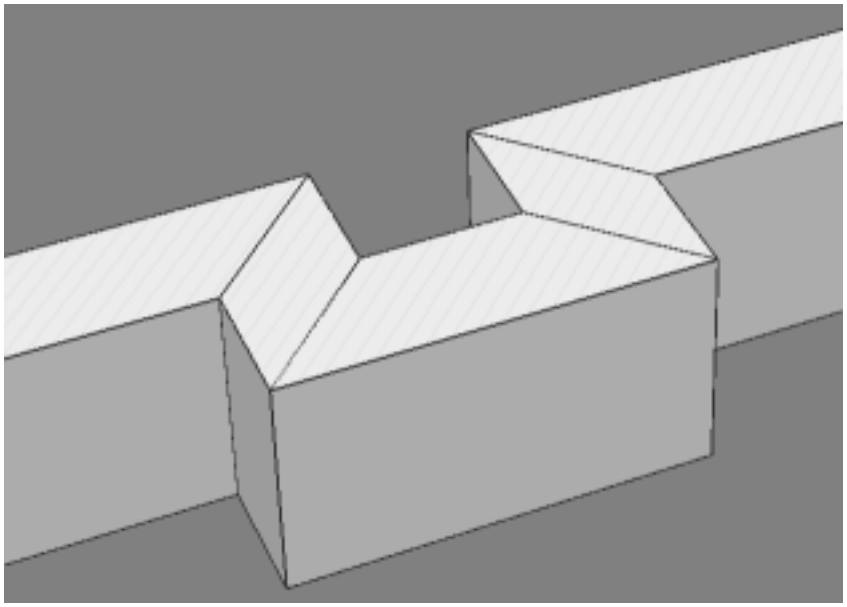


Figure 6.392

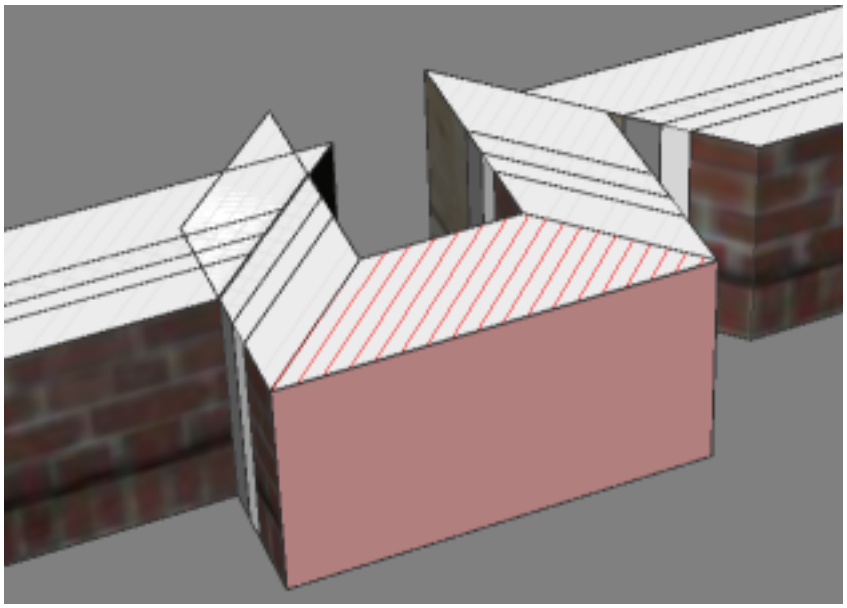


Figure 6.393

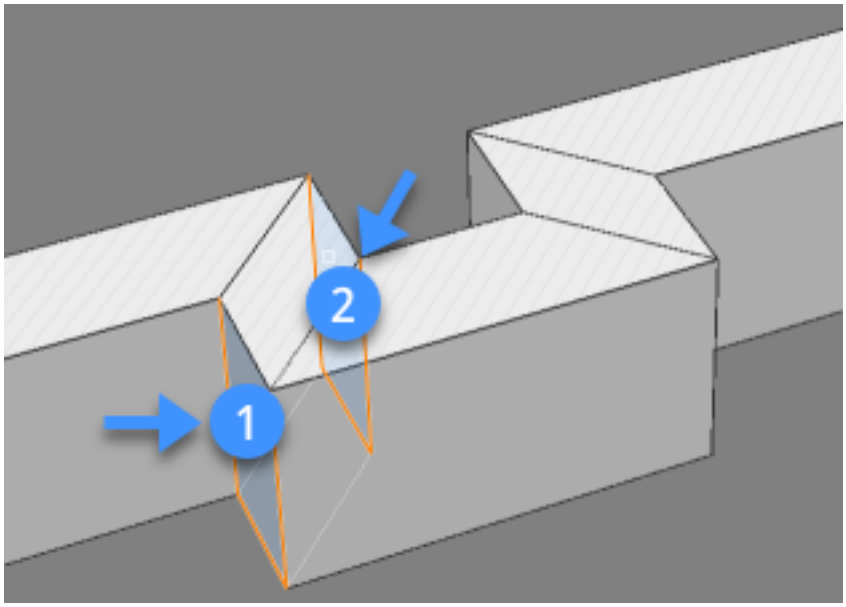


Figure 6.394

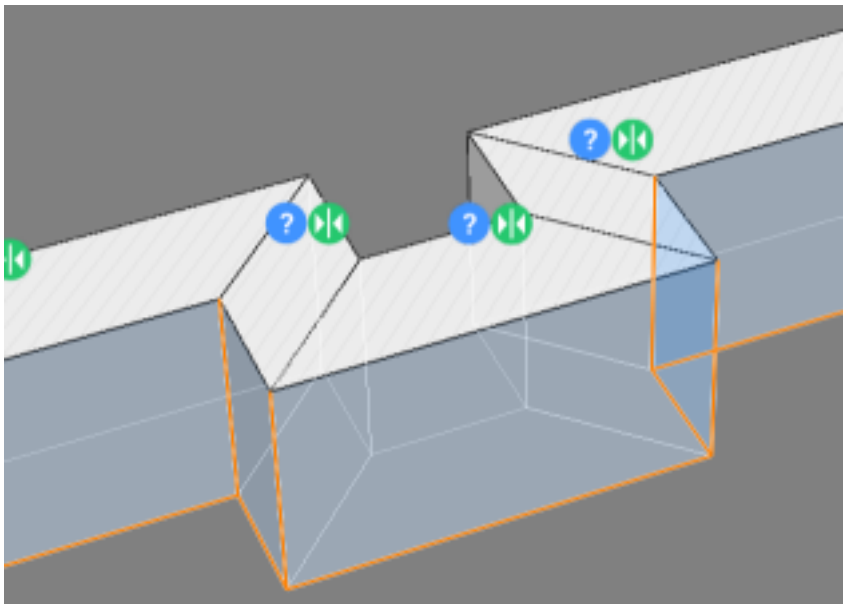


Figure 6.395

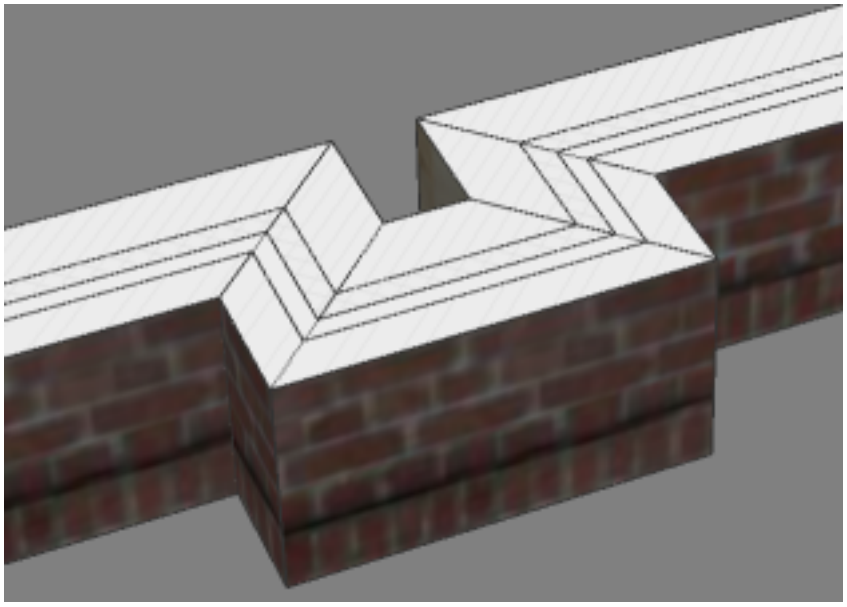


Figure 6.396



6.7.44 BimRecalculateAxis

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Recalculates the axis of linear building elements and repositions it back on the center line of a linear element.

Accessing the Command

 command bar: **bimrecalculateaxis** Quad: **Model** : **BimReCalculateAxis Prompts you in the command bar:** Select several entities/subentities: **(Select a linear solid.)** Entities in set: 1 Select several entities/subentities: **(Select more linear solids or press Enter.)**

Command Options

No command options.

Related Commands

BimAddEccentricity (6.7.1) - allows to control the relative position of the axis of a linear solid. BimProfiles (6.7.17) -Opens the profiles library in the current project and the referenced library.

6.8 Bm (Mechanical)

6.8.1 BmBalloon

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png})
Creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout. Each balloon contains an index of the component in the associated BOM-table created by BmBom (6.8.2) command.

Figure 6.397



Accessing the Command



command bar: **bmballoon** menu bar: **Assembly | Balloon** toolbar: **Assembly | Balloon** ribbon:

- **Assembly | Inquire | Balloon**
- **Annotate | Assembly | Balloon**

: bmballoon Prompts you in the command bar: Select a component insert [select Table/select Style/Auto mode]: **(Click a component insert.)** Pick point to place balloon: **(Define the location of the balloon.)** Select a component insert [select Table/select Style/Auto mode]:**(Click the next component or press Enter to stop.)**

NOTE	Balloons are created as Multileader (6.21.20) entities using the current Multileader style (6.21.25). Create a Multileader style with the **Content** set to **Block** and adjust the **Scale** property to control the size of the balloon frame.

Command Options

Option	**Description**
Select a component insert	Selects the component and sets the location of the arrowhead.
Pick point to place balloon	Locates the balloon; prompts you: Pick point to place balloon - pick a point. Hold down the SHIFT key to align the balloon with the previous one.
Select table	Allows to select a different BOM-table in case multiple BOM-tables exist in the layout. The most recently insert BOM-table is selected by default; prompts you: **Select a BOM table <Back>** - **(Select the BOM-table or press Enter to proceed with the currently selected BOM-table.)**
select Style	Allows to define the shape of the balloon; prompts you: **Choose balloon style**: - **(choose an option in command menu.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M

(\vref {fig:C:/Dev/Task-2448/images/	(\vref {fig:C:/Dev/Task-	(\vref {fig:C:/Dev/Task-	(\vref {fig:C:/Dev/Task-
Circle	No frame	Underline	Slot
(\vref {fig:C:/Dev/Task-2448/images/	(\vref {fig:C:/Dev/Task-	(\vref {fig:C:/Dev/Task-	(\vref {fig:C:/Dev/Task-
Box	Hexagon	Triangle	Rhomb

Predefined balloon styles from _BalloonStyles.dwg can be used, as well as any custom user-defined styles.

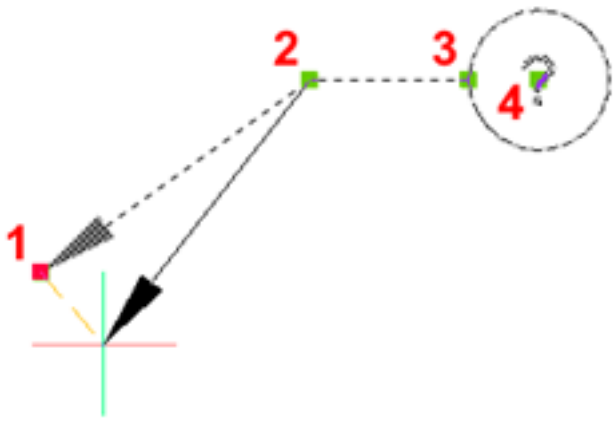
Auto mode	Allows automatic placement of balloons on all components in a particular drawing view. prompts you: **Select a viewport** [select other Table/choose balloon Style/placement Mode/Back]: • select other Table -Allows selecting a different BOM-table in case multiple BOM-tables exist in the layout. prompts you: Select a BOM table <Back> - Select the BOM-table or press Enter to proceed with the currently selected BOM-table; • choose balloon Style - choose an option in command menu; • placement Mode -allows select whether multiple balloons with the same values are generated or not. prompts you: Select placement mode <All> [Unique/All]: - controls whether multiple balloons with same values are generated or not. • Back -Returns to the previous prompt. Select first point [Circle]: - select the first point of a polyline or switch placement mode in command menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Align leaders along polyline: - select the next point of a polylineor press Enter to stop.
	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/ placement by the Polyline

Grips Editing

1. Select the balloon in the drawing. For a single segment balloon entity, four grips display: Endpoint (1), Vertex (2), Landing (3) and Text (4).
2. Drag the grips to perform the following actions:
   - Move the arrowhead (1) If move the arrowhead to a different component, the balloon automatically updates the position number according to the associated BOM-table. If the arrowhead hits a blank space, the question mark displays.
   - Move the landing and the balloon (2)
   - Edit the landing length(3)
   - Move the balloon (4)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Mechanical/ART/BmBalloonGripEdi

Figure 6.398



Related Commands

BimTag (6.7.33) - creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model. BmBom (6.8.2) - inserts a Bill of Materials (BOM) table in the current drawing. Ddedit (6.10.14) - edits single-line text, multi-line text, attribute definitions, and attribute text. DimReassociate (6.11.18) - reassociates or associates dimensions to entities or points on entities. MLeader (6.21.20) - creates multileader entities using the current multileader style. MLeaderEdit (6.21.23) - adds leader lines to and removes leader lines from a multileader entity. MLeaderStyle (6.21.25) - creates and manages multileader styles through the Drawing Explorer.

Figure 6.399

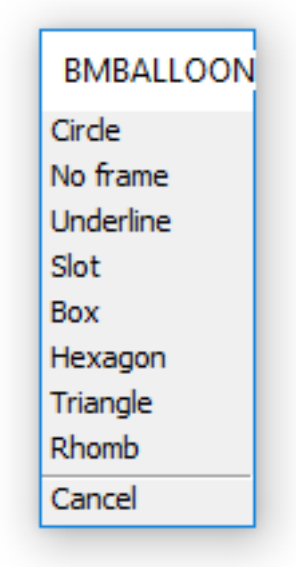


Figure 6.400

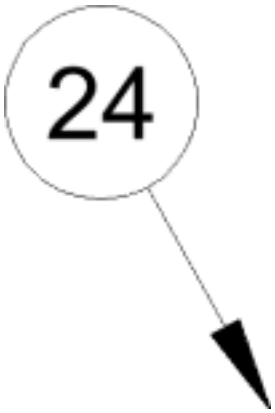


Figure 6.401

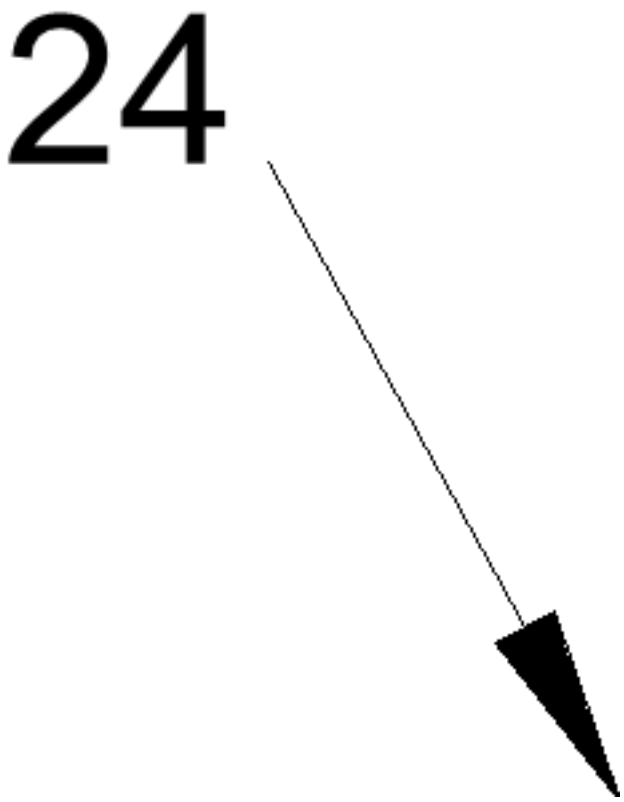


Figure 6.402

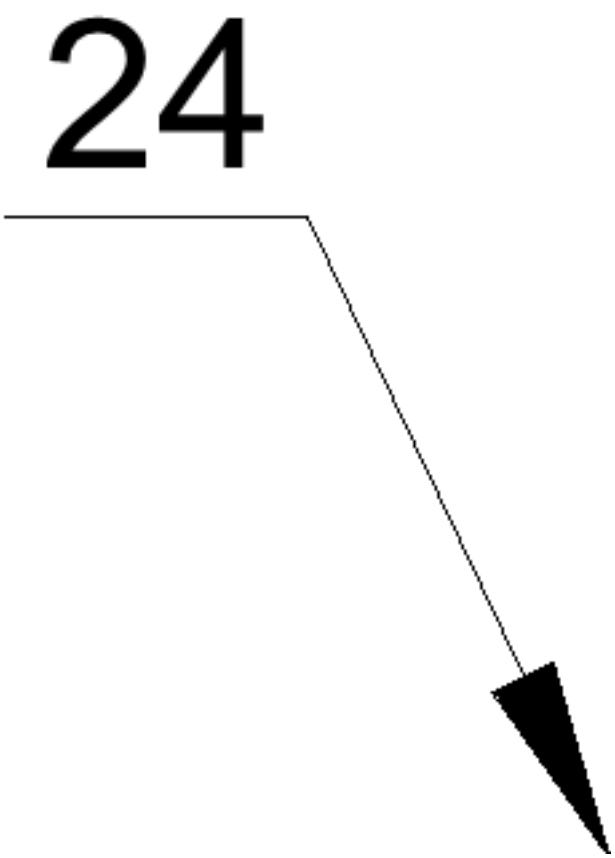


Figure 6.403

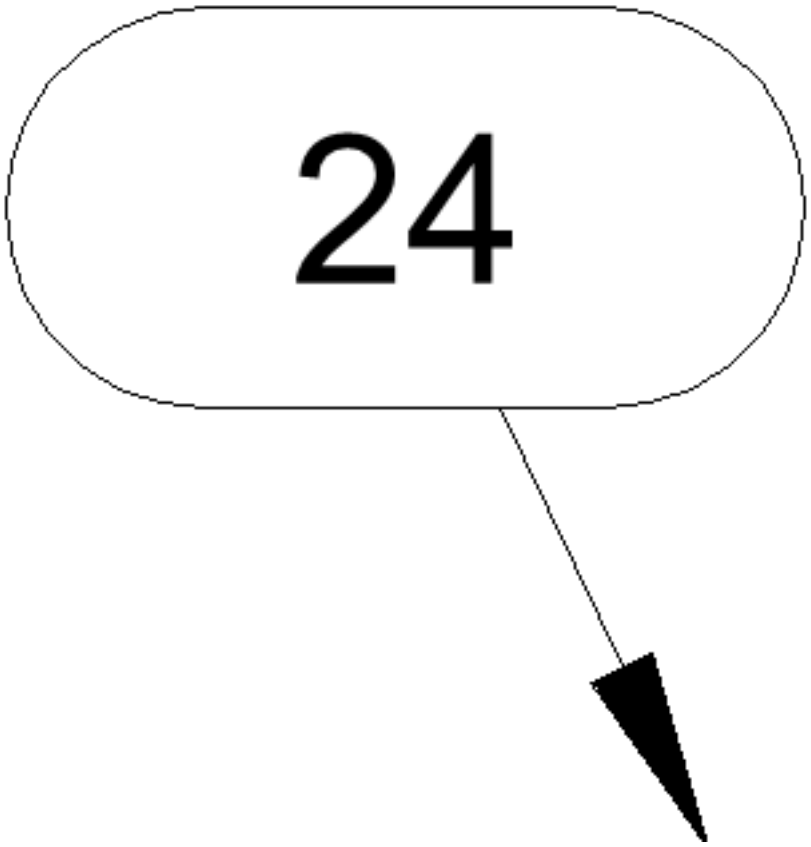


Figure 6.404

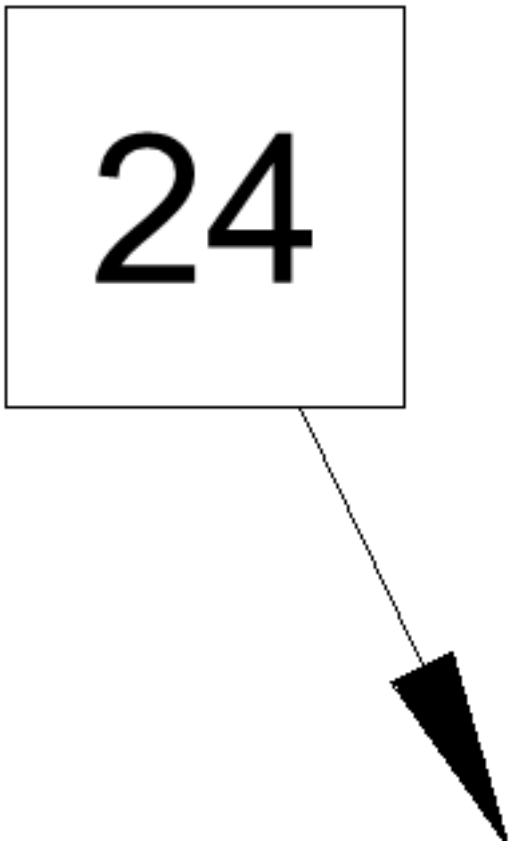


Figure 6.405



Figure 6.406

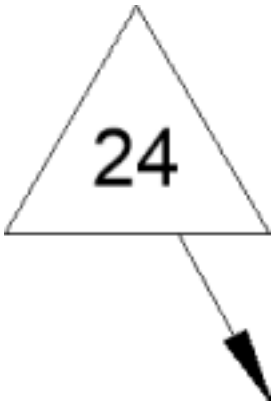


Figure 6.407



Figure 6.408

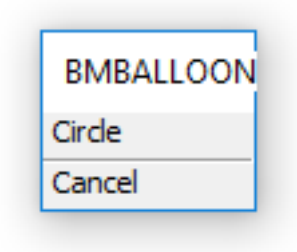


Figure 6.409

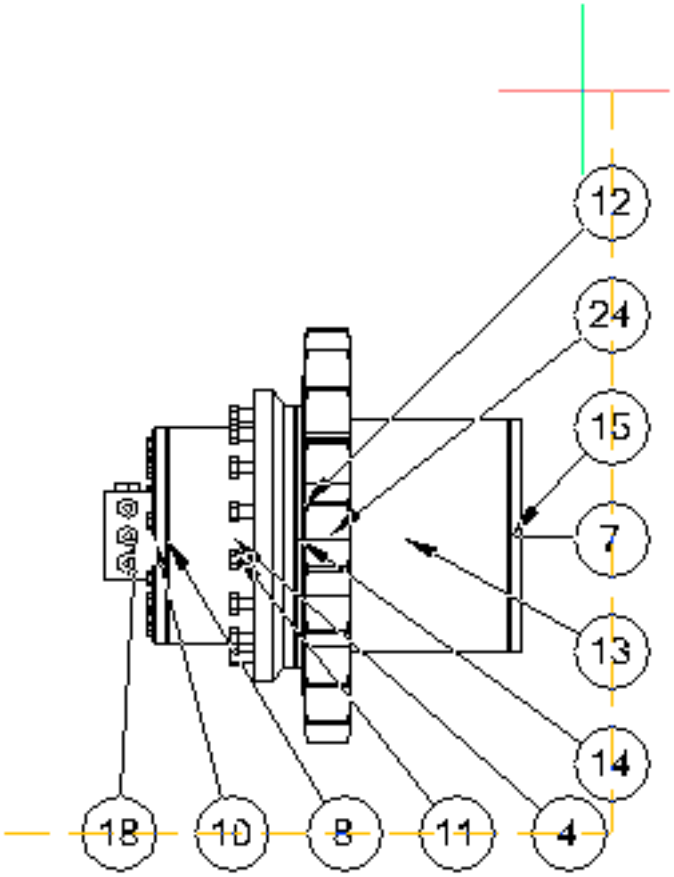


Figure 6.410

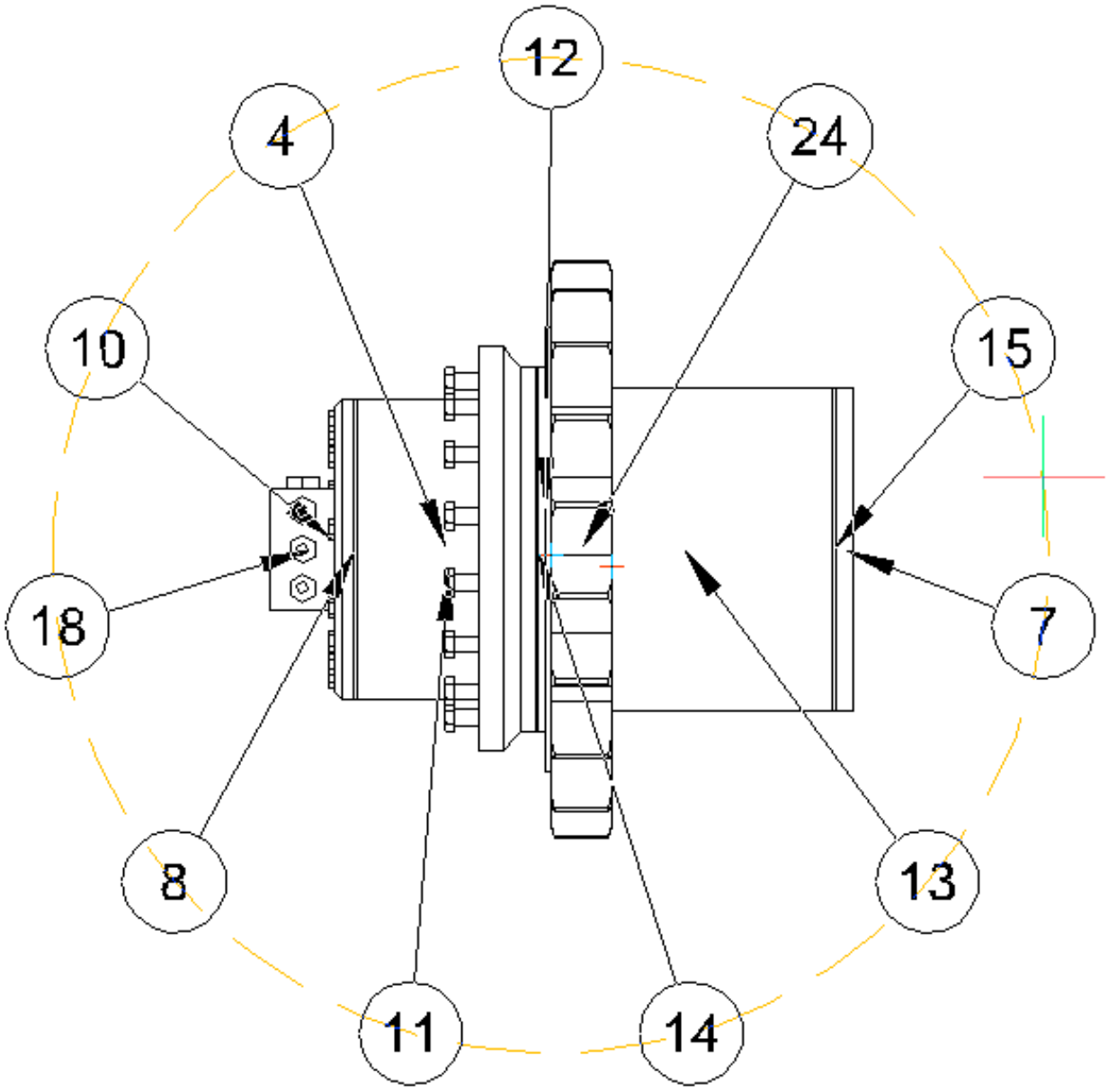


Figure 6.411



6.8.2 BmBom

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Inserts a Bill of Materials (BOM) table in the current drawing.

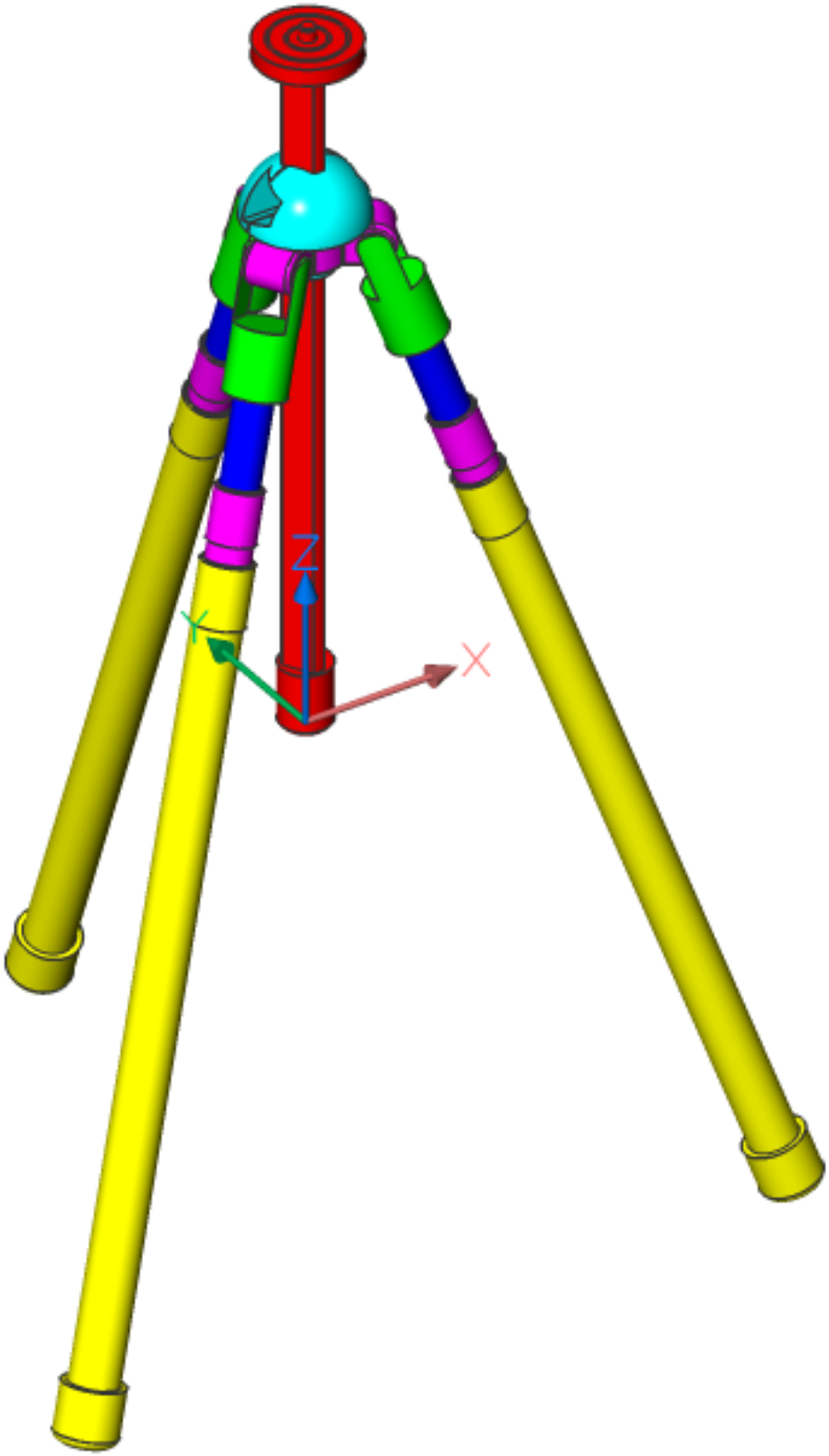
Accessing the Command

command bar: **bmbom** menu bar: **Assembly | Bill Of Materials** toolbar:**Assembly |Bill Of Materials** ribbon:

- **Assembly | Inquire | Bill Of Materials**
- **Annotate |Assembly | Bill Of Materials**

: **bmbom Prompts you in the command bar:** Insertion point [Name/Top level/Bottom level/Hierarchical/Configure]: **(Specify the insertion point or enter an option.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdE

Figure 6.412



Command Options

Option	Description
Name	Prompts you to type a name in the command bar: Table name <Default Name>: Type the name, then press Enter. Prompts you: **Insertion point** [Name/Top level/Bottom level]: -**Specify the insertion point or enter an option.**
Top level	Creates a BOM of the top level components only. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
Bottom level	Creates a BOM of the subcomponents. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Compared to the Top Level BOM, the base_assy component is broken up into the base and bracket components.
Hierarchical	Allows one to see assembly subcomponents in a single BOM table. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
Configure	Allows to add additional columns to your BOM table: Description, Density, Volume, Mass, Material, Thickness, and Parameters (for parametric components). Prompts you: **Toggle columns** [Descrip-tion/dEnsity/Volume/Mass/parameterS/mAterials/Thickness/Bac <Back>: Choose an option: • Description: adds the Description column. • Density: adds the Density column. • Volume: adds the Volume column. • Mass: adds the Mass column. • Parameters: If several inserts of the same parametric component exist in the assembly they will be grouped in different BOM rows depending on the values of their parameters. • Materials: adds the Material column. • Thickness: adds the Thickness column. If a component is a sheet metal part, this column will show the thickness of this part. Otherwise, it will be empty. • Back: Returns to the previous prompt. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M BOM with Densitiy, Volume and Mass.

Related Commands

Table (6.29.1) - creates a table entity in the drawing through a dialog box. -Table (6.29.2) - creates a table entity in the drawing through the command bar. Tabledit (6.29.3) - Edits texts in a table cell. BmBalloon (6.8.1) - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout. BmExplode (6.8.6) - creates a block with an exploded representation of the current assembly and allows to insert this block at a user defined location. BmMassProp (6.8.14) - computes mass properties for the current model using densities assigned its the components. BmUpdate (6.8.25) - reloads all referenced components from external files and updates BOM tables.

Figure 6.413

Bill of Materials tripod_main_assy		
No.	Component	Quantity
1	tripod_axis_assy	1
2	tripod_foot_assy	3

Figure 6.414

Bill of Materials tripod_main_assy		
No.	Component	Quantity
1	lower_tube_prt	3
2	middle_tube_prt	3
3	tripod_axis_prt	1
4	tripod_bond_prt	3
5	tripod_crosspiece_prt	1
6	tripod_sphere_prt	1
7	upper_tube_prt	3

Figure 6.415

Bill of Materials tripod_main_assy		
No.	Component	Quantity
1	tripod_axis_assy	1
1.1	tripod_axis_prt	1
1.2	tripod_crosspiece_prt	1
1.3	tripod_sphere_prt	1
2	tripod_foot_assy	3
2.1	bond_assy	3
2.1.1	tripod_bond_prt	3
2.2	tubes_assy	3
2.2.1	lower_tube_prt	3
2.2.2	middle_tube_prt	3
2.2.3	upper_tube_prt	3

Figure 6.416

Bill of Materials tripod_main_assy					
No.	Component	Quantity	Density	Volume	Mass
1	tripod_axis_assy	1	8000.00 kg/m³	330.77 cm³	2.65 kg
1.1	tripod_axis_prt	1	8000.00 kg/m³	194.42 cm³	1.56 kg
1.2	tripod_crosspiece_prt	1	8000.00 kg/m³	60.06 cm³	0.48 kg
1.3	tripod_sphere_prt	1	8000.00 kg/m³	76.29 cm³	0.61 kg
2	tripod_foot_assy	3	8000.00 kg/m³	166.24 cm³	1.33 kg
2.1	bond_assy	3	8000.00 kg/m³	21.32 cm³	0.17 kg
2.1.1	tripod_bond_prt	3	8000.00 kg/m³	21.32 cm³	0.17 kg
2.2	tubes_assy	3	8000.00 kg/m³	144.92 cm³	1.16 kg
2.2.1	lower_tube_prt	3	8000.00 kg/m³	62.85 cm³	0.50 kg
2.2.2	middle_tube_prt	3	8000.00 kg/m³	34.83 cm³	0.28 kg
2.2.3	upper_tube_prt	3	8000.00 kg/m³	47.24 cm³	0.38 kg

Figure 6.417



6.8.3 -BmCreateComponent

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Creates a component from a selection set or drawing and adds the component to the library.

Accessing the Command

Keyboard: **-bmcreatecomponent : -bmcreatecomponent Prompts you:** Select entities to create component from or use <Drawing>: **(Select entities or press Enter to select the entire drawing.)** Entities in set: 1 Select entities to create component from or use <Drawing>: **(Select more entities or press Enter to stop selecting.)** Specify component category: [1 - All/ 2 - Building/ ... - 0 - Add new category/] <Current>: **(Type the number of a category or press Enter to add the component to the current category.)** Specify the component name: **(Type a name for the new component.)** Enter component base point :<0, 0, 0>: **(Specify the base point or press Enter to accept the WCS origin.)** Enter thumbnail position [Top Front Left/Top Front Right/Top Back Left/Top Back Right/Top] <Top Front Left>: **(Choose an option.)** Generating thumbnails... done.

Command Options

Option	Description
Select entities	Do one of the following: • Select the entities to be included in the component. Press Enter to stop selecting. • Press Enter at the first prompt to include all entities in the drawing.
Specify component category	The options are: • **Current**: The currently selected category in theComponents Panel (6.9.61). • **1 - All/ 2 - Building/ 3 - Doors/ 4 ...**: All subcategories of the currently selected category are listed. Enter the number of a category. • **0 - Add new category/**: creates a new category as a subcategory of thecurrently selected category in theComponents Panel (6.9.61). Type 0. Prompts you: **Specify new category name: Type a name.**The name is added to the available categories list. Type the number of the newly created category.
Base point	Defines the base point of the component. Do one of the following: • Press Enter to accept the origin of the WCS (0,0,0). • Type the x,y,z coordinates. • Pick a point in the drawing.

Thumbnail position	Defines the view orientation of the thumbnail image. The options are: • Four isometric views:Top Front Left, Top Front Right, Top Back Left and Top Back Right • Top

Related Commands

ComponentsPanelOpen (6.9.61) -Opens the Components panel as a docked panel. ComponentsPanelClose (6.9.38) -Closes the Components panel.

Figure 6.418



6.8.4 BmDependencies

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Lists all files, containing component definitions inserted in the assembly, in the command window.

Accessing the Command

command bar: **bmdependencies** : **bmdependencies** Reports in the command bar: (\vref {fig:C:/Dev/Task-2448/images/staticb

Figure 6.419

```
Found 10 dependencies for this assembly:
C:\MCAD\Lift\axis_b.dwg      - OK
C:\MCAD\Lift\axis_m.dwg      - OK
C:\MCAD\Lift\axis_s.dwg      - OK
C:\MCAD\Lift\bar_D.dwg       - OK
C:\MCAD\Lift\bar_U.dwg       - OK
C:\MCAD\Lift\base.dwg        - OK
C:\MCAD\Lift\base_assy.dwg   - NOT FOUND
C:\MCAD\Lift\bracket.dwg     - OK
C:\MCAD\Lift\connector.dwg   - OK
C:\MCAD\Lift\platform_U.dwg  - NOT FOUND
```

Command Options

No options.

Related Commands

MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser. BmRecover (6.8.20) - recovers a broken mechanical structure. BmUpdate (6.8.25) - reloads all referenced components from external files and updates BOM tables.

Figure 6.420



6.8.5 BmDissolve

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Dissolves a mechanical component inserted in the current drawing.

Accessing the Command


command bar: **bmdissolve** menu bar: **Assembly | Dissolve Component** toolbar: **Assembly |Dissolve Component** ribbon: **Assembly | Modify |Dissolve Component** >Quad cursor menu>: **Model |Dissolve Component** : **bmdissolve Prompts you in the command bar:** Select entities: **(select an entity)** Entities/subentities in set: 1 Select entities: **(select more entities or right click to stop selecting entities.)** Only components at the first level, directly branching from the root component, in the mechanical hierarchy can be dissolved. To dissolve a component at a lower level, its parent component(s) must be dissolved first. All subcomponents of a dissolved component are moved up one level in the mechanical hierarchy. Dissolving a component insert does not affect the source drawing of the component. The selection set must contain component inserts only. If not, a message displays in the command bar: ”No components selected.”

Command Options

No options.

Related Commands

BmExternalize (6.8.8) - converts local components to external components. BmForm (6.8.9) - creates a new mechanical component and inserts it into the current drawing. BmInsert (6.8.12) - inserts an existing mechanical component into the current drawing. BmLocalize (6.8.13) - converts external components to local components. BmUnmech (6.8.24) - converts the current mechanical component into a plain drawing.

Figure 6.421



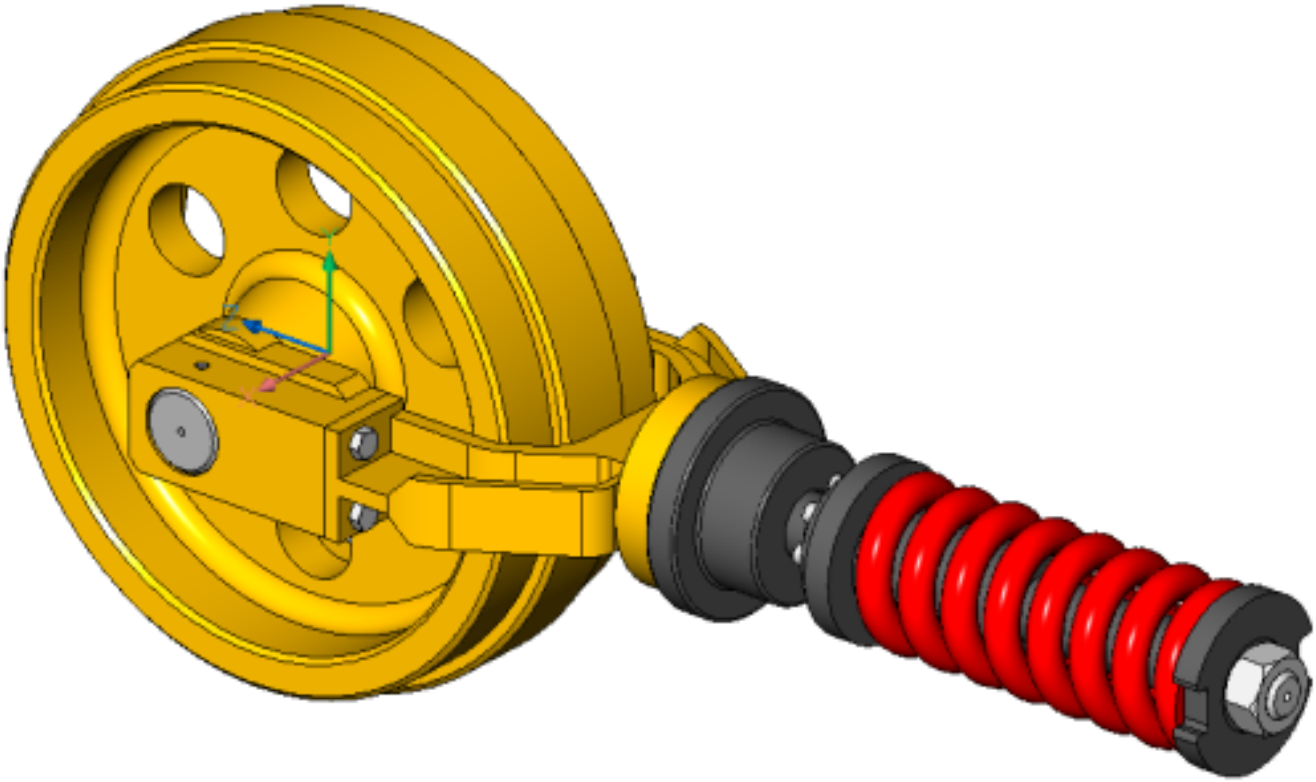
6.8.6 BmExplode

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Creates a block with an exploded representation of the current assembly and allows to insert this block at a user defined location. It supports two levels of the exploded representation: top and bottom. Top level means that the assembly will be exploded down to top level components; bottom level means that the assembly will be exploded down to bottom level components. The steps in creating an exploded representation are automatically completed, and are displayed in the Mechanical Browser (3.2.2) for adding, deleting, and reordering. It's also possible to animate a selected step of an explosion or the entire sequence.

Accessing the Command

 command bar: **bmexplode** menu bar: **Assembly | Explode** toolbar: **Assembly |Explode** ribbon: **Assembly | Tools |Explode** : **bmexplode Prompts you in the command bar:** Select explosion algorithm [Table by types/Linear/Manual/Setti <Manual>: **(Choose the algorithm or set the necessary options)** Select exploded view behaviour [Edit/Generate drawing views/Finish] <Finish>: **(Choose to further action)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

Figure 6.422



Command Options

Option	Description
Table by Types	Selects the Table by Types. The Table by Types algorithm groups same parts on the same row and different parts on different rows. prompts you: **Select exploded view behaviour** [Edit/Generate drawing views/Finish] <Finish>:- **choosean optionor press Enter**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
Linear	Allows to automatically create exploded representations of assemblies in a given direction, taking into account possible physical collisions between components. prompts you: **Select axial entity or define axis by** [2Points/Xaxis/Yaxis/Zaxis] <2Points>: -**specify movement direction by using coordinate axis, linear edge or two arbitrary points** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
Manual	Selects the manual mode, which creates the exact copy of the assembly ready for further editing in order to create a customized exploded view. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M prompts you: **Select exploded view behaviour** [Edit/Generate drawing views/Finish] <Finish>: -**choosean optionor press Enter.**

Settings	Allows to access the command settings; prompts you: **Select explode level or set block name** [Top/Bottom/set Name/BAck] - **choose an option**. • Top: If selected, the assembly will be exploded using only top level components. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • Bottom: If selected, the assembly will be exploded down to bottom level components. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • set Name: Allows to set the name for the exploded representation; prompts you: **Name for exploded block:** - **Type the name of the exploded representation.**If a such name is already assigned to another exploded representation, a postfix to make it unique will be added, starting from '_2.' • BAck: Returns to the previous prompt.
Edit	Opens the exploded representation for editing. prompt you: **-bedit,Block name or ? to list existing blocks:** Manual_Top_2 **Use BCLOSE or the Block Edit toolbar to end the block edit session.**
Generate drawing views	Allows to generate drafting views from exploded representation. prompt you: **Enter new or existing layout name to make current** <Layout1>: **-Enter theLayout nameof the layout for the exploded representation**
Finish	Concludes the command

Procedures

Levels of the exploded representation (3.6.7) Selecting an algorithms (3.6.7) To create an exploded representation (3.6.7) To generate drawing views of the exploded representation (3.6.7) To update an exploded representation (3.6.7)

Related Commands

BmBalloon (6.8.1) - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout. BmBom (6.8.2) - inserts a Bill of Materials (BOM) table in the current drawing. BmExplodeMove (6.8.7) - creates a block with an exploded representation of the current assembly and allows to insert this block at a user defined location. Bedit (6.6.7)-opens the exploded representation for editing.

Figure 6.423

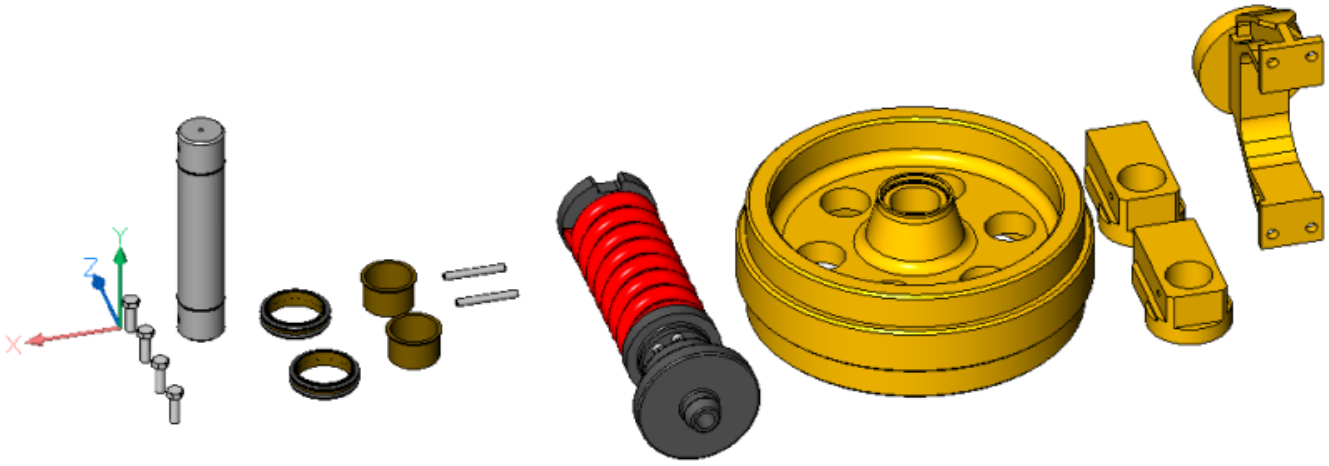


Figure 6.424

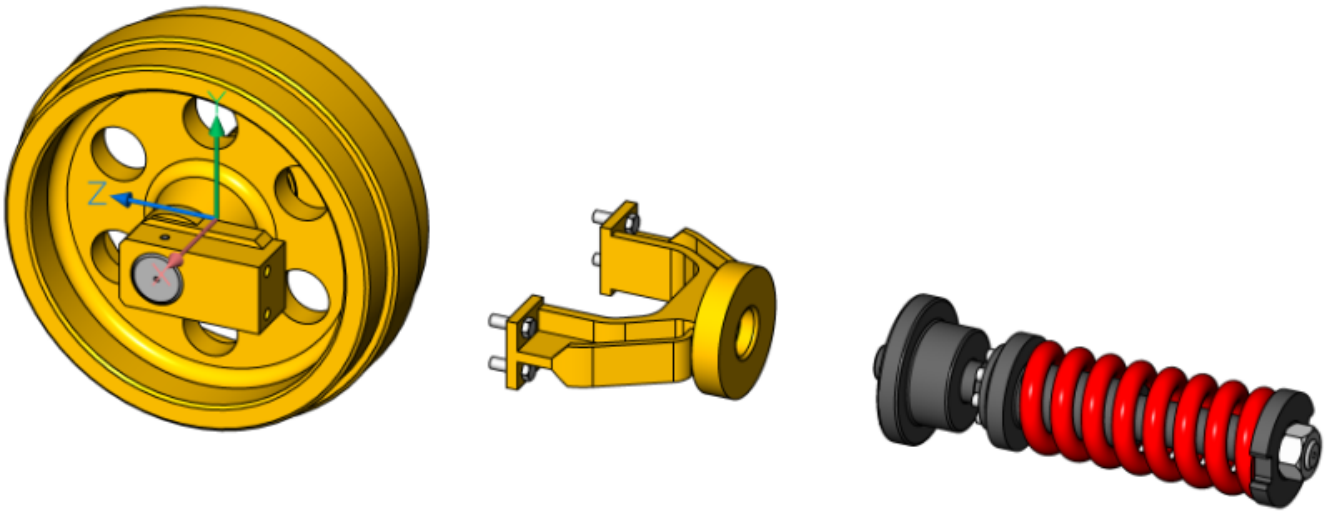


Figure 6.425

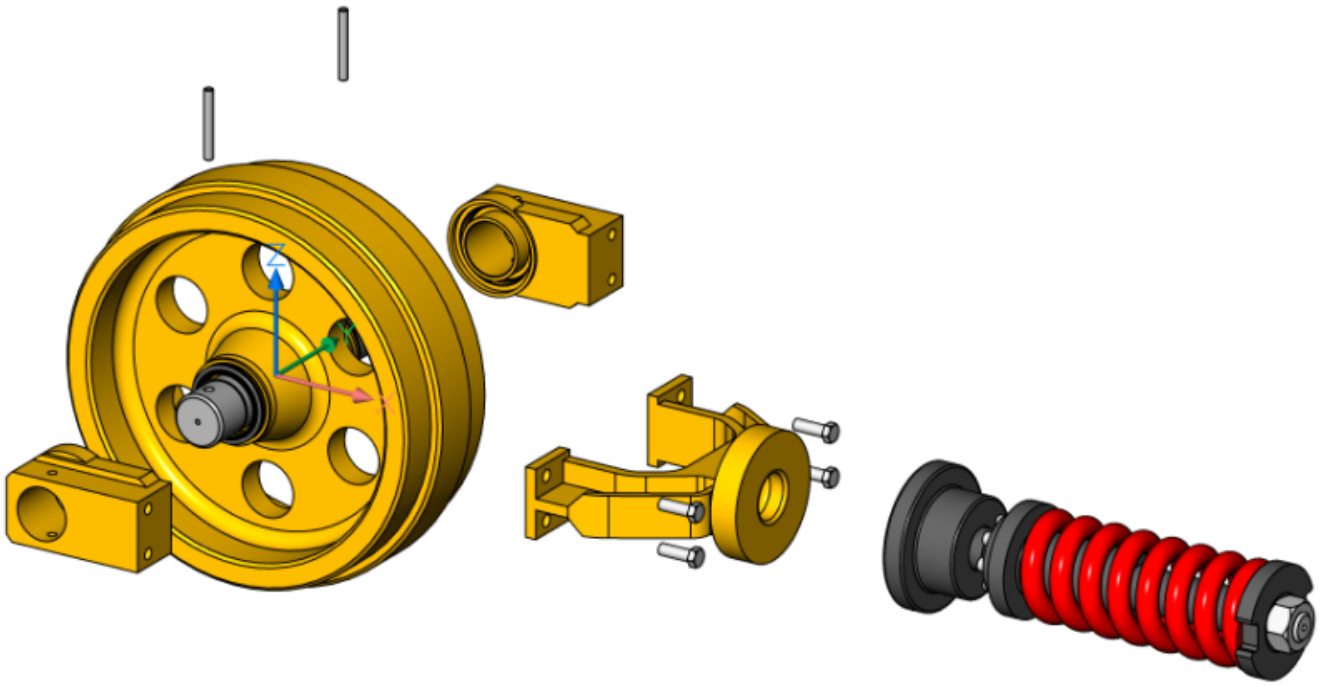


Figure 6.426

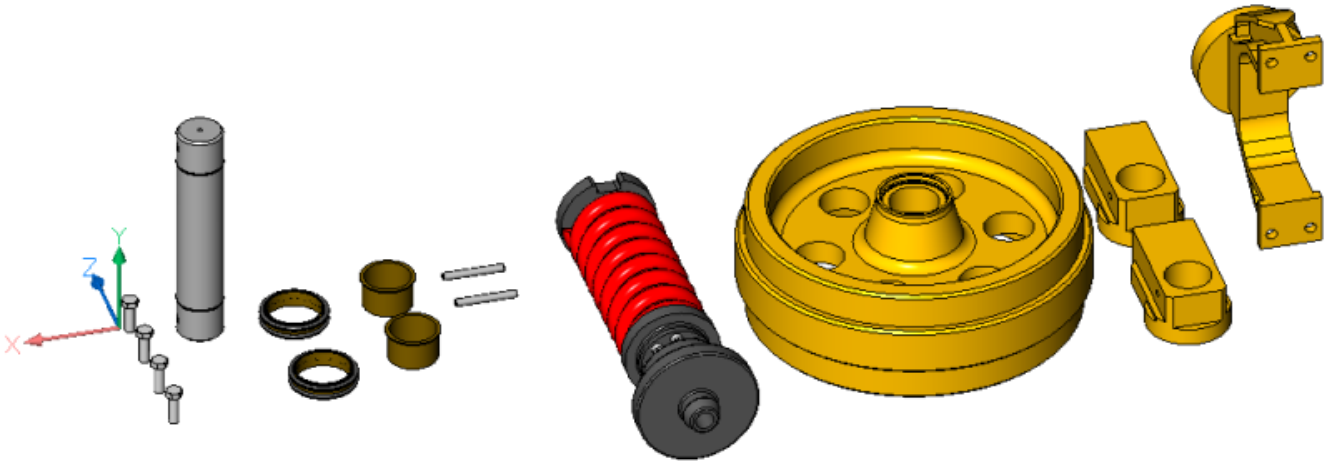


Figure 6.427



Figure 6.428



6.8.7 BmExplodeMove

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Moves selected parts to form an exploded representation in a given direction, taking into account possible physical collisions between components (similar toLinear option ofBMEXPLODEcommand).

Accessing the Command


command bar: **bmexplodemove** menu bar: **Assembly | Assembly Explode | Explode Move** toolbar: **Assembly |Explode Move** ribbon: **Assembly | Tools |Explode Move** :**bmexplodemove Prompts you in the command bar:** Select base part <None>: **(Select the base component which does not move during explosion process)** Select axial entity or define axis by [2Points/Xaxis/Yaxis/Zaxis] <2Points>: (Specify movement direction by using coordinate axis, linear edge or two arbitrary points)

Procedures

- Levels of the exploded representation (3.6.7)
- Selecting an algorithms (3.6.7)
- To create an exploded representation (3.6.7)
- To generate drawing views of the exploded representation (3.6.7)
- To update an exploded representation (3.6.7)

Related Commands

BmBalloon (6.8.1)- creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout. BmBom (6.8.2)- inserts a Bill of Materials (BOM) table in the current drawing. BmExplode (6.8.6) - creates a block with an exploded representation of the current assembly and allows to insert this block at a user defined location.

Figure 6.429



6.8.8 BmExternalize

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Converts local components to external components.

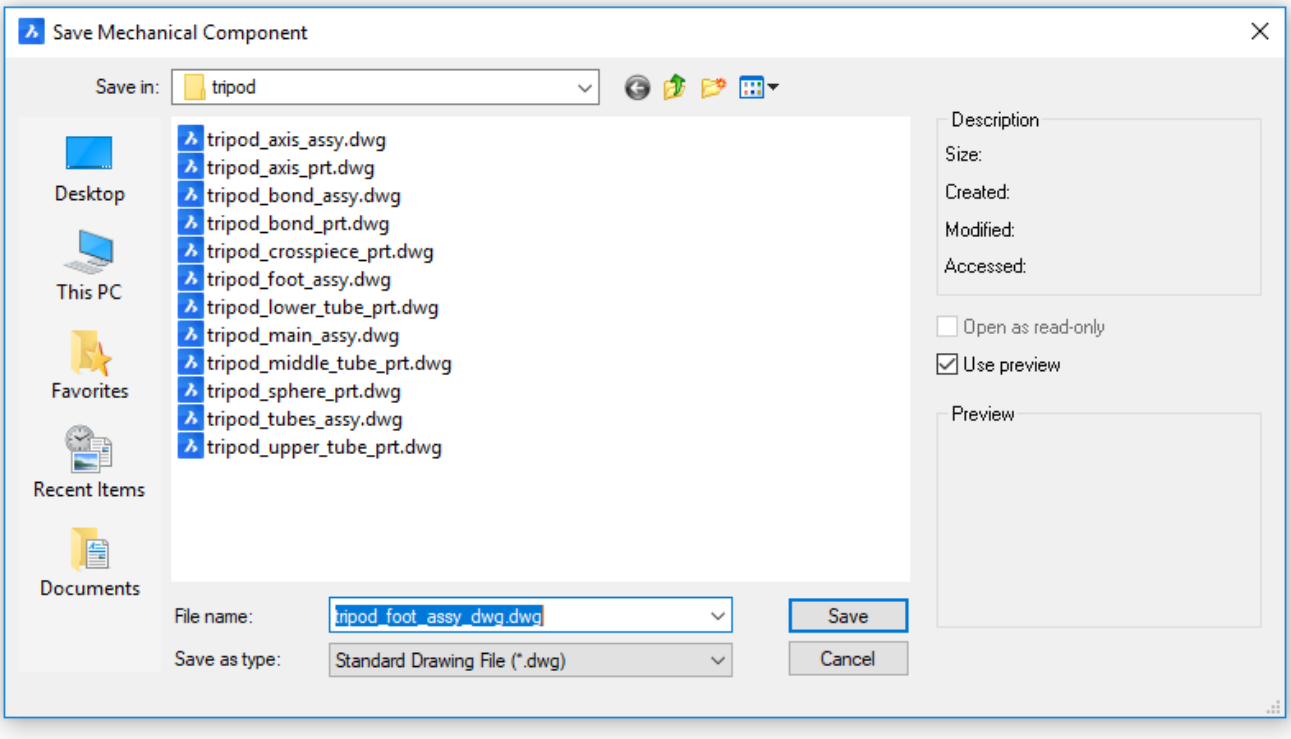
Accessing the Command

 command bar: **bmexternalize** menu bar: **Assembly | Switch Component to External** toolbar:Assembly |Switch Component to External >ribbon: **Assembly | Modify |Switch Component to External** Quad> cursor menu>: >**Model |Switch Component to External>** : **bmexternalize Prompts you in the command bar:** Select component instances [ASk for file names/Entire model] <Entire model>: **(Select a local component.)** Entities in set: 1 Select component instances [ASk for file names/Entire model] <Entire model>: **(Select a local component or press Enter.)** A file dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Mechanical/ART
Do one of the following:

- Press the **Save** button to accept the default name, which is the name of the selected local component.
- Type a name in the **File Name** field, then click the **Save** button.

This procedure is repeated for each of the selected local components.

Figure 6.430



Command Options

Option	Description
Entire Model	For each of the local components in the assembly a file dialog displays. Do one of the following: • Press the Save button to accept the default name, which is the name of the local component. • Type a name in the File Name field, then click the Save button.
Ask for file names	You can select to ask for file names of externalized components or generate them automatically. Prompts you: **Ask for file names of external components to be saved?** [Yes/No] <Yes>: • Yes: You are prompted to enter file names. • No: File names are generated automatically. Please note that some symbols used in component names cannot be used in file names. Such symbols will be automatically replaced with an underscore (_).

Procedures

Working with components and component inserts (1.22.22)

Related Commands

BmDissolve (6.8.5) - dissolves a mechanical component inserted in the current drawing. BmLocalize (6.8.13) - converts external components to local components. BmReplace (6.8.21) - replaces a component insert.

Figure 6.431



6.8.9 BmForm

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Creates a new mechanical component and inserts it into the current drawing. If necessary, run the BmMech (6.8.15) command to initialize the mechanical structure in the current drawing.

Accessing the Command


command bar: **bmform** menu bar: **Assembly | Form Component** toolbar:Assembly |Form Component ribbon: **Assembly | Modify | Form Component** Quad cursor menu: **Model | Form Component** : **bmform Prompts you in the command bar:** Select entities <Empty component>: **(Select a 3D solid.)** Entities in set: 1 If a history based solid is selected, prompts you: Selection set contains 3D solid/solids with recorded history. Confirm selection and purge history? <Yes>/No: **(Press Enter to accept the default option to include the X-Solid in a mechanical component.)** Select entities: **(Select a 3D solid or press Enter to stop selecting 3D solids.)** Enter external component base point or switch component type to [Local/External] <0,0,0>: **(Specify a point.)** Name for new mechanical component <Component_1>: **(Specify a name, or press Enter to accept the default name.)** A new internal block, containing the selected solid(s), is created and inserted as an internal reference in the current drawing. The selected solid(s) are removed from the current drawing.

Command Options

Option	Description
Create External component	Creates a new drawing, containing the selected solid(s). The selected solid(s) are removed from the current drawing. The new drawing is inserted as an external reference in the current drawing. Displays a File dialog box: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360018718 Select a folder, type a name in the File Name field, then press the Save button.

Related Commands

BmDissolve (6.8.5) - dissolves a mechanical component inserted in the current drawing. BmExternalize (6.8.8) - switches internal components to external components. BmInsert (6.8.12) - inserts an existing mechanical component into the current drawing. BmLocalize (6.8.13) - switches external components to internal components. bmXConvert (6.8.27) - converts X-Hardware solids in the current drawing to mechanical components.

Figure 6.432

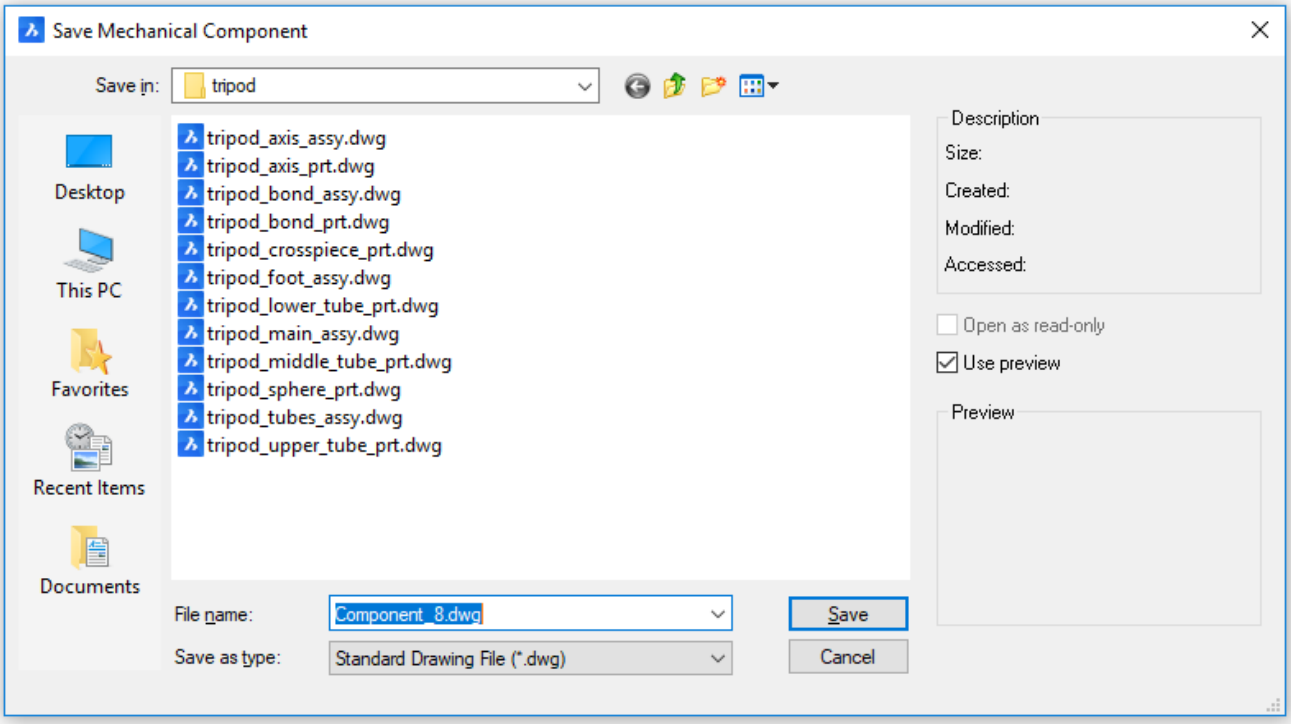

Save Mechanical Component
Save in:
tripod
Desktop
This PC
Favorites
Recent Items
Documents
tripod_axis_assy.dwg
tripod_axis_prt.dwg
tripod_bond_assy.dwg
tripod_bond_prt.dwg
tripod_crosspiece_prt.dwg
tripod_foot_assy.dwg
tripod_lower_tube_prt.dwg
tripod_main_assy.dwg
tripod_middle_tube_prt.dwg
tripod_sphere_prt.dwg
tripod_tubes_assy.dwg
tripod_upper_tube_prt.dwg
Description
Size:
Created:
Modified:
Accessed:
Open as read-only
Use preview
Preview
File name:
Component_8.dwg
Save as type:
Standard Drawing File (*.dwg)
Save
Cancel

Figure 6.433



6.8.10 -BmHardware

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Inserts a standard hardware part as a mechanical component in the current drawing. Prompts you in the command bar. If necessary, run the BmMech (6.8.15) command to initialize the mechanical structure in the current drawing.

NOTE

- This command is obsolete and creates non-parametric standard hardware parts. To insert parametric standard hardware, use theComponents panel (6.9.61), which contains parametricStandard Mechanical Components (3.6.9).

Accessing the command

command bar: **-bmhardware** : **-bmhardware Prompts you in the command bar:** Input a category (? to list available categories) <ALL>: **(Type a category name.)** Input a type (? to list available types) <ACORN-RVT>: **(Specify a type.)** Input a size (? to list available sizes) <.500>: **(Specify the size.)** Input a length (? to list available lengths) <1.0>: **(Specify the length.)** Insertion point [Rotate/Base point/Name/mUltiple] <0,0,0>: **(Specify a point in the drawing.)**

Command Options

Option	Description
Rotate	Allows to specify a rotation angle. Prompts you: **Rotation angle** [Move/Base point/Name/mUltiple]: **Type a value or choose an option.**
Base point	Allows to specify a new base point for the insert. Prompts you: **New base point** <0,0,0>: **Specify a point.**
Name	Allows to rename the insert. Prompts you: **Component insert name** <current_name >: **Type a name.**
Multiple	Allows to insert multiple copies. Prompts you for each copy: **Insertion point** [Rotate/Base point/Name] <0,0,0>: **Specify a point.** Press Enter to stop inserting more copies.

Related Commands

ComponentsPanelOpen (6.9.61)- opens the Components panel, allowing to insert standard parts, items and components supplied with BricsCAD as well as components from the user's libraries.

Figure 6.434



6.8.11 BmHide

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Hides a mechanical component inserted in the current drawing. The command affects the visibility of the selected inserts only. Hidden inserts will still be taken into account by commands such as BmBom, BmMassProp etc.

Accessing the Command

 command bar: **bmhide** menu bar: **Assembly Visualization | Hide** toolbar: **Assembly Visualization |Hide** ribbon: **Assembly l Modify | Hide** Quad cursor menu: **Model | Hide** : **bmhide Prompts you in the command bar:** Select insert or enter an option [Component name/Insert name]<Insert name>: **(Select an insert in the drawing or choose an option.)**

Command Options

Option	Description
Component name	Prompts you: **Component name**, all its inserts will be hidden: **Type the component name of the component insert(s) you want to hide.** All inserts of the specified component will be hidden.
Insert name	Prompts you: **Component insert name**, use / to separate names of nested inserts: **Type the insert name of the component insert you want to hide.** If you want to hide a nested component insert (subcomponent insert) first type the insert name of the parent component insert, then the component insert name separated by a slash (/)

Related Commands

MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser. BmShow (6.8.22) - shows a previously hidden mechanical component or subcomponent in the current drawing. BmVStyle (6.8.26) - applies a visual style to a mechanical component.

Figure 6.435



6.8.12 BmInsert

(\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/360030972414/mceclip0.png}) Inserts a sheet metal form feature, existingmechanical and BIM component. Inserts a RFA* file as a BIM component into the current drawing. The source drawing of the component is not affected. A virtual component is created, representing the drawing in the assembly. * RFA import is a work-in-progress. Some general guidelines about the current V19 RFA import functionality are:

1. We don't support RFA files older than 2011.
2. Files from 2011 to 2014 get imported, but since there was a major format change from 2014 to 2015, there might be issues with nested families.
3. Files from 2015 to 2018 get imported correctly. At the moment, we import extrusions, blends, revolutions, and sweeps.
4. Only the geometry of the family is imported, no parameters nor properties.
5. We are working on RFA 2019 support.

Accessing the Command

 **Insert Component**  **Insert Form Feature** command bar: **bminsert** menu bar:

- **Assembly | Insert Component**
- **Sheet Metal | Insert Form Feature**

toolbar:

- **Assembly |Insert Component**
- **Sheet Metal | Insert Form Feature**

ribbon:

- **Assembly | Modify | Insert Component**
- **Sheet Metal | Modify | Insert Form Feature**

: **bminsert** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Me
Select a mechanical component drawing, then double click or press the **Open** button. The component is attached to the cursor with its origin point. Dynamic dimensions display X and Y coordinates from the origin of the current coordinate system. For Windows and Doors components the command displays four dynamic dimensions similar to BIMINSERT (6.7.14). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Mechanical/AR
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Mechanical/ART/InsertComponentC
Prompts you in the command bar: **Select insertion point or [Edit inserted entity/Rotate component/set Base point/Name/insertion Type/Flip/mUltiple] <0, 0, 0>:(Specify the insertion point or choose an option.)**

Figure 6.436

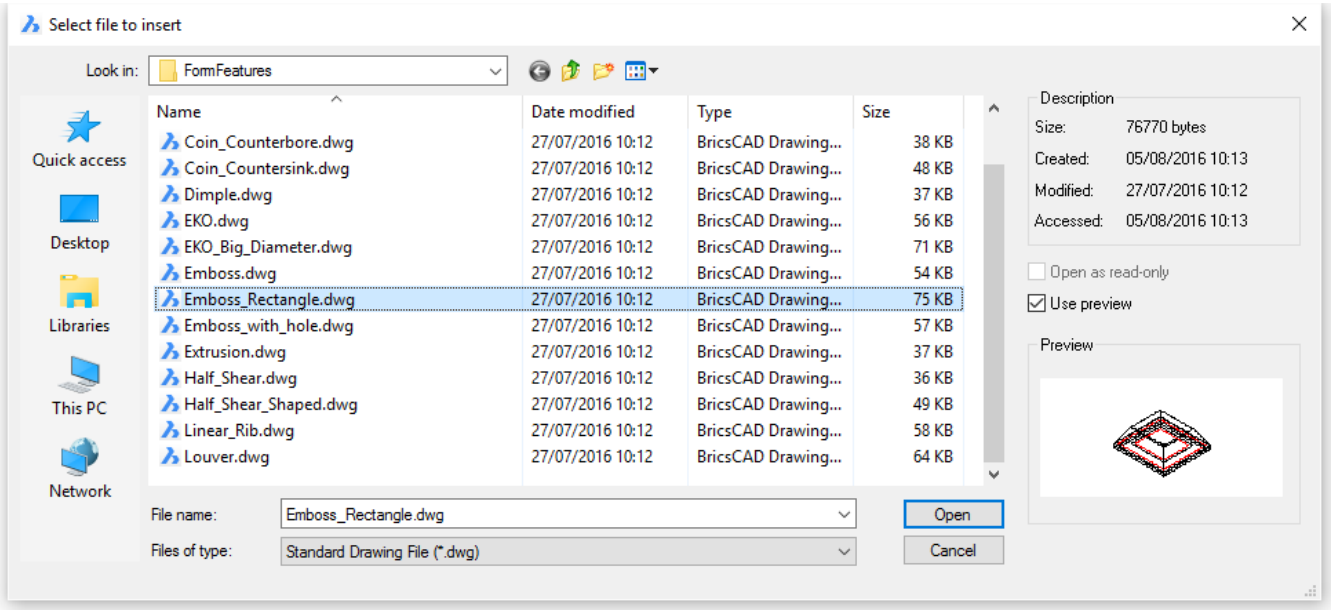


Figure 6.437

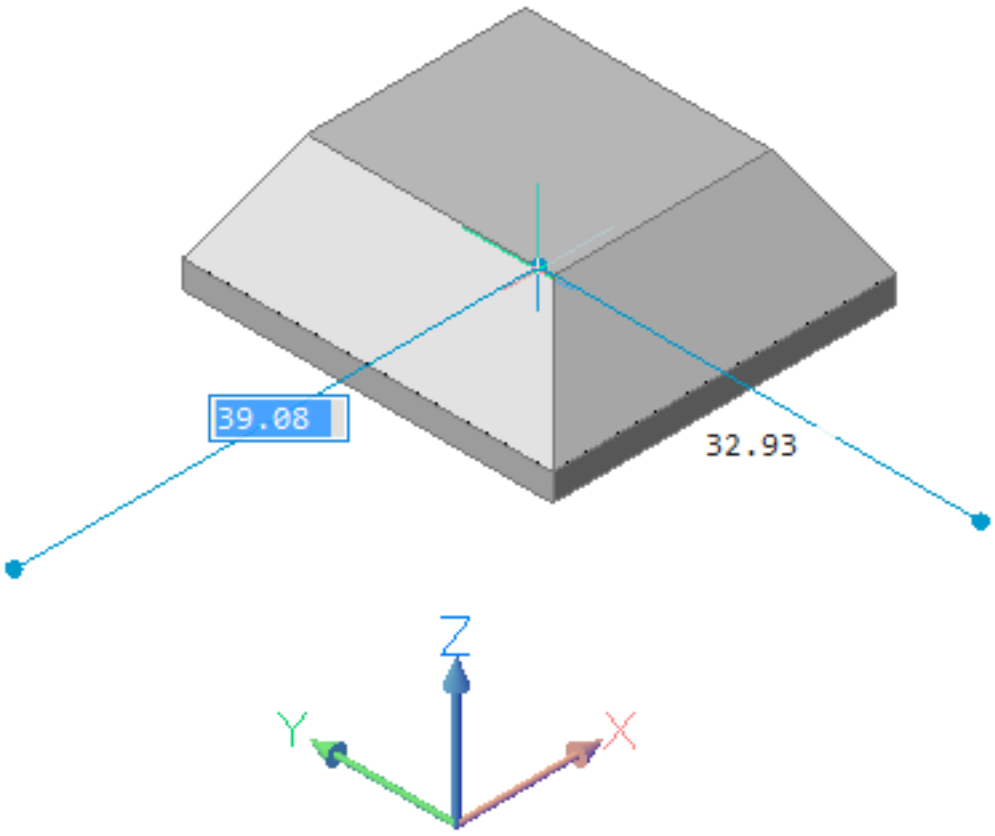
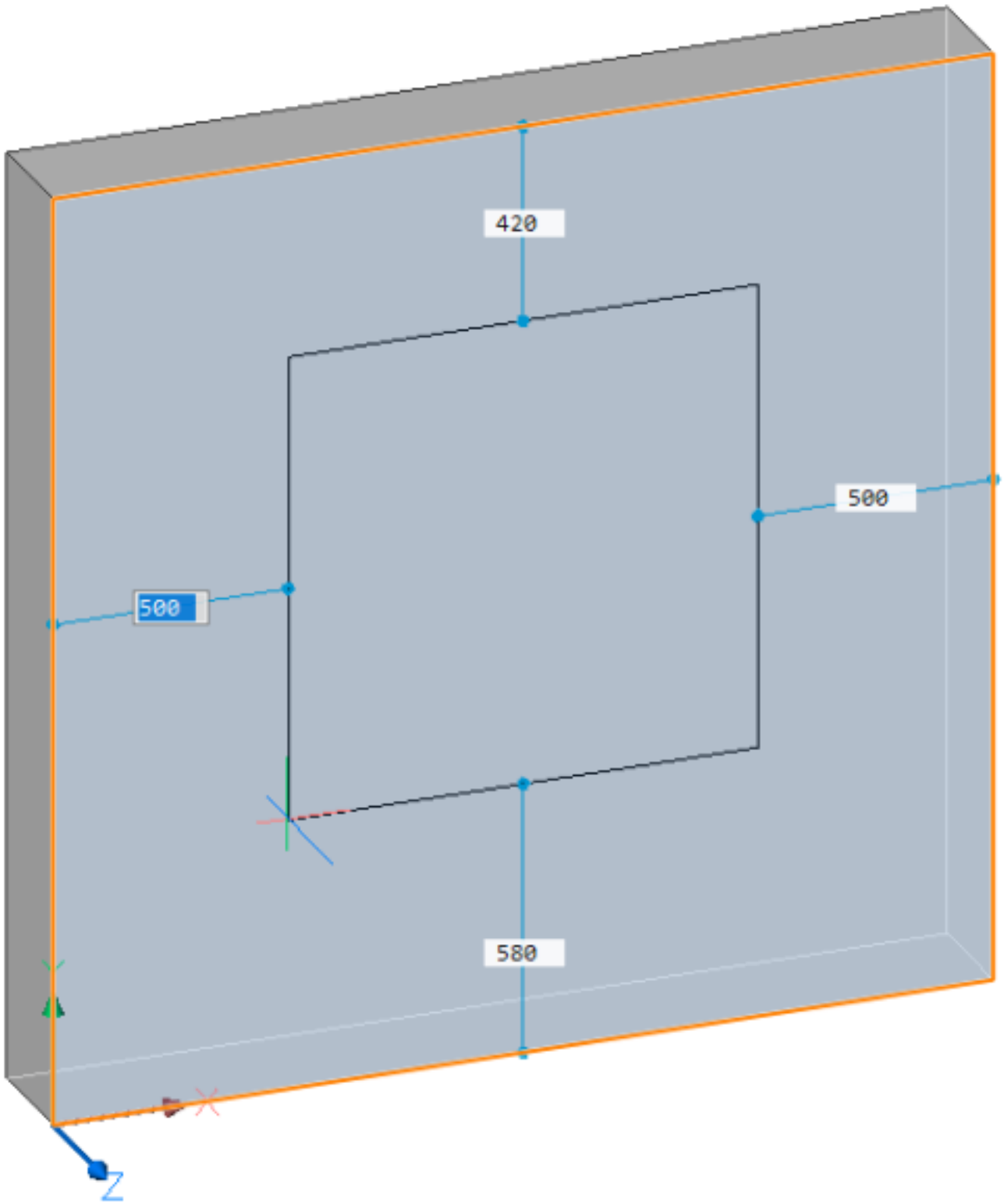


Figure 6.438



Command Options

Option	Description
Edit inserted entity	Allows to edit the parameters of the selected insert. Prompts you: **Enter the parameter name or press ENTER to finish: Press F2 to open the Prompt History (6.29.13) window.** (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ Type the name of a parameter. Prompts you: **Enter expression < 30.00 >:**- **Type a value, then press Enter.** Prompts you: **Enter the parameter name or press ENTER to finish:** - **Type the name of a parameter or press Enter.** The insert is updated. To edit the parameters of the selected insert user can useInserted ComponentParameterssettings group of Properties Panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N
Rotate component	Allows to define the rotation angle of the component. Prompts you: **Specify rotation angle of the insert:** - **Enter the rotation angle.** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N
set Base point	Allows to specify a different base point to insert the component. Prompts you: **New base point <0,0,0>:** - **Specify a new base point.** The new base point is attached to the cursor.
Name	Allows to define the insert name of the component. Prompts you: **Component insert name <DefaultName>:** - **Type the insert name for the component.**
insertion type	Allows to specify the insertion type: Local or External Prompts you: **Specify component type [Local/External] <External>:** - **Choose an option.**
Flip	Inserts the selected insert in the opposite face of the solid.
Multiple	Allows to insert multiple inserts of the same component or create an array. Prompts you: **Select insertion point or [Edit inserted entity/Rotate/Base point/Name/Insertion Type/Flip/Array]:** Optionally, choose an option, then do one of the following: • Select multiple insertion points. Press Enter to stop inserting more copies. • Array - creates an associative array of components. Prompts you: **Specify base point for the array:** - Specify a point. **Enter columns spacing or set [Direction]:** - Specify a distance or choose Direction. **Enter the distance between rows or set [Single row/Rectangular/Direction]:** - Specify a distance or choose an option. **Select an end point for creating the array:** - Move the cursor to dynamically define the extents of the array, then click a point. **Accept resulting array or edit [Base point/Columns/Rows] <Accept>:** - Press Enter to accept the array or choose an option.

Figure 6.439

```
--------------------------------------------------------------------------------
Parameter: _SM_THICKNESS Expression:                          Value: 2.00 mm
Parameter: A             Expression:                         Value: 30.00
Parameter: L             Expression:                         Value: 16.00 mm
Parameter: W             Expression:                         Value: 14.00 mm
Parameter: H             Expression:                         Value: 8.00 mm
--------------------------------------------------------------------------------
```

Figure 6.440

Inserted Component Properties	
Name	*Emboss Rectangle:2
File	Emboss_Rectangle.dwg
_SM_THICKNESS	2 mm
A	30
L	16 mm
W	14 mm
H	8 mm

Related Commands

BmDissolve (6.8.5) - dissolves a mechanical component inserted in the current drawing. BmForm (6.8.9) - creates a new mechanical component and inserts it into the current drawing. ComponentsPanelOpen (6.9.61) -opens the Components panel as a docked panel. BmMech (6.8.15) - converts the current drawing into a mechanical main component. BmUnlink (6.8.23) - breaks the connection between a component insert (e.g. a Window) and a 3D solid (e.g. a Wall) retaining the opening. DmCopyFaces (6.12.8) - copies a set of 3D solid faces forming a feature (e.g. hole, rib or protrusion) to a new position on the same or another 3D solid. SmReplace (6.28.25) - replaces form features (including recognized ones) in sheet metal parts with form features from the built-in or user library. SmRibCreate (6.28.26) - creates associative Rib form features on sheet metal parts from 2D profiles.

Figure 6.441

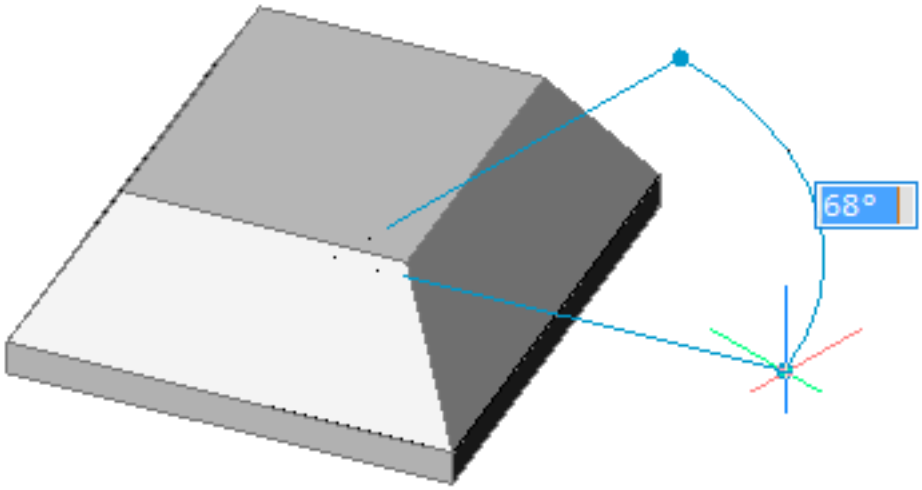


Figure 6.442



6.8.13 BmLocalize

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Converts external components to local components.

Accessing the Command

command bar: **bmlocalize** menu bar: **Assembly | Switch Component to Local** toolbar:**Assembly | Switch Component to Local** ribbon: **Assembly | Modify | Switch Component to Local** Quad cursor menu: **Model | Switch Component to Local**: **bmlocalize Prompts you in the command bar:** Select component instances/<Entire model>: **(Select an external component.)** Entities in set: 1 Select component instances/<Entire model>: **(Select an external component or press Enter to convert the selected component(s) to local components.)** If multiple inserts of the same external component exist in the model, all inserts are converted to local components.

Command Options

Option	Description
Entire model	Converts all external components in the model to local components.

Procedures

Working with components and component inserts (1.22.22)

Related Commands

BmDissolve (6.8.5) - dissolves a mechanical component inserted in the current drawing. BmExternalize (6.8.8) - converts local components to external components. BmReplace (6.8.21) - replaces a component insert.

Figure 6.443



6.8.14 BmMassProp

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical-bim.png}) Computes mass properties for the current model using densities of materials assigned to its components. The density values used for the mass property calculations are taken from materials defined by the **Material** property of the components and subcomponents, which can be either inherited from the main component of the assembly or set explicitly. The density of the main component is defined by the materials set in the **Material** field in the Mechanical Browser (6.8.29) dialog. If a material does not have an assigned density, or its density is less or equal to zero, an error message displays, and the mass properties will not be computed. All materials involved in the assembly must have non-zero non-negative densities.

Accessing the Command


command bar: **bmmassprop** menu bar: **Assembly | Mass Properties** toolbar:**Assembly | Mass Properties** ribbon: **Assembly | Inquire | Mass Properties** : **bmmassprop Prompts you in the command bar:** Select 3D solids and subcomponents or [Entire model] <Entire model>: **(Select an item.)** Entities in set: 1 Select 3D solids and subcomponents or [Entire model] <Entire model>: **(Select more items or right click.)** Reports in the Prompt History window: ---------- Component ---------- Mass: 17.61 kg Volume: 6522.94 cm³ Centroid: X= 22.59 mm Y= 235.71 mm Z= 201.09 mm Moments of inertia: X= 2.30 kg·m² Y= 0.95 kg·m² Z= 1.39 kg·m² Products of inertia: XY: 0.09 kg·m² YZ: 0.87 kg·m² ZX: 0.11 kg·m² Radii of gyration: X= 361.23 mm Y= 231.76 mm Z= 281.19 mm Principal moments and X-Y-Z directions about centroid: I: 0.22 kg·m² along X= -0.0022 Y= 0.9819 Z= 0.1891 J: 0.41 kg·m² along X= 0.1346 Y= -0.1871 Z= 0.9731 K: 0.61 kg·m² along X= 0.9909 Y= 0.0276 Z= -0.1318 Keep UCS aligned with principal moment axes? [Yes/No] <No>: (Press Enter or right click to keep the current UCS)

NOTE	BmMassProp will format the values of mass, volume and linear nature with their corresponding units only if the PROPUNITS (1.13.11) system variable has the corresponding bits set to 1, or the corresponding flag in the Settings dialog (1.13.11) for the Property units field is checked. See To set the Mass Units (3.6.8).

Command Options

Option	**Description**
Keep UCS aligned with principal moment axes? Yes / No	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N Yes: Aligns the UCS with the principal moment axes. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N No: Keeps the current UCS.

Related Commands

BmBom (6.8.2) - inserts a Bill of Materials (BOM) table in the current drawing. MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser.

Figure 6.444

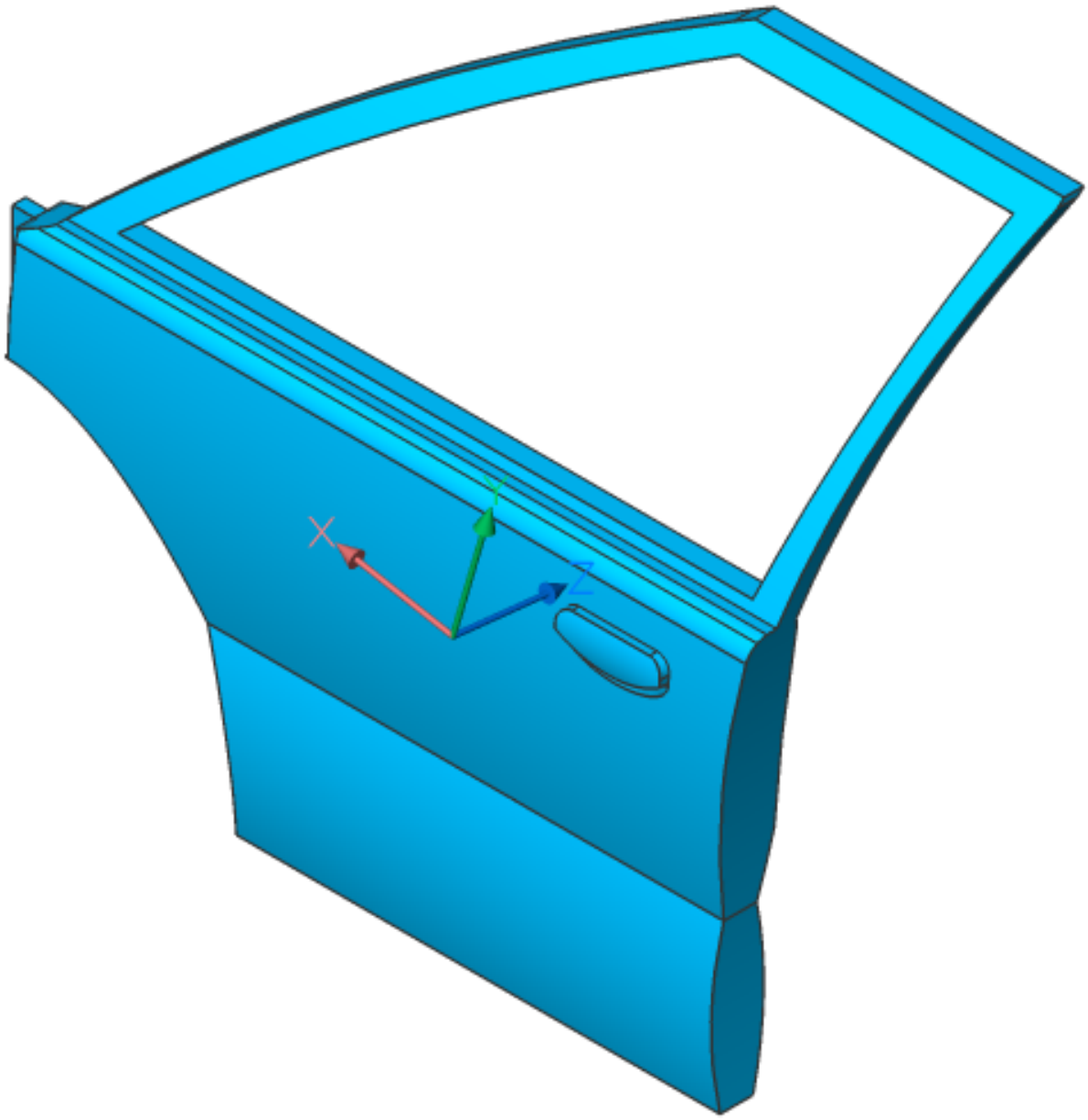


Figure 6.445

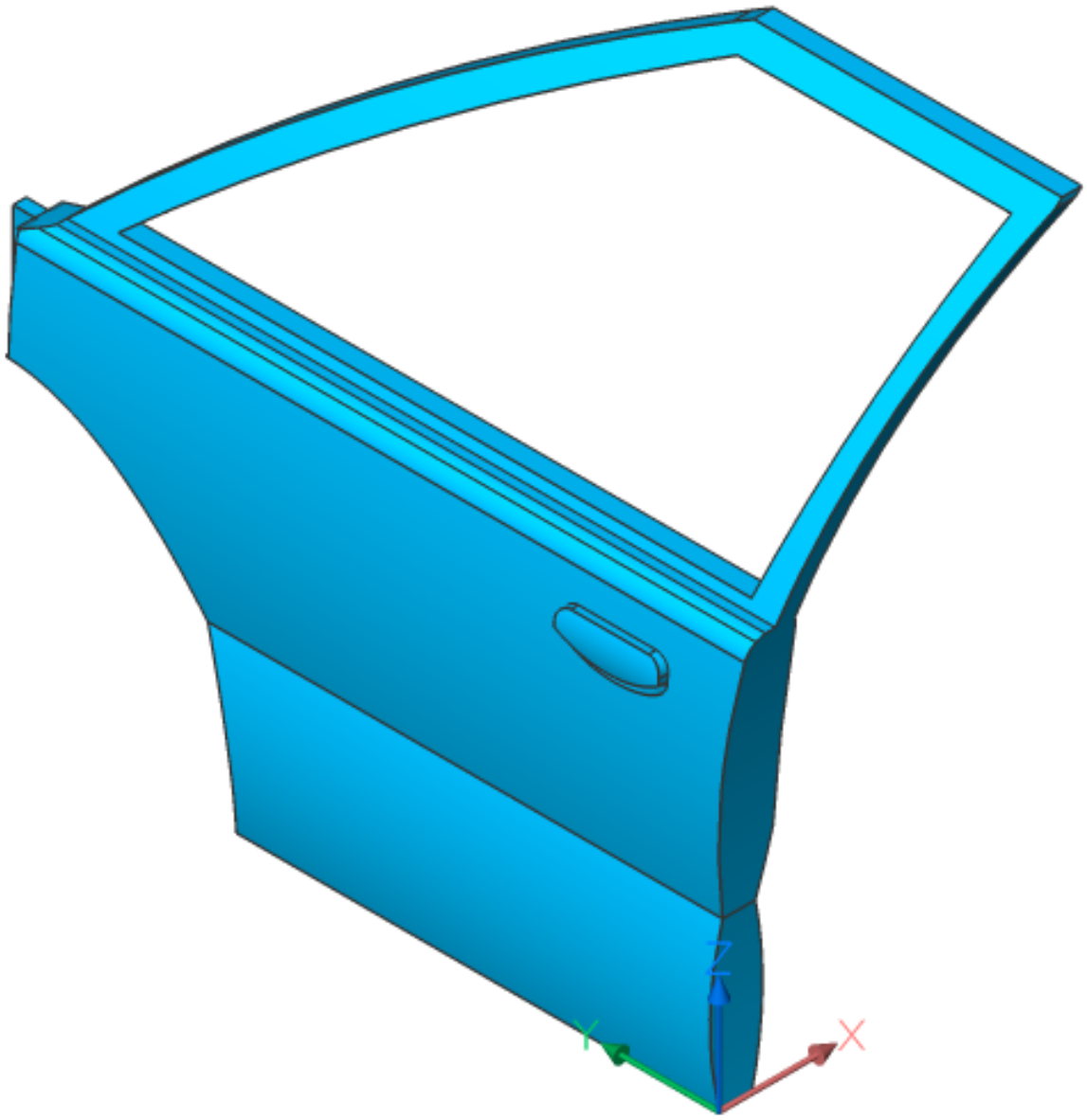


Figure 6.446



6.8.15 BmMech

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Converts the current drawing into a mechanical component. If the drawing contains block references or external references, they can be optionally converted into inserts of local and external components correspondingly. The command does nothing if the drawing is a mechanical component already.

Accessing the Command

 command bar: **bmmech** menu bar: **Assembly | Initialize Mechanical Structure** toolbar:Mechanical | Initialize Mechanical Structure ribbon: **Assembly | Create | Initialize Mechanical Structure** : **bmmech Prompts you in the command bar:** Name for new mechanical component <CurrentDrawingName>: **(Press Enter to accept the current name or type a new name if you want to keep the original. If the drawing was not yet saved before, a default name, Component_1, is suggested.)** Convert blocks and external references to mechanical components? [No/Yes] <Yes>: **(Press Enter or choose an option.)**

Command Options

Option	Description
Yes	If the drawing contains block references or external references, blocks are converted to internal components, external references are converted to external components. Blocks are no longer available in the INSERT (6.17.13) command. Use BMINSERT (6.8.12) to insert them as internal components.
No	The mechanical structure is initialized, but the blocks and external references are not converted.

Related Commands

BmExternalize (6.8.8) - converts local components to external components. BmInsert (6.8.12) - inserts an existing mechanical component into the current drawing. BmLocalize (6.8.13) - converts external components to local components. BmNew (6.8.16) - creates a new mechanical component as a new drawing file. BmUnmech (6.8.24) - converts the current mechanical component into a plain drawing.

Figure 6.447



6.8.16 BmNew

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Creates a mechanical component as a new drawing file.

Accessing the Command

 command bar: **bmnew** menu bar: **Assembly | New Component** toolbar:**Assembly |New Component** ribbon: **Assembly | Create |New Component** : **bmnew Prompts you in the command bar:** Name for new mechanical component <Component_x>: **(Type a name for the new component or press Enter to accept the default name.)**

Command Options

No options.

Related Commands

BmMech (6.8.15) - converts the current drawing into a mechanical main component. BmUnmech (6.8.24) - converts the current mechanical component into a plain drawing.

Figure 6.448



6.8.17 BmOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Opens the source drawing of an external mechanical component.

Accessing the Command


command bar: **bmopen** menu bar: **Assembly | Open Component File** toolbar:**Assembly | Open Component File** ribbon: **Assembly | Modify | Open Component File** Quad cursor menu: **Model | Open Component File** : **bmopen Prompts you in the command bar:** Select insert: **(Select an external component in the model or the Mechanical Browser (3.2.2).)**

Command Options

No options.

Related Commands

BmOpenCopy (6.8.18) - opens a copy of a component insert as a new drawing. XOpen (6.34.6) - opens an externally-referenced drawing (Xref) for editing in a separate window.

Figure 6.449



6.8.18 BmOpenCopy

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Opens a copy of a component insert as a new drawing. The command can be used to create a drawing of a local component. You can then use the BmReplace (6.8.21) command to replace the original local component with the newly created drawing. If you open a copy of an insert of a parametric component the current values of the various parameters are applied to the copy.

Accessing the Command


command bar: **bmopencopy** menu bar: **Assembly | Open Component Copy** toolbar:**Assembly | Open Component Copy** ribbon: **Assembly | Modify | Open Component Copy** Quad >cursor menu: **Model | Open Component Copy** : **bmopencopy Prompts you in the command bar:** Select insert: **(Select an external component in the model or the Mechanical Browser (3.2.2).)** A new drawing: <component_name>1 is created. Use the SAVE (6.27.1) command to rename and save the drawing.

Command Options

No options.

Related Commands

BmOpen (6.8.17) - opens the source drawing of an external mechanical component.

Figure 6.450



6.8.19 -BmParameters

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical-bim.png}) Allows to list and edit the parameters of inserted components and associative array. Parameters of components nested inside an array can be changed or assigned to an expression via the -BMPARAMETERS command.

Accessing the Command

command bar: **-bmparameters** : **-bmparameters Prompts you in the command bar:** Select component instances or arrays: **(Select a component or an associative array.)** Entities in set 1 Select component instances or arrays: **(Select more components or associative arrays, or press Enter.)** Enter an option to change parameter expression [Edit/Delete/Link to parameter/**?**] <?>: **(Press Enter to list the parameters of the selected components in the Prompt History window.)**

Command Options

Option	Description
Edit	Allows to edit the value of a parameter. Prompts you: **Enter parameter name:** Type the name of a parameter. Enter expression: Type a value in the command bar.
Delete	Allows to reset a previously edited parameter to its default value. Prompts you: **Enter parameter name:** Type the name of a parameter.
Link to parameter	Assigning parameters as an expression for a parameter of a dimensional constraint (both 2D and 3D) associated with an entity which has been put into an associative array. Prompts you: **Enter parameter name** ('*' to link all) <*>: Type the name of a parameter.
?	Lists the parameters of the selected components in the Prompt History window (6.29.13). Press the F2 function key to display the Prompt History window.

Related Commands

ArrayPath (6.4.21) - associatively distributes entity copies evenly along a path into multiple rows, and levels. ArrayPolar (6.4.23) - associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. ArrayRect (6.4.25) - associatively distributes entity copies into any number of rows, columns, and levels. -DesignTableEdit (6.10.27) - allows to edit design tables. DesignTable (6.10.26) - creates or imports .csv files containing sets of parameter values. Drop-down lists with the parameters values from the tables are added to the Mechanical Browser (3.2.2) and parametric component properties. -Parameters (6.24.5) - manages dimensional constraint parameters in the current drawing.

Figure 6.451



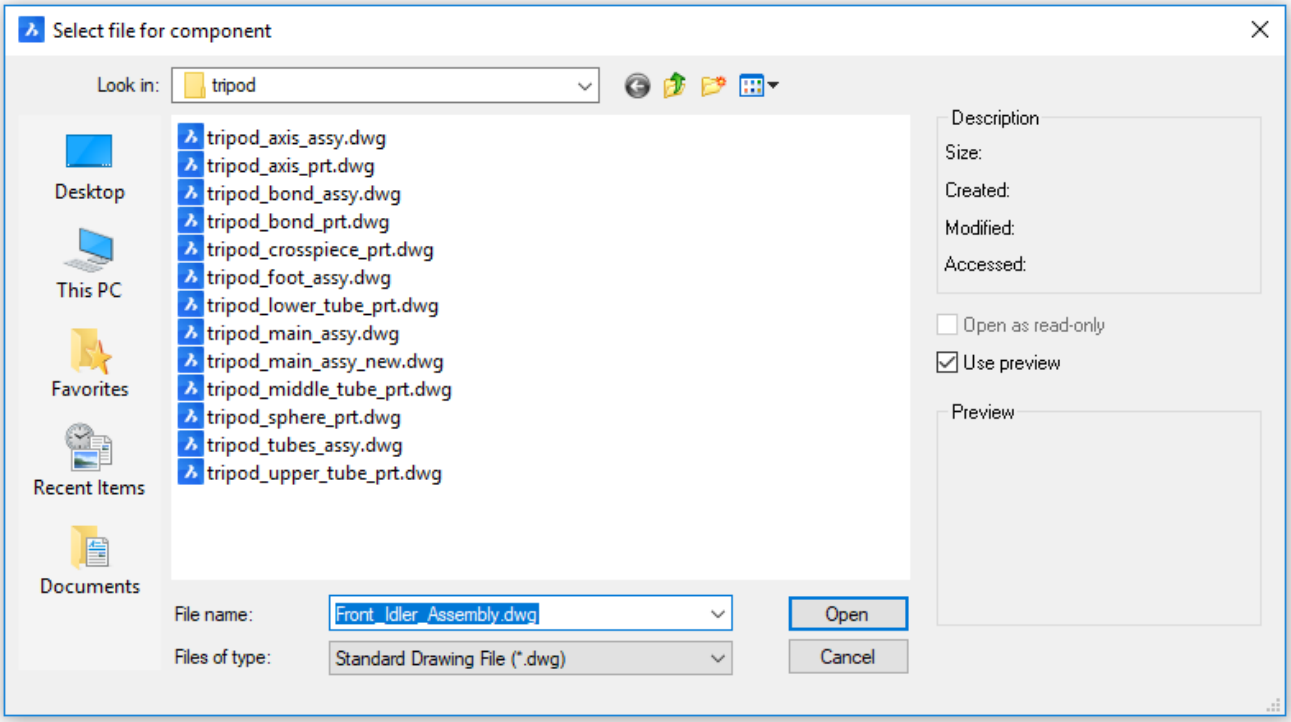
6.8.20 BmRecover

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Recovers a broken mechanical structure. The command is different from the basic Audit (6.4.32) and Recover (6.26.3) commands. It is recommended to use BmRecover to mend issues with a MCAD drawing. The command overwrites the original drawing.

Accessing the Command


command bar: **bmrecover** menu bar: **Assembly | Recover** toolbar:Assembly | Recover ribbon: **Assembly | Tools | Recover** : **bmrecover** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cm (Select a file, then double click or click the Open button.) The command reports in the command window: Fixing drawing. ... Saving... Done.

Figure 6.452



Command Options

No options

Related Commands

BmDependencies (6.8.4) - lists all files, containing component definitions inserted in the assembly, in the command window. BmUpdate (6.8.25) - reloads all referenced components from external files and updates BOM tables.

Figure 6.453



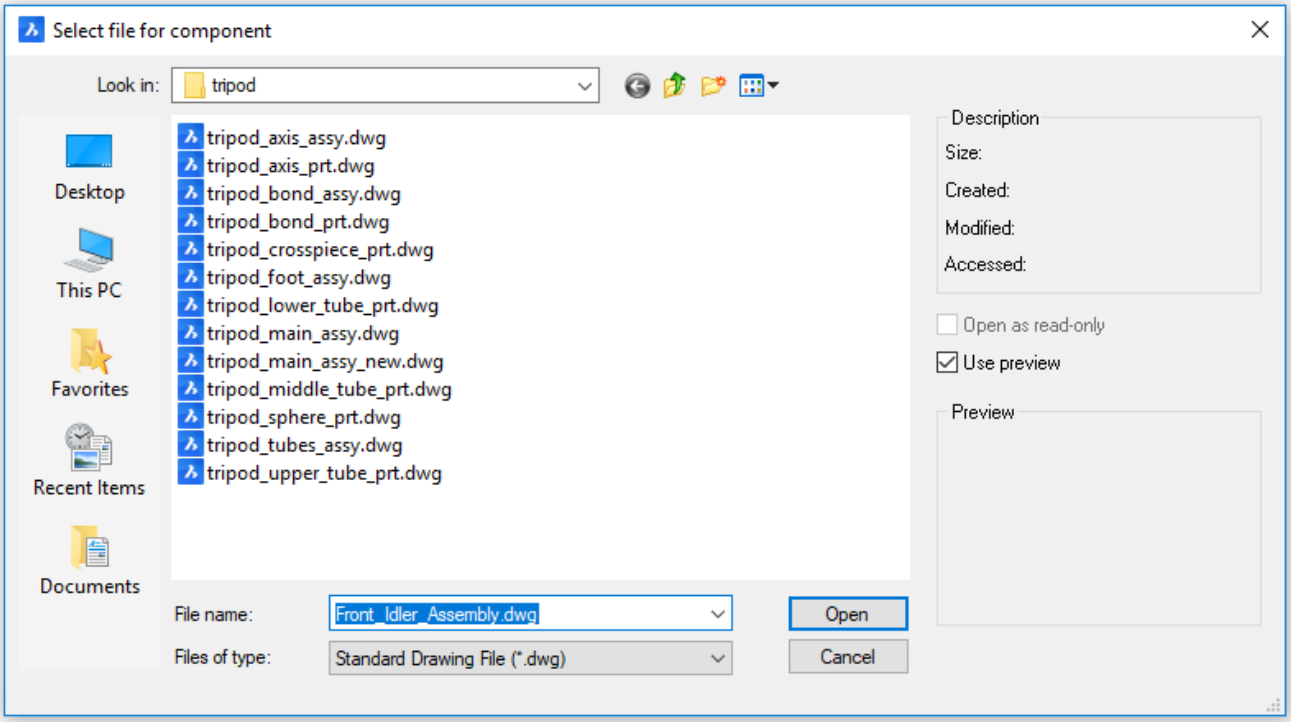
6.8.21 BmReplace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Replaces a component insert.

Accessing the Command


command bar: **bmreplace** menu bar: **Assembly | Replace Component** toolbar:Assembly | Replace Component ribbon: **Assembly | Modify | Replace Component** Quad cursor menu:**Model | Replace Component** : **bmreplace Prompts you in the command bar:** Select component inserts to be replaced: **(Select an insert in the drawing.)** Entities in set: 1 Select component inserts [SIMilar inserts/component Type]: **(Select an insert in the drawing or press Enter.)** Select component insert to use as a replacement [SIMilar inserts/component Type/Parameter changes/from FIle] <from FIle>: **(Selectcomponent insert, an option, or press Enter.)** A File dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Mechanical/ART/Replacedialog.png}) Select a file, then double click or click the Open button.

Figure 6.454



Command Options

Option	Description
SIMilar inserts	Allows to replace similar inserts automatically. Prompts you: **Replace file for all similar inserts** [Yes/No] <Yes>: • Yes: Replaces all similar inserts of the selected mechanical component(s) in assembly. • No: Replaces the selected component(s) only.
component Type	Allows to choose the insertion type for a replacement component. See External and local components (1.22.22) in the User Guide. Prompts you: **Component type** [Local/External/Keep as is/component Default] <component Default>:- Choose an optionor press Enter. • Local: the new component will be local. • External: the new component will be external. • Keep as is: the new component inherits the insertion type from the replaced component. • Component Default: the new component will be linked as specified in its source file.

Parameter changes	Allows to control which parameter values will be used after the replacement is complete. Prompts you: **Reapply changes of parameters for the component being replaced?** [Yes-reapply/No-use replacement as is] <Yes-reapply>:- Choose an optionor press Enter. • Yes-reapply: parameter values of the replaced component are applied. • No-use replacement as is:parameter values of the replacement component are applied.
from FIle	Allows to select a .dwg file for the replacement component.

Related Commands

BmExternalize (6.8.8) - converts local components to external components. BmOpen (6.8.17) - opens the source drawing of an external mechanical component.

Figure 6.455



6.8.22 BmShow

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Shows previously hidden mechanical components inserted in the current drawing.

Accessing the Command

 command bar: **bmshow** menu bar: **Assembly Visualization | Show** toolbar:**Assembly Visualization |Show** ribbon: **Assembly | Modify | Show** Quad cursor menu:**Model | Show** : **bmshow Prompts you in the command bar:** Select insert or enter an option [show ALl/COmponent name/Insert name] <show ALl>: **(Select a component of which some inserts are currently hidden or choose an option.)**

Command Options

Option	Description
show ALl	Shows all component inserts.
COmponent name	Prompts you: **Component name, all its inserts will be shown:** - **Type the component name of the component insert(s) you want to show.** This affects all inserts of the specified component.
Insert name	Prompts you: **Component insert name, use / to separate names of nested inserts:** - **Type the insert name of the component insert you want to show.** If you want to show a nested component insert (subcomponent insert) first type the insert name of the parent component insert, then the component insert name separated by a slash (/).

Related Commands

MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser. BmHide (6.8.11) - hides a mechanical component or subcomponent in the current drawing. BmVStyle (6.8.26) - applies a visual style to a mechanical component.

Figure 6.456



6.8.23 BmUnlink

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Breaks the connection between a component insert (e.g. a Window) and a 3D solid (e.g. a Wall) retaining the opening.

Accessing the Command

command bar: **bmunlink** : **bmunlink Prompts you in the command bar:** Select component instances: **(Select a component.)** Entities in set: 1 Select component instances: **(Select another component or press Enter.)**

Command Options

No command options.

Related Commands

BimInsert (6.7.14) - inserts a window or door in a solid. Inserts a copy of a window or door in a solid. BmInsert (6.8.12) - inserts an existing mechanical component or sheet metal form feature into the current drawing.

Figure 6.457



6.8.24 BmUnmech

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Converts the current mechanical component into a plain drawing. The command applies to drawings that are a mechanical component only.

Accessing the Command


command bar: **bmunmech** menu bar: **Assembly | Remove Structure** toolbar:**Assembly |Remove Structure** ribbon: **Assembly | Tools | Remove Structure** : **bmunmech Prompts you in the command bar:** Warning: this command removes all mechanical structure from the current document. Proceed? No/<Yes>: **(Press Enter or choose an option)**

Command Options

Option	Description
Yes	Removes the mechanical structure from the drawing, thus turning it into a plain drawing. Inserts of local components are converted to block references (the component name becomes the block name); inserts of external components are converted to external references.
No	Aborts the command.

Related Commands

BmDissolve (6.8.5) - dissolves a mechanical component inserted in the current drawing. BmMech (6.8.15) - converts the current drawing into a mechanical component. BmNew (6.8.16) - creates a mechanical component as a new drawing file.

Figure 6.458



6.8.25 BmUpdate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Reloads all referenced components from external files and updates BOM tables. Use this command to

- Reload referenced components from external files (BmInsert (6.8.12));
- Rebuild component based features, sheet metal form features and BIM windows/doors components (see about BC_SUBTRACT (1.22.22) layer, Form Features (3.5.22), BmInsert (6.8.12));
- Update the Bill Of Material (BmBom (6.8.2));
- Regenerate drawing views produced out of 3D model (ViewBase (6.31.3)).

NOTE	If the BMAUTOUPDATE (1.13.11) system variable is ON, external assembly components are reloaded on opening the file.

Accessing the Command

Update **Update Selected** command bar: **bmupdate** menu bar: **Assembly | Update** and **Update Selected** ribbon: **Assembly | Tools | Update**and **Update Selected** toolbar: **Assembly | Update**and **Update Selected** Quad cursor menu:**Model | Update Selected** : **bmupdate Prompts you in the command bar:** Select entities to update or [MOde/Entire model] <Entire model>: **(Select a component insert or generated drawing view; or select one of the faces of the component based or sheet metal form feature; or press Enter to select the entire model.)** Entities in set: 1 Select entities to update or [MOde/Entire model] <Entire model>: **(Select more objects.)** Updated xx definitions.

Command Options

Option	**Description**
Mode	Allows to change the way how the new placement face is defined. Each inserted component-based feature or BIM Window component has an associated placement face of the correspondent 3D solid the feature is attached to. This is the face which was used for feature placement in the BmInsert (6.8.12) command. The associated placement face of the feature can be lost in modeling or copying operations or it can be absent if the feature has been inserted in an empty area. The **BmUpdate** command allows to re-associate the feature with its placement face in one of two modes. Prompts you: Select placement face [Automatic/Manual] <Automatic>: • Automatic - the placement face for the feature is detected automatically • Manual - prompts you to select the new placement face
Entire model	Updates the entire model.

Related Commands

BmBom (6.8.2) - inserts a Bill of Materials (BOM) table in the current drawing. MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser. BmDependencies (6.8.4) - lists all files, containing component definitions inserted in the assembly, in the command window. BmInsert (6.8.12) - inserts an existing mechanical component or sheet metal form feature into the current drawing. BimInsert (6.7.14) - inserts a window or door in a solid. Inserts a copy of a window or door in a solid.

Figure 6.459



6.8.26 BmVStyle

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Applies a visual style to a mechanical component insert.

Accessing the Command

 command bar: **bmvstyle** menu bar: **Assembly Visualization / Visual Style** ribbon: **Assembly / Modify | Visual Style** toolbar: **Assembly |Visual Style** Quad cursor menu: **Model |Visual Style** : **bmvstyle Prompts you in the command bar:** Select visual style [by Viewport/Wireframe/Hidden/Realistic/Conceptual/Modeling/by Name] <Wireframe>: **(Choose an option.)** Select insert or enter an option [All/Insert name] <All>: **(Select an option or select a component insert - press Enter to select all components.)** Component insert name, use / to separate names of nested inserts: **(Type the names of one or more component inserts separated by slashes (/). Component insert names are case sensitive.)**

Command Options

Option	Description
by Viewport	Applies the current visual style of the viewport.
Wireframe	Applies the **3D Wireframe** visual style. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N
Hidden	Applies the **Hidden** visual style. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N
Realistic	Applies the **Realistic** visual style. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N
Conceptual	Applies the **Conceptual** visual style. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Mechanical	Applies the **Mechanical** visual style. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N
Modeling	Applies the **Modeling** visual style. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N
High Quality	Applies the **High Quality** visual style. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
By Name	Applies a named visual style.

Related Commands

Hide (6.16.10) - removes hidden lines from 3D entities. Render (6.26.22) - generates a photorealistic rendering using materials and lights through a dialog box. -Render (6.26.23) - generates a photorealistic rendering using materials and lights through the command bar. Shade (6.27.27) - generates shaded images of 3D drawings. Shademode (6.27.29) - Sets the shading style for 3D drawings. VisualStyles (6.31.13) - creates and controls visual style definitions in the Drawing Explorer. MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser. BmHide (6.8.11) - hides a mechanical component or subcomponent in the current drawing. BmShow (6.8.22) - shows a previously hidden mechanical component or subcomponent in the current drawing.

Figure 6.460

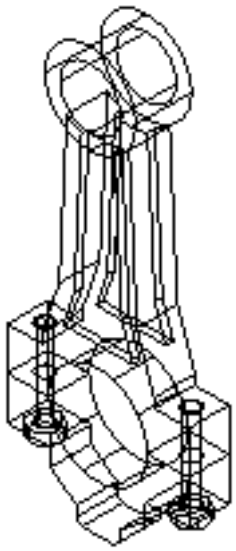


Figure 6.461

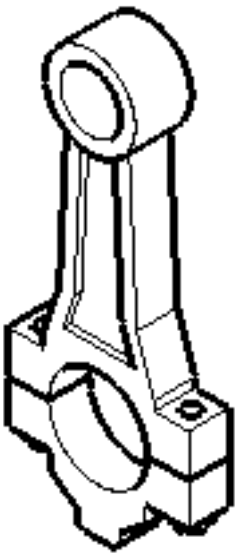


Figure 6.462

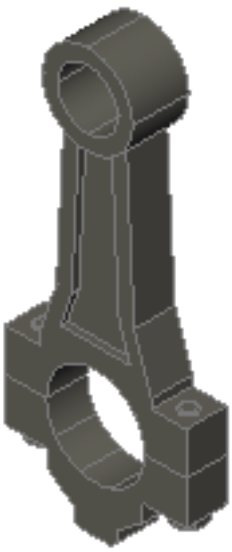


Figure 6.463

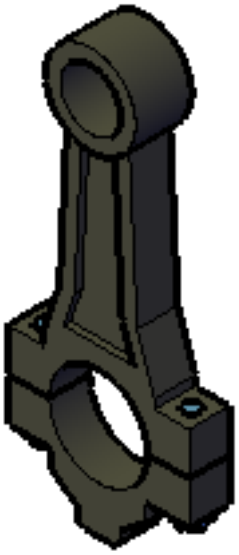


Figure 6.464

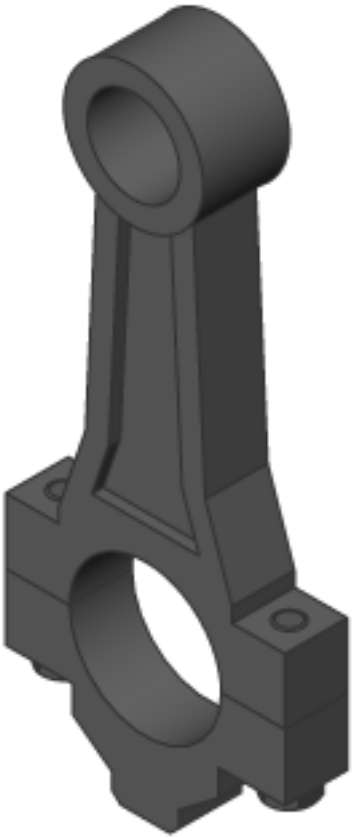


Figure 6.465

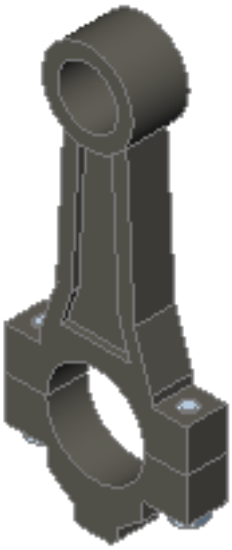


Figure 6.466

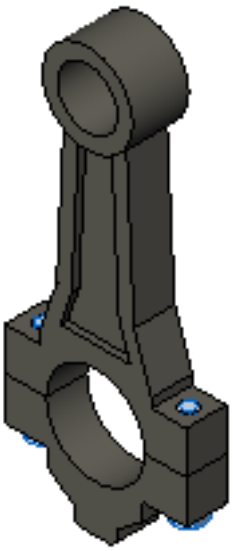


Figure 6.467



6.8.27 BmXConvert

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Converts X-Hardware solids in the current drawing to mechanical components.

Accessing the Command

command bar: **bmxconvert** : **bmxconvert** Reports you in the command bar: Number of X-Hardware solids converted to mechanical components: #

Command Options

No options.

Related Commands

BmForm (6.8.9) - creates a new mechanical component and inserts it into the current drawing. StandardPartsPanelOpen (3.3.2) - inserts a standard hardware part as a mechanical component in the current drawing.

Figure 6.468



6.8.28 MechanicalBrowserClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Closes the Mechanical Browser panel.

Accessing the Command

command bar: **mechanicalbrowserclose** :**mechanicalbrowserclose** Displays no prompts you in the command bar:

Command Options

No command options.

Related Commands

MechanicalBrowserOpen (6.8.29) - opens the Mechanical Browser panel. -ToolPanel (6.29.21) - controls the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Figure 6.469



6.8.29 MechanicalBrowserOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Opens the Mechanical Browser panel.

Accessing the Command

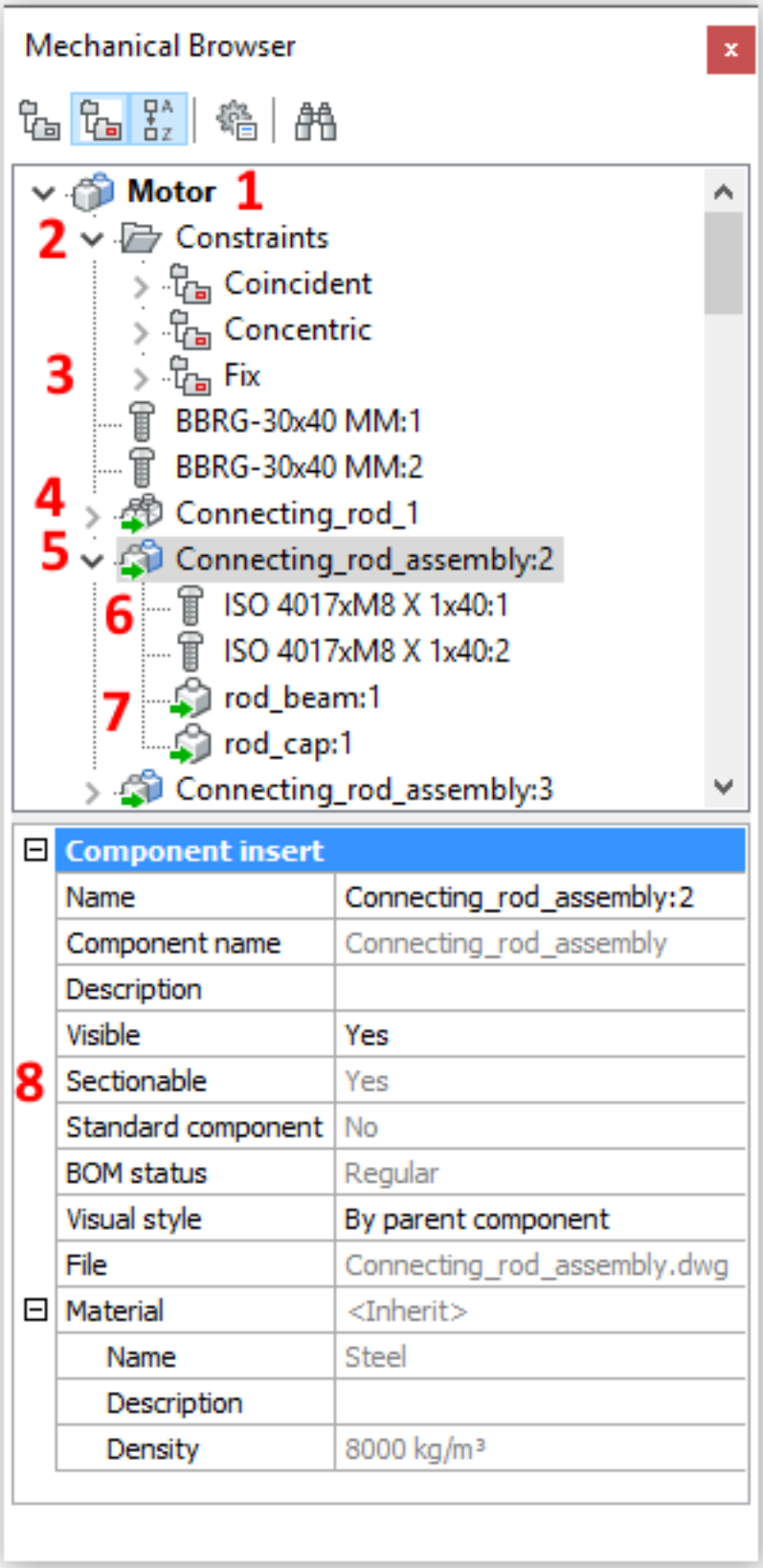

command bar: **mechanicalbrowseropen** menu bar: Assembly |**Mechanical Browser**andParametric|Mechanical Browser toolbar:Assembly |**Mechanical Browser**andParameters|Mechanical Browser ribbon: **Assembly | Tools | Mechanical Browser** context menu*: **Mechanical Browser** * Place the cursor over a toolbar or a ribbon panel, then right click. See also: Dockable Panels (1.12.7). : **mechanicalbrowseropen** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Mechanical/ART/MechanicalBrowserOpe The Mechanical Browser allows navigation on the hierarchy of mechanical components for the current drawing and calling different commands for the inserts.

1. Component Name
2. 3D Constraints
3. Standard Part Component Insert (= block reference)
4. Collapsed Component Insert (= external reference)
5. Expanded Component Insert (= external reference)
6. Standard Part Subcomponent Insert (= nested block reference)
7. Subcomponent Insert (= nested external reference)
8. Properties of the selected item

Properties set depends on the selected item: Component (3.6.3),Parameter (1.19.3),Array (3.6.5),Feature. (3.8.6)

Keyboard accelerators	• Use **Enter** to accept a new value • Use **F2** to rename a node • Use **Space** to select an entity/component • Use **Alt+Up**, **Alt+Down** keys to move separator of Mechanical Browser

Figure 6.470


Mechanical Browser
Motor
1
2
Constraints
Coincident
Concentric
3
Fix
BBRG-30x40 MM:1
BBRG-30x40 MM:2
4
Connecting_rod_1
5
Connecting_rod_assembly:2
6
ISO 4017xM8 X 1x40:1
ISO 4017xM8 X 1x40:2
7
rod_beam:1
rod_cap:1
Connecting_rod_assembly:3
Component insert
Name
Connecting_rod_assembly:2
Component name
Connecting_rod_assembly
Description
Visible
Yes
8
Sectionable
Yes
Standard component
No
BOM status
Regular
Visual style
By parent component
File
Connecting_rod_assembly.dwg
Material
<Inherit>
Name
Steel
Description
Density
8000 kg/m³

Command Options

Option Description

Model Group by Entity Groups 3d constraints by entity or mechanical component they are applied to. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Mechanical/ART/MechanicalBrowserGrou

Group by Type Groups mechanical components and 3d constraints in separate trees. Groups 3d constraints by type. Expand a constraint to see the 3D features affected by the constraint. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc

Sort Alphabetically When pressed ( ) the mechanical components and 3d constraints are sorted alphabetically. When depressed ( ) the mechanical components and 3d constraints are listed in the order they were added to the assembly. Settings for visual representation of parameters Constraints expressions control whether the numeric value or assigned parameter name displayed. Expressions at sub-components control visual representation of sub-component parameter: numeric value or assigned parameter name. Show sub-component parameters control visibility ofsub-component parameters. Parameters at properties add parameters section for selected instance to the mechanical browser properties. Search field When pressed demonstrates the search filed which allows to type a name of an object to search in the mechanical browser.

Procedures

- To open or close a panel (1.12.7)
- To dock a panel (1.12.7)
- To dock a panel with respect to another docked panel (1.12.7)
- To stack panels in a tabbed panel (1.12.7)

Related Commands

3dCompare (6.3.3) - executes a geometrical comparison between solids and surfaces in 2 drawing files. The result is shown in a new drawing and reported in the Mechanical Browser panel. BmDependencies (6.8.4) - lists all files, containing component definitions inserted in the assembly, in the command window. BmExternalize (6.8.8) - switches internal components to external components. StandardPartsPanelOpen (3.3.2) - inserts a standard hardware part as a mechanical component in the current drawing. BmLocalize (6.8.13) - switches external components to internal components. BmOpen (6.8.17) - opens the component insert drawing. BmOpenCopy (6.8.18) - opens a copy of a component insert as a new drawing. BmReplace (6.8.21) - replaces a component insert. BmUpdate (6.8.25) - reloads all referenced components from external files and updates BOM tables. DesignTable (6.10.26) - creates or imports .csv files containing sets of parameter values. Drop-down lists with the parameters values from the tables are added to the Mechanical Browser and parametric component properties. -DesignTableEdit (6.10.27) - allows to edit design tables. MechanicalBrowserClose (6.8.28) - closes the Mechanical Browser panel. -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line. SmParametrize (6.28.21) - creates a consistent set of 3D constraints for a selected sheet metal part.

Figure 6.471

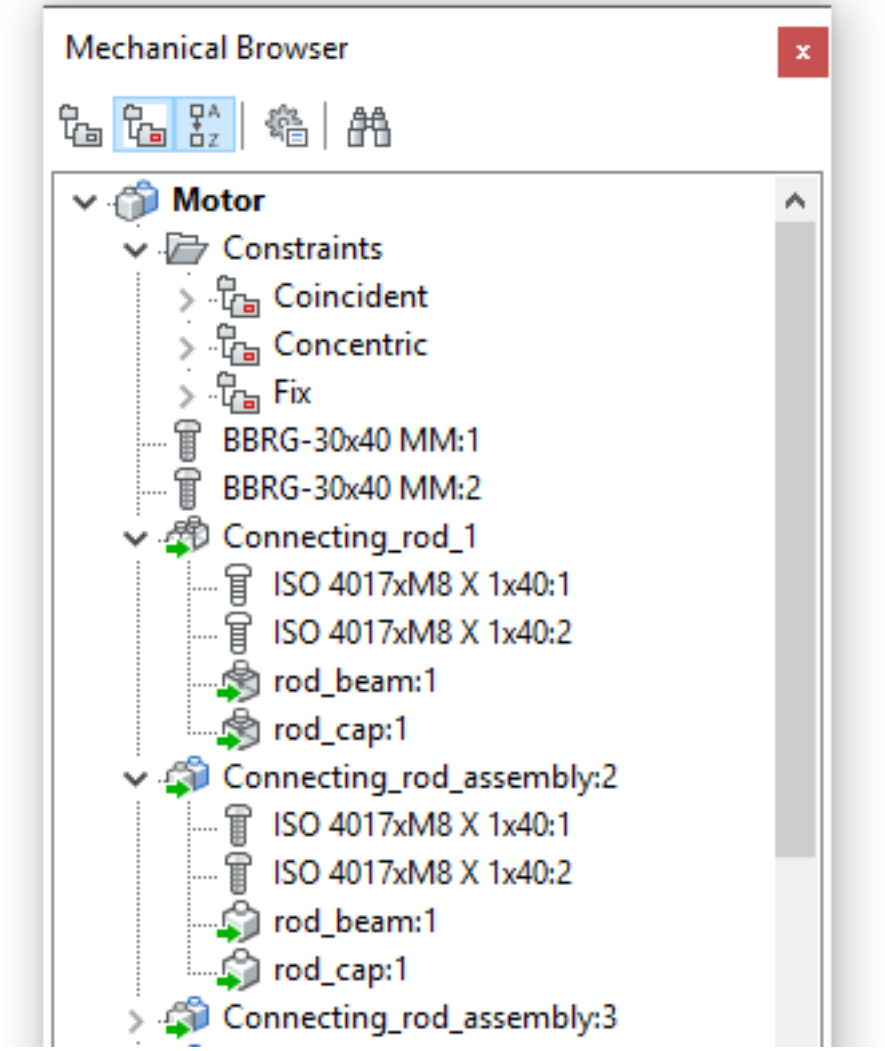

Mechanical Browser
Motor
Constraints
Coincident
Concentric
Fix
BBRG-30x40 MM:1
BBRG-30x40 MM:2
Connecting_rod_1
ISO 4017xM8 X 1x40:1
ISO 4017xM8 X 1x40:2
rod_beam:1
rod_cap:1
Connecting_rod_assembly:2
ISO 4017xM8 X 1x40:1
ISO 4017xM8 X 1x40:2
rod_beam:1
rod_cap:1
Connecting_rod_assembly:3

Figure 6.472

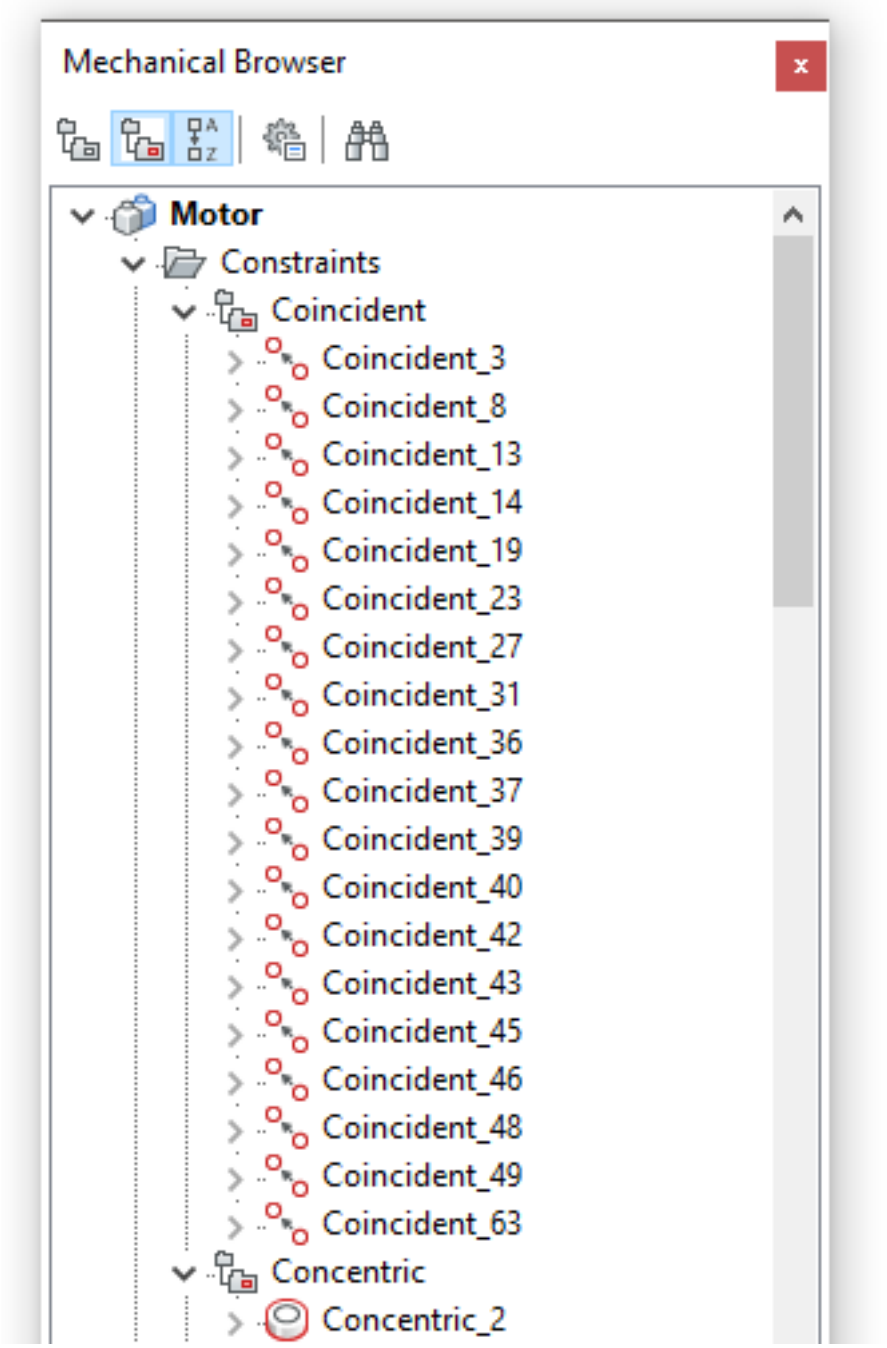

Mechanical Browser
Motor
Constraints
Coincident
Coincident_3
Coincident_8
Coincident_13
Coincident_14
Coincident_19
Coincident_23
Coincident_27
Coincident_31
Coincident_36
Coincident_37
Coincident_39
Coincident_40
Coincident_42
Coincident_43
Coincident_45
Coincident_46
Coincident_48
Coincident_49
Coincident_63
Concentric
Concentric_2

Figure 6.473



6.8.30 ParametersPanelClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Closes the parameters Panel.

Accessing the command

Command bar: parameterspanelclose **:parameterspanelclose** Displays no prompt.

Command Options

No command options

Related Commands

ParametersPanelOpen (6.24.8)- opens the Parameters Panel.

Figure 6.474



6.8.31 ReportPanelClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Closes the Report panel.

Accessing the Command

command bar: **reportpanelclose** context menu*: **Report** * Place the cursor over a toolbar or a ribbon panel, then right click. : **reportpanelclose** Displays no prompts in the command bar:

Command Options

No command options.

Related Commands

ReportPanelOpen (6.8.32) - opens the Report panel. -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Figure 6.475



6.8.32 ReportPanelOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Opens the Report panel.

Accessing the Command

command bar: **reportpanelopen** context menu*: **Report** * Place the cursor over a toolbar or a ribbon panel, then right click. : **reportpanelopen** Displays a dialog box. Some commands use the **Report** panel to report the command output; e.g. all SheetMetal (Sm*) commands, DmAudit (6.12.2) and DmStitch (6.12.28). The **Report** panel is used instead of the command bar only if the REPORTPANEL (1.13.11) system variable is ON.

Command Options

No command options.

Procedures

To open or close a panel (1.12.7) To dock a panel (1.12.7) To dock a panel with respect to another docked panel (1.12.7) To stack panels in a tabbed panel (1.12.7)

Related Commands

ReportPanelClose (6.8.31) - closes the Report panel. -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

6.9 C

6.9.1 Cal

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the operating system's Calculator prgram.

Figure 6.476



Accessing the Command

command bar: **cal** transparent: **'cal : Cal** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en

1. Enter numbers and operations.
2. Press =.
3. For perform scientific calculations, choose **View | Scientific**.
4. To close the calculator, click **X**.

Figure 6.477



Command Options

Option	Description
View	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C Click the **Mode** button (☰) and choose an option.

Related Commands

Shell (6.27.32) - access the command prompt of the operating system; runs Windows programs. Files (6.14.5) - displays the operating system's File Manager.

Figure 6.478

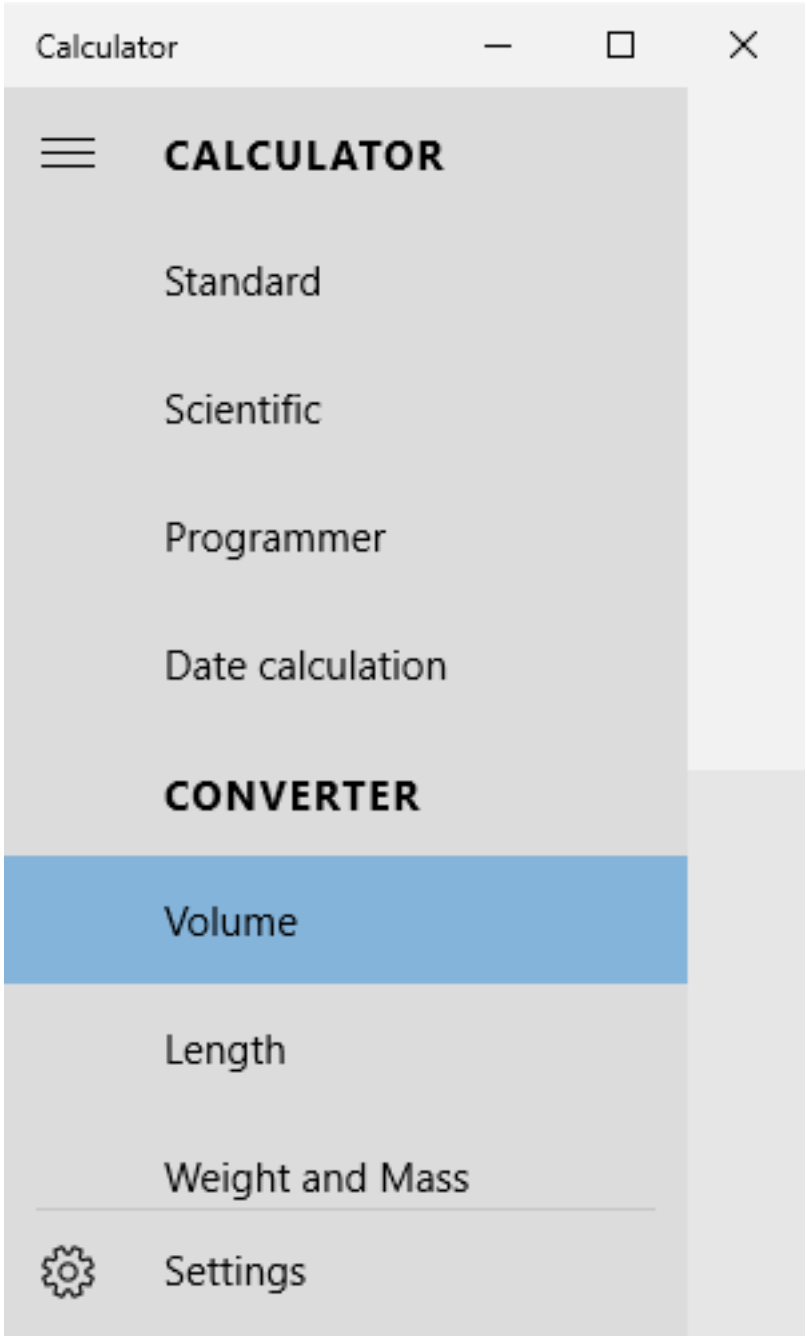

Calculator
CALCULATOR
Standard
Scientific
Programmer
Date calculation
CONVERTER
Volume
Length
Weight and Mass
Settings

Figure 6.479



6.9.2 Camera

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Places camera glyphs in drawings that point at target points, and creates named views.

Accessing the Command


command bar: **camera** menu bar: **View | Camera** toolbar: **View |** ribbon:**View | View | Viewpoint | Camera :camera You are prompted:** Specify camera location: **(Enter a point to specify the location of the camera)** Specify target location: **(Enter a point to specify the location of the target, the point at which the camera looks)** Enter an option [?/Name/LOcation/Height/Target/LEns/CLipping/View/eXit] <eXit>: **(Enter an option, or else press Enter to conclude the command)** A camera glyph displays in the drawing to indicate the position of the camera.


Named views are saved with the drawing, named after the cameras, which are accessed by the **View (6.31.1)** command.

Command Options

Option	Description
?	Displays a list of existing cameras.Prompts you: **Enter camera name(s) to list <*>** - **(do one of the following)**: • Press Enter to list all existing cameras. • Use wildcards to (* or ?) list a selection of cameras. For example,Cam* lists all cameras names that start with 'cam' and?a* lists all camera names of which the second letter is 'a'
Name	Names the new camera; prompts you: **Enter name for new camera <Camera1>** - **(type a name for the saved view, or else press Enter to accept the default name).**
Location	Positions the camera. Prompts you: **Specify camera location <x,y,z>** - **(do one of the following):** • Pick a point in the drawing • Type the x, y, z coordinates in the command bar, then press Enter.
Height	Sets the height (z coordinate) of the camera; prompts you: **Specify camera height <current height>** - **(enter a value).**

Target	Positions the target, the point at which the camera looks. Prompts you: **Specify target location <x,y,z>** - **(one of the following):** • Pick a point in the drawing. • Type the x-, y- and z-coordinates in the command bar, then press Enter.
Lens	Defines the lens length. Prompts you: **Specify lens length in mm <50.00>** - **(Enter a value).** A smaller number, such as 20, provides a wider field of view, while a higher number, such as 200, provides a closer look, like a zoom lens on a camera.
Clipping	Defines the front and back clipping planes, which cut off the view. Prompts you: 1. **Enable front clipping plane? Yes/<No>** - **(type Y to define the front clipping plane)**. Prompts you: 2. **Specify front clipping plane offset from target plane <0.00>-(do one of the following):** • Pick a point in the drawing. • Type a value in the command bar, then press Enter 3. **Enable back clipping plane? Yes/<No>**- **(type Y to define the back clipping plane).** Prompts you: 4. **Specify back clipping plane offset from target plane <0.00>** - **(do one of the following)**: • Pick to point in the drawing. • Type a value in the command bar, then press Enter
View	Sets the camera view current. Prompts you: **Switch to camera view? Yes/<No>** - **(type Y , then press Enter).**
Exit	Exits the command, and then creates the camera.

Editing a Camera

Properties of a camera view can be edited:

- In the dialog box displayed by the View (6.31.1) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
- In the Properties (6.24.49) bar: Select the camera glyph in the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys

Figure 6.480

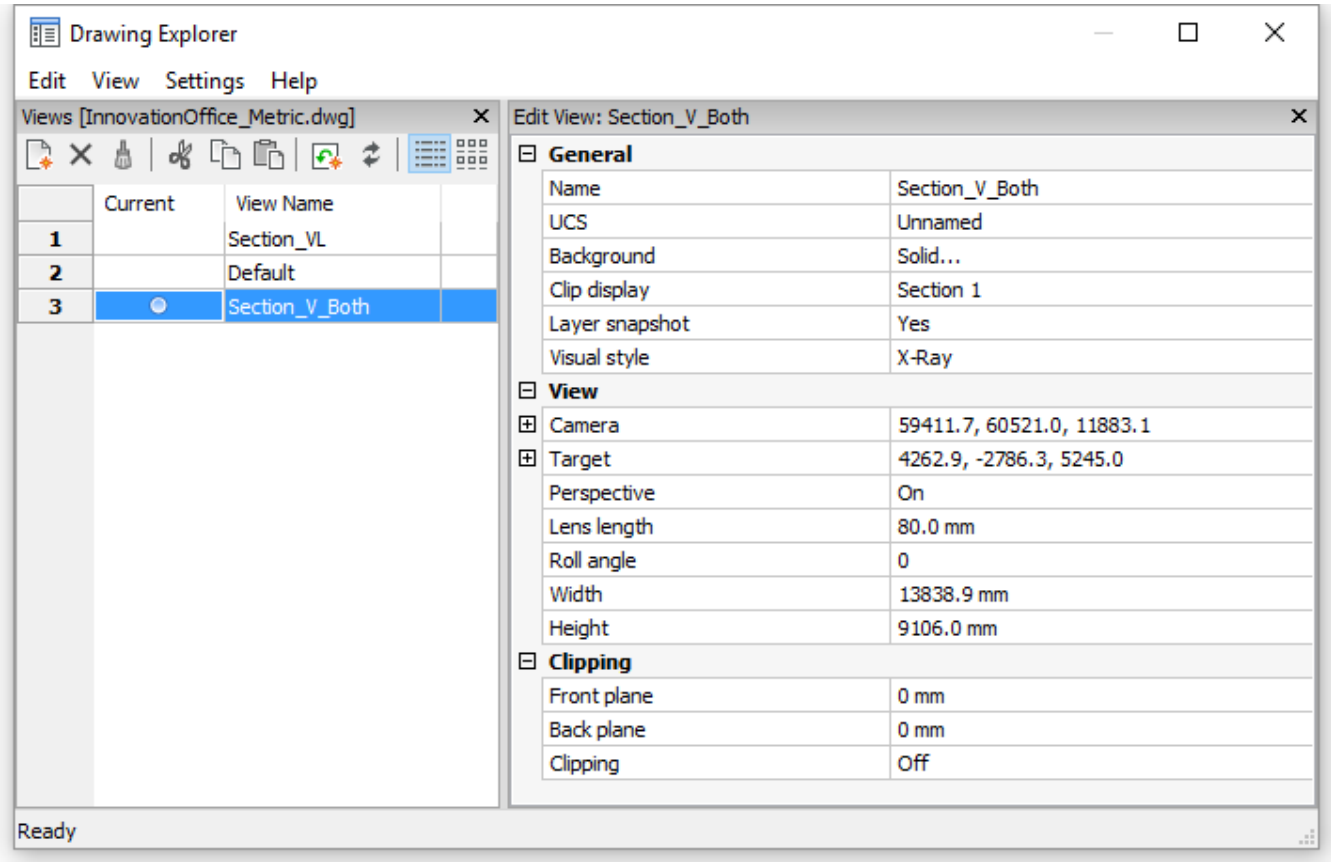


Figure 6.481

Camera	
Name	Camera 1
Camera	-422.40, -2318.27, 0.00
X	-422.40
Y	-2318.27
Z	0.00
Target	5551.65, -2396.75, 0.00
X	5551.65
Y	-2396.75
Z	0.00
Lens Length	50.00
Field of view	38°34'48"
Roll angle	0°0'0"
Plot glyph	No
Clipping	
Front plane	4500.00
Back plane	1000.00
Clipping	Front and back on

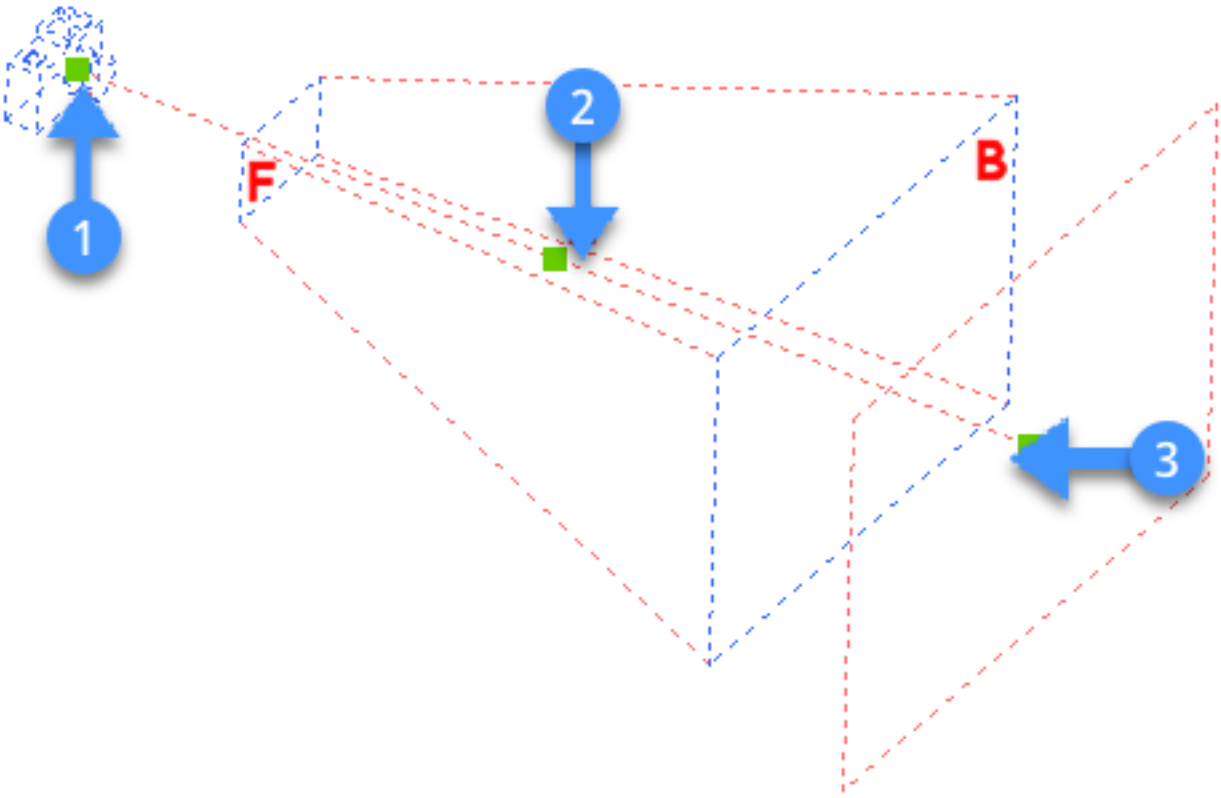
Grips Editing

Cameras can be edited directly through grips:

1. Select the camera glyph in the drawing. Notice that it has three grips.Front (F) and back (B) clipping planes display in a blue dotted line.
2. Drag the grips to perform the following actions:
   - Position the camera (1).
   - Move the camera definition as a whole (2).
   - Position the target (3).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/CameraGripEditinga.png})

Figure 6.482



Related Commands

AniPath (6.4.8) - records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file. DView (6.10.45) - changes the 3D viewpoint interactively, and turns on perspective mode. MatchPerspective (6.21.7) - matches the current model space perspective view with a background image by selecting 3 or more matching point pairs. View (6.31.1) - creates and sets named views through the Drawing Explorer.

Figure 6.483



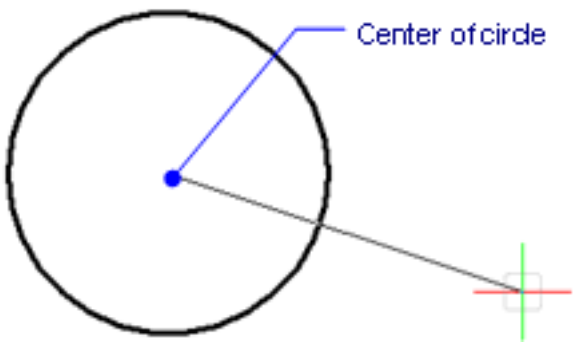
6.9.3 Center

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles Center entity snap; snaps to the center of circles, arcs, and other entities with a circular curve.

Accessing the Command

 command bar: **center** transparent: **cen** toolbar: **Entity Snaps** | ribbon:**Home | Entity Snaps | Snap to Center**
: center Displays no prompts in the command bar. Each time you enter the command, the program toggles the Center entity snap mode between on and off. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou
When center entity snap is on, the cursor snaps to the center points of curved entities, such as circles, arcs, and polyarcs.

Figure 6.484



Command Options

Option	Description
Center	Entering the command toggles Center entity snap.This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.485



6.9.4 CenterDisassociate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Breaks the association of centerline and center mark entities with the associated entities, such as lines, arcs, and circles.

Accessing the Command


command bar: **centerdisassociate** menu bar: **Dimension | Center Marks and Lines | Center Disassociate** toolbar:**Dimensions | Center Marks and Lines |** **: centerdisassociate You are prompted:** Select a center mark or centerline to disassociate: **(Select a center mark or centerline entity)** Entities in set: 1 Select a center mark or centerline to disassociate: **(Select a center mark or centerline entity, or else press Enter)**

Command Options

Option	Description
Select a center mark or centerline to disassociate	Selects a center mark or centerline to disassociate from the associated entity(ies), such as circles, arcs, or lines The command repeats automatically, so press Enter to end it.

Related Commands

CenterLine (6.9.5) - creates a centerline entity associated with two selected lines. CenterMark (6.9.6) - creates a center mark entity associated with a selected circle or arc. CenterReassociate (6.9.7) - recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc. CenterReset (6.9.8) - resets centerline and center mark entities.

Figure 6.486



6.9.5 CenterLine

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a centerline associated with two selected lines or polyline segments.

Accessing the Command


command bar: **centerline** alias: **cl** menu bar:**Dimension | Center Marks and Lines |Centerline** toolbar:**Dimensions | Center Marks and Lines |** **: centerline You are prompted:** Select first line: **(Select a line entity)** Select second line: **(Select another line entity)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Cen
When applying a centerline to the intersection of two lines, the location of the picked points determine the direction of the centerline. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Centerlines2.png})

Figure 6.487

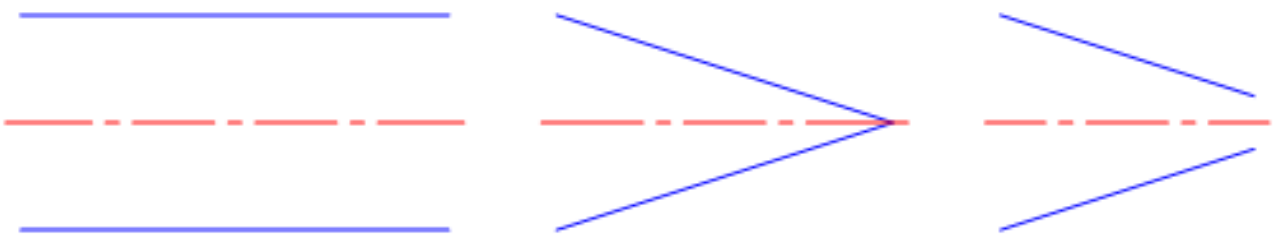
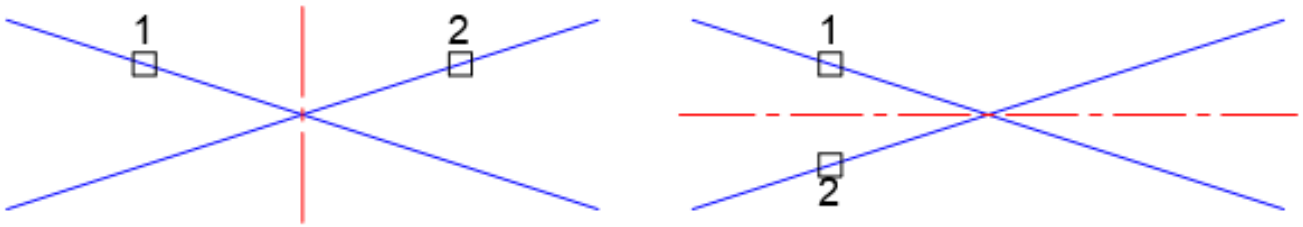


Figure 6.488



Command Options

Option	Description
Select first line	Select a line or polyline segment
Select second line	Select another line or polyline segment

The lines and polyline segments can be selected inside blocks and in drawing view viewports. You can pick two segments of the same polyline. The centerline is associative, so when one or both lines are moved, the centerline repositions itself.

Procedures

Center lines and center marks (1.21.11)

Grips Editing

Centerlines can be edited through grips. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I

- 1: Overshoot grips: drag the grip to lengthen the centerline
- 2: Midpoint grip: drag the grip to move the centerline*.

When an extension is applied to one or both ends of the centerline by dragging an overshoot grip, four grips display. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/CenterlinesOvershootGripsEditinga

- 3: Overshoot grips: drag the grip to lengthen the centerline
- 4: Midpoint of the original centerline; drag the grip to move the centerline*.
- 5: Midpoint of the extended centerline; drag the grip to move the centerline*.

*Use the CenterReset (6.9.8) command to reset the centerline.

Figure 6.489

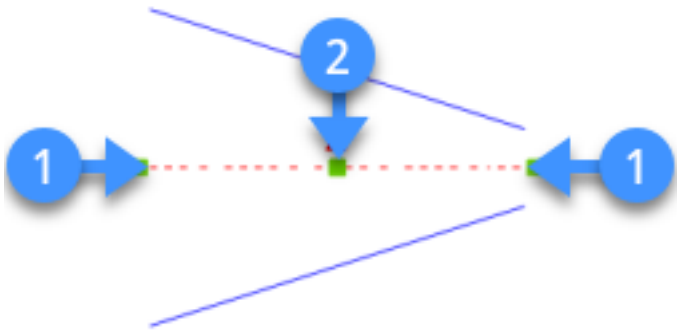
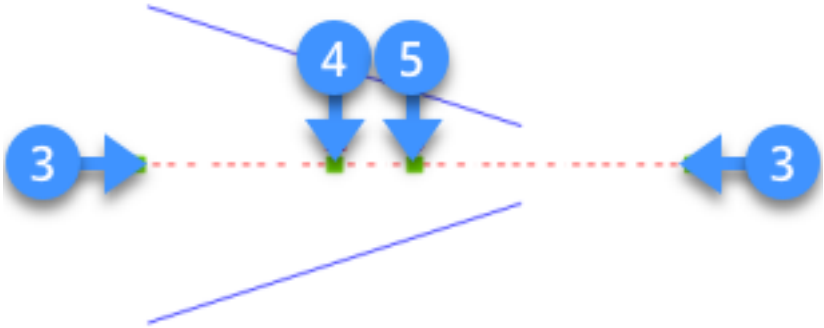


Figure 6.490



Related Commands

CenterDisassociate (6.9.4) - breaks the association of centerline and a center mark entities. CenterMark (6.9.6) - creates a center mark entity associated with a selected circle or arc. CenterReassociate (6.9.7) - recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc. CenterReset (6.9.8) - resets centerline and center mark entities.

Figure 6.491



6.9.6 CenterMark

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a center mark entity associated with a selected circle, arc, or polyline arc.

Accessing the Command

 command bar: **centermark** alias: **cm** menu bar:**Dimension | Center Marks and Lines |Center Mark** toolbar:**Dimensions | Center Marks and Lines|** **: centermark You are prompted:** Select circle or arc: **(Select a circle or a circular arc.)** Depending on the current value of the CENTERMARKEXE system variable, the center mark is drawn with or without extension lines.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/CenterMark.png})
Center mark only (CENTERMARKEXE = OFF)
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/CenterMark+Extensions.png})
Center mark with extension lines (CENTERMARKEXE = ON)

Command Options

Option	Description
Select circle or arc	Select a circle, an arc, or a polyarc segment, which can be selected inside blocks and in drawing view viewports.

The centermark is associative, so when the arc or circle is moved, the centermark repositions itself.

Procedures

- Center lines and center marks (1.21.11)

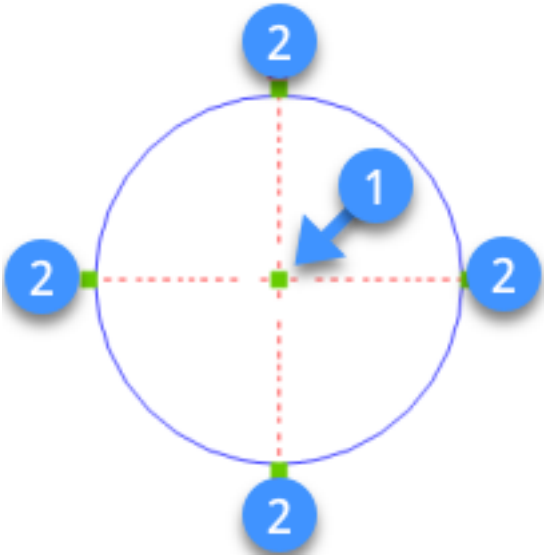
Grips Editing

Center marks can be edited through grips. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands

1. Center grip: drag the grip to move the center mark*.
2. Overshoot grips: drag the grip to lengthen the extension line**.

*Use the CenterReset (6.9.8)command to reset the centerline. **If you lengthen an extension line, it loses its associativity, and so changing the diameter does not adjust the changed extension line. It will, however, still moved along with a circle.

Figure 6.492



Related Commands

DimCenter (6.11.8) - places non-associative center marks at the center points of circles and arcs. CenterDisassociate (6.9.4) - breaks the association of centerline and a center mark entities. CenterLine (6.9.5) - creates a centerline entity associated with two selected lines. CenterReassociate (6.9.7) - recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc. CenterReset (6.9.8) - resets centerline and center mark entities.

Figure 6.493

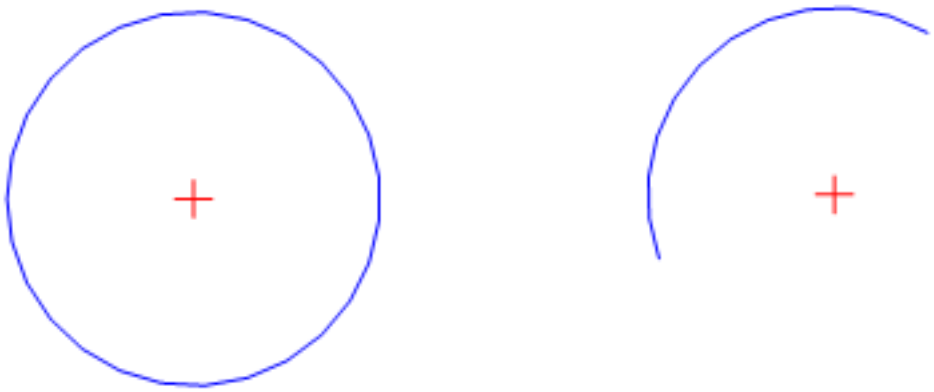


Figure 6.494

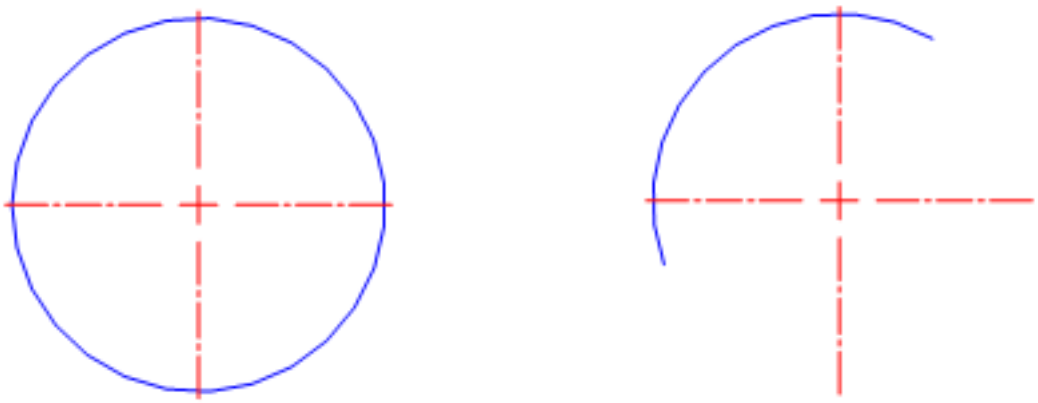


Figure 6.495



6.9.7 CenterReassociate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Associates a centerline entity with two selected lines, or else a center mark with a circle, arc, or polyarc.

Accessing the Command


command bar: **centerreassociate** menu bar:**Dimension | Center Marks and Lines | Center Reassociate** toolbar:**Dimensions | Center Marks and Lines |** **: centerreassociate You are prompted:** Select a center mark or centerline to reassociate: **(Select a center mark or centerline entity)** When you select a center mark, **prompts you:** Select circle or arc: **(Select a circle, circular arc, or polyarc)** When you select acenterline, **prompts you instead:** Select first line: **(Select a line entity)** Select second line: **(Select another line entity)**

Command Options

Option	Description
Select a center mark or centerline to reassociate	Selects the center mark or a centerline to associate with entities; prompts you: • When you select a center mark, prompts you,**Select circle or arc**- **(select a circle, circular arc, or polyarc)** • When you select acenterline, prompts you instead**Select first line**- **(select a line entity**), and then prompts you**Select second line:**- **(select another line entity)**

Related Commands

CenterDisassociate (6.9.4) - breaks the association of centerline and a center mark entities. CenterLine (6.9.5) - creates a centerline entity associated with two selected lines. CenterMark (6.9.6) - creates a center mark entity associated with a selected circle or arc. CenterReset (6.9.8) - resets centerline and center mark entities.

Figure 6.496



6.9.8 CenterReset

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Resets centerline and center mark entities.

Accessing the Command



command bar: **centerreset** menu bar:**Dimension | Center Marks and Lines | Center Reset** toolbar:**Dimensions | Center Marks and Lines |** **: centerreset You are prompted:** Select a center mark or centerline to reset: **(Select a center mark or centerline entity)** Entities in set: 1 Select a center mark or centerline to reset: **(Select a center mark or centerline entity, or else press Enter)**

Command Options

Option	Description
Select a center mark or centerline to reset	Select a center mark or centerline entity, or else press Enter to end the command

Centerlines are reset by the program when one of the associated lines is moved.Center marks are reset automatically when the associated circle or arc is moved or when the their radius or diameter is modified.

Related Commands

CenterDisassociate (6.9.4) - breaks the association of centerline and a center mark entities. CenterLine (6.9.5) - creates a centerline entity associated with two selected lines. CenterMark (6.9.6) - creates a center mark entity associated with a selected circle or arc. CenterReassociate (6.9.7) - recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc.

Figure 6.497



6.9.9 Chamfer

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates chamfers at intersections, defined by two lengths, or a length and an angle.

Accessing the Command



command bar: **chamfer** alias: **cha** menu bar: **Modify | Chamfer** toolbar: **Modify |**  ribbon:**Change | Modify | Chamfer : chamfer You are prompted:** Chamfer (dist1=0, dist2=0) Select first entity [Chamfer Settings... /Polyline/ Angle/ Distance/ mEthod/Trim/Undo/Multiple]: **(Select an entity, or else enter an option)** Select second entity (select with pressed SHIFT to make corner): **(Select another entity)** Chamfered intersection: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image88.png}) Chamfered polyline: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image64.jpg})

Figure 6.498

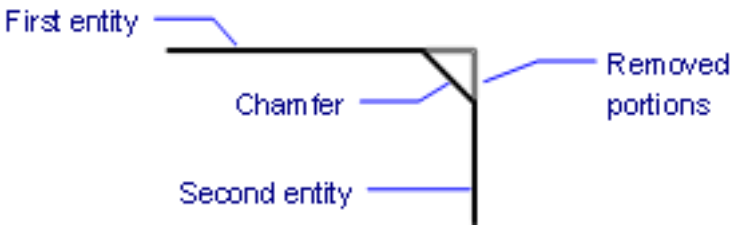
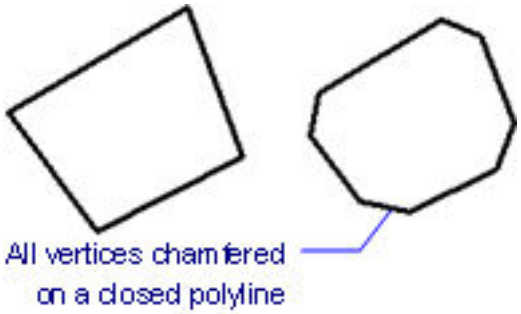


Figure 6.499



Command Options

Option	Description
Select first entity	Selects the first entity to be chamfered.Only open entities can be chamfered, with one exception: vertices of closed polylines can be chamfered with the **Polyline** option. This command cannot place a chamfer between two polylines,but it can place a chamfer between a line and a polyline, even a closed polyline.
Select second entity	Selects the second entity.The two entities need not meet, but cannot be parallel to each other. To connect connect both entities without creating a chamfer, hold the **Shift** key when selecting the second entity; this acts like a combined Trim-Extend command.
Settings	Displays the Settings (1.13.11) dialog box for setting chamfer parameters.
Polyline	Chamfers the vertices of a polyline; prompts you: **Select 2D polyline to chamfer** - **(Select a polyline with at least one vertex)**
Angle	Changes to the Length-Angle method of determining the chamfer line; prompts you: 1. **Length on first entity <0>** - **(specify a length, or pick two points, or else press Enter to accept the default)** 2. **Angle relative to first entity <0>** - **(specify an angle, or pick two points, or else press Enter to accept the default)** When you pick two points to specify the angle, the program measures the angle from 0 degrees, the x axis.
Distance	Changes to the Distance-Distance method of determining the chamfer line; prompts you: 1. **Chamfer distance on first entity <0>** - **(specify a length, or pick two points, or else press Enter to accept the default)** 2. **Chamfer distance on second entity <0>** - **(specify an length, or pick two points, or else press Enter to accept the default)**

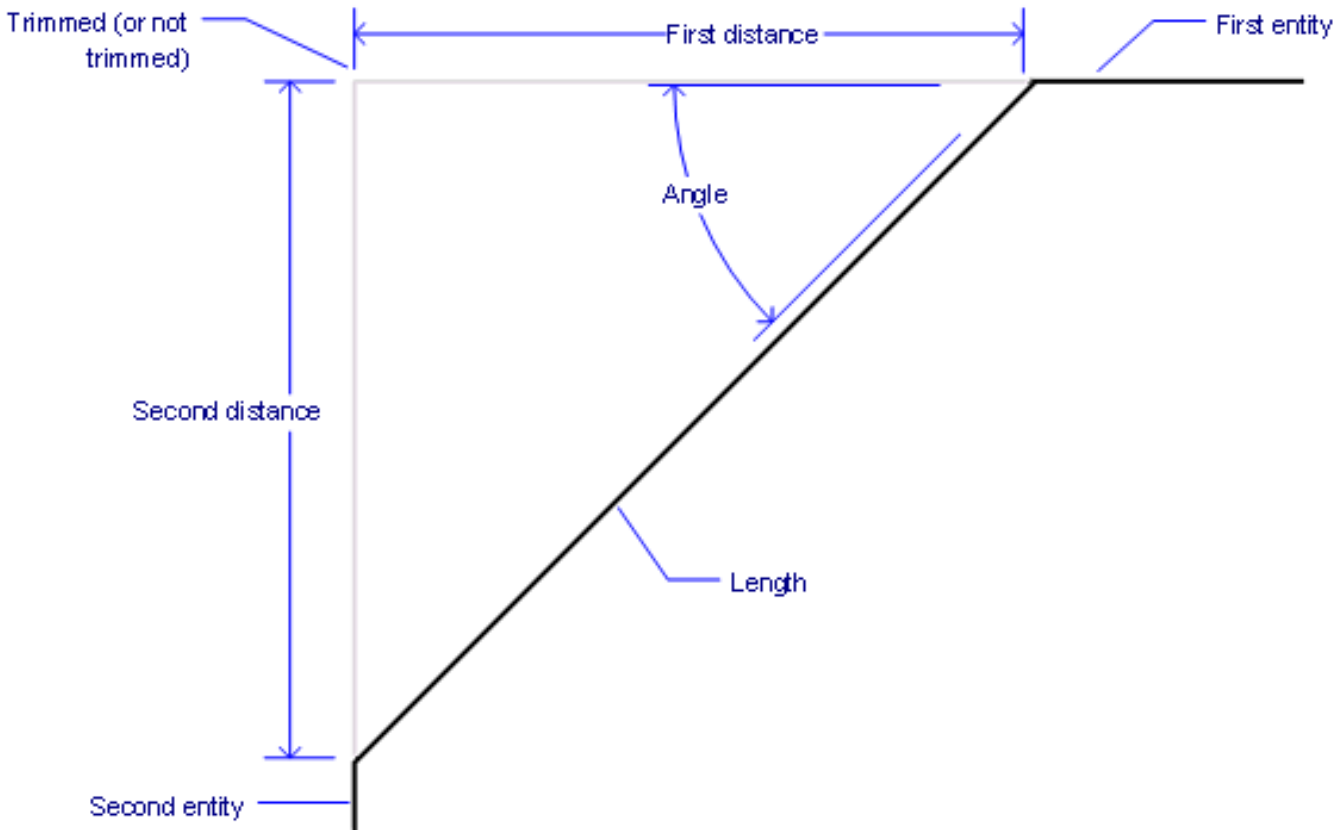
Method	Switches between Length-Angle and Distance-Distance modes; remember to enter an **e** for the this option's abbreviation, and not an **m**. Prompts you: Chamfer using [Angle/Distance] <Distance> - **(enter an option)**: • **Angle** -changes to the Length-Angle method of determining the chamfer line • **Distance** -changes to the Distance-Distance method of determining the chamfer line The program will keep using the same method for determining the chamfer until you change it again.
Trim	Determines if the program trims the entities; prompts you: **[Trim/No] <Trim>** - **(choose an option or press Enter to keep the current mode):** • **Trim** - trims or extends the selected entities to meet the chamfer line • **No** - creates the chamfer and leaves the selected entities unchanged
Undo	Undoes the last chamfer when in Multiple mode.
Multiple	Lets you create multiple chamfers using the same settings over and over again; press **Esc** to exit the command.

Settings

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image93.png})

Setting	**Description**
Chamfer Mode	Determines how the chamfer is created: • **Distance-Distance** - specify two distances. • **Length-Angle** - specify a length and an angle.
Chamfer First Distance	Specifies the default value of the first chamfer distance.
Chamfer Second Distance	Specifies the default value of the first second distance.
Chamfer Length	Specifies the default length of the chamfer.
Chamfer Angle	Specifies the default angle of the chamfer.
Trim Mode	Toggles whether intersecting entities are trimmed: • **Yes** - trims or extends entities to the chamfer line. • **No** - does not trim intersecting entities.

Figure 6.500



Procedures

- Chamfering using the distance-distance method (1.23.28)
- Chamfering using the length-angle method (1.23.28)
- Chamfering all vertices of a polyline (1.23.28)

Related Commands

Break (6.6.18) - cuts parts out of entities. dmChamfer (6.12.4) - creates an equal distance chamfer between adjacent faces sharing a sharp edge. dmFillet (6.12.14) - creates a smooth fillet between adjacent faces sharing a sharp edge. Extend (6.13.19) - extends open entities to a boundary. Fillet (6.14.7) - fillets intersections. Settings (6.27.23) - specifies parameters for chamfers. SolidEdit (6.27.39) - edits faces, edges, and bodies of solid models. Trim (6.29.26) - trims entities where they cross other entities.

Figure 6.501



6.9.10 Change

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Changes the properties of entities, through the command bar. This command has largely been superseded by the Properties panel.

Accessing the Command



command bar: **change** alias: **-ch : change You are prompted:** Select entities to change: **(Choose one or more entities)** Entities in set: xx Select entities to change: **(Press Enter to end entity selection)** Change point [special Entities/Properties...]: **(Enter an option, or specify a new endpoint)** When you enter **P**at the prompt: Property to change [Color/Elevation/LAyer/LineType/linetype Scale/Line Weight/Thickness/TRansparency/Material/Annotative]: **(Enter an option, or press Enter to exit the command)**

Command Options

Option	Description
Select entities to change	Choose one or more entities in the drawing.
Special Entities	Specifies options for changing the properties of rays and infinite lines (xlines): **Point change for Rays and Infinite Lines affects [Angles/Points/None] <No change>** - **(enter an option)**: • **Angles** - changes the angle. • **Points** - changes the location of the points defining the rays and infinite lines. • **None** - leaves entities unchanged.
Properties Options	
Color	Changes the color of the entity; prompts you: **New color** - **(enter the name or number of a color.)** See **Color (6.9.33)** command.
Elevation	Changes the elevation (z coordinate); prompts you: **New elevation** - **(enter a positive or negative number change the elevation above the x,y-plane)**.See **Elev (6.13.4)** command.
LAyer	Changes the layer to which the entities are assigned; layer name must exist in the drawing. Prompts you: **New layer** - **(enter the name of a layer).**See **Layer (6.20.2)** command.
LType	Changes the linetype; linetype must be loaded into the drawing. Prompts you: **New linetype** - **(enter the name of a linetype).**See **Linetype (6.20.33)** command.
ltScale	Changes the linetype scale; prompts you: **New linetype scale** - **(enter a number)**.See **LtScale** system variable.
LWeight	Changes the lineweight; prompts you: **Enter new lineweight** - **(enter a lineweight unit)**.See **LWeight (6.20.41)** command.
Thickness	Changes the thickness; prompts you: **New thickness** - **(enter a positive or negative number.)** See Elev (6.13.4) command.

Transparency	Changes the transparency; prompts you: **New transparency <ByLayer>** - **(enter a value between 0 and 90)**
Material	Applies a material; prompts you: **New material <ByLayer>** - **(enter the name of a material).** See theMaterial (6.21.11)command.
Annotative	Sets the annotative property; prompts you: **Make Annotative?[Yes/No] <No>** - **(enter Yes or No).**
Change Point	Changes the size of the entity, depending on what the entity is: • **Lines** - moves the endpoint closest to the pick point • **Arcs** - changes the length closest to the pick point • **Circles** - changes the diameter • **Text** - moves the text • **Blocks** - moves the block This option has no effect on polylines.

Procedures

- To change text entities (1.23.15)
- To relocate blocks (1.23.15)

Related Commands

ChProp (6.9.18) - changes the properties of entities in the command bar. LayCur (6.20.1) - moves the selected entities to the current layer. Properties (6.24.49) - changes the properties and sizes of entities in a palette.

6.9.11 ChapooDownload

Obsolete. Replaced by the CloudDownload (6.9.27) command.

6.9.12 ChapooLogoff

Obsolete. Replaced by the CloudLogoff (6.9.28) command.

6.9.13 ChapooLogon

Obsolete. Replaced by the CloudLogon (6.9.29) command.

6.9.14 ChapooOpen

Obsolete. Replace by the CloudOpen (6.9.31) command.

6.9.15 ChapooProject

Obsolete. Replaced by the CloudProject (6.9.30) command.

6.9.16 ChapooUpload

Obsolete. Replaced by the CloudUpload (6.9.32) command.

6.9.17 ChapooWeb

Obsolete.

Figure 6.502



6.9.18 ChProp

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Changes the properties of entities, with fewer options than the Change command (short for "change properties.") This command is largely superseded by the Properties panel.

Accessing the Command

command bar: **chprop : chprop You are prompted:** Select entities to change: **(Choose one or more entities)** Select entities to change: **(Press Enter to end entity selection)** Property to change [Color/LAyer/LType/ltScale/LWeight/Thickness/TR **(Enter an option, or else press Enter to exit the command)**

Command Options

Option	Description
Select entities to change	Choose one or more entities in the drawing.
Color	Changes the color of the entity; prompts you: **New color** - **(enter the name or number of a color)**.See **Color (6.9.33)** command.
Elevation	Changes the elevation (z coordinate); prompts you: **New elevation** - **(enter a positive or negative number change the elevation above the x,y-plane)**.See **Elev (6.13.4)** command.
LAyer	Changes the layer to which the entities are assigned; layer name must exist in the drawing. Prompts you: **New layer** - **(enter the name of a layer).**See **Layer (6.20.2)** command.
LType	Changes the linetype; linetype must be loaded into the drawing. Prompts you: **New linetype** - **(enter the name of a linetype).**See **Linetype (6.20.33)** command.
ltScale	Changes the linetype scale; prompts you: **New linetype scale** - **(enter a number)**.See **LtScale** system variable.
LWeight	Changes the lineweight; prompts you: **Enter new lineweight** - **(enter a lineweight unit).**See **LWeight (6.20.41)** command.
Thickness	Changes the thickness; prompts you: **New thickness** - **(enter a positive or negative number)**.See Elev (6.13.4) command.
Transparency	Changes the transparency; prompts you: **New transparency <ByLayer>** - **(enter a value between 0 and 90)**
Material	Applies a material; prompts you: **New material <ByLayer>** - **(enter the name of a material)**. See theMaterial (6.21.11)command.
Annotative	Sets the annotative property; prompts you: **Make Annotative?[Yes/No] <No>** - **(enter Yes or No)**.

Related Commands

Change (6.9.10) - changes the properties and sizes of entities in the command bar. LayCur (6.20.1) - moves the selected entities to the current layer. Properties (6.24.49) - changes the properties and sizes of entities in a palette.

Figure 6.503



6.9.19 ChSpace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Moves entities from paper space to model space and vice versa (short for "change space").

Accessing the Command

The command applies to layouts only. command bar: **chspace** menu: **Modify | Change Space Note:** This command can only be used in a Layout Tab ('Paper Space').This command operates in two modes:

Moving Entities from Paper Space to Model Space

Select the layout that contains the entities to be moved, and then enter the command: **: chspace You are prompted:** Select entities: **(Select an entity)** Entities in set: 1 Select entities: **(Select more entities, or else press Enter)** Select TARGET viewport and press ENTER to continue: **(Click inside a viewport, or press Enter to accept the currently selected viewport)** <number of objects> objects were moved from PAPER space to MODEL space. Objects were scaled to maintain visual appearance by a factor of <scale factor>. **Note:** The scale factor is the inverse of custom scale of the selected target viewport. For example, if the custom scale = 1/2 (0.5), then the scale factor = 2.

Moving Entities from Model Space to Paper Space

When in a layout, double-click a viewport in which the entities are that need to be moved, and then enter the command: **: chspace You are prompted:** Select entities: **(Select one or more entities)** Entities in set: 1 Select entities: **(Select more entities, or else press Enter to continue)** Select SOURCE viewport and press ENTER to continue: **(Click inside a viewport, or else press Enter to accept the currently selected viewport.)** <number of objects> objects were moved from MODEL space to PAPER space. Objects were scaled to maintain visual appearance by a factor of <scale factor> **Note:** The scale factor equals the custom scale of the selected source viewport.

Command Options

Option	Description
Select entities	Selects one or more entities to be moved from one space to the other. Press Enter to end entity selection.
Select TARGET viewport	Start in the source viewport in paper space, and then click the target in model space
Select SOURCE viewport	Start in the target viewport in model space, and then click the target in paper space

Related Commands

CopyClip (6.9.52) - copies entities to the Clipboard. CutClip (6.9.59) - copies entities to the Clipboard, and then erases them from the drawing. MSpace (6.21.34) - switches to model space. PasteClip (6.24.10) - pastes entities from the Clipboard into the drawing (short for "paste clipboard"). PSpace (6.24.54) - switches to paper space.

Figure 6.504



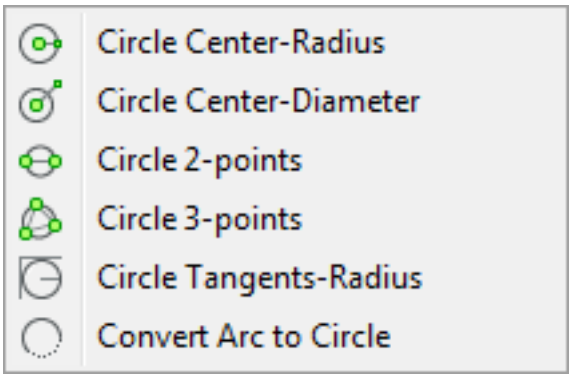
6.9.20 Circle

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) This command draws circles by two points, three points, and by tangent, tangent, and radius; it alsoconverts arcs to circles.

Accessing the Command

command bar: **circle** alias: **c** menu bar: **Draw | Circle** toolbar: **Draw |** ribbon:**Draw | Draw | Circles** : **circle You are prompted:** Select center of circle or [2Point/3Point/TTR/Arc/Multiple]: **(Pick a point, or enter an option)** Set Radius or [Diameter] <x.xx>: **(Pick a point, enter a distance, or type D to define the diameter)** The complete list of circle drawing methods is summarized by the Draw | Circle menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/CircleMenu.png}) In the icons, the larger green dot indicates the starting point, the smaller green dots the additional pick points. When an icon has all small dots, then the pick order does not matter.

Figure 6.505



Command Options

Option	Description
Select center of circle (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image97.png})	**Specifies the center of the circle:** • Enter x,y,z coordinates • Pick a point in the drawing The z coordinate is rarely entered, so most often you enter x,y coordinates.
Diameter	After you pick the center point, this prompt asks you to specifies the diameter of the circle: **Diameter of circle** - **(enter a distance, coordinates, or pick a point)**
Set Radius	After you pick the center point, this prompt asks you to specify the radius of the circle: **Radius of circle** - **(enter a distance, coordinates, or pick a point)**
2Point (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image98.png})	Defines the circle by two points that you pick on its circumference. prompts you: 1. **First point on diameter** - **(enter coordinates, or pick a point)** 2. **Second point on diameter** - **(enter coordinates, or pick another point)** The abbreviation for this option is 2P.
3Point (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image99.png})	Defines the circle by three points that you pick on its circumference. prompts you: 1. **First point on circle** - **(enter coordinates, or pick a point)** 2. **Second point** - **(enter coordinates, or pick another point)** 3. **Third point** - **(enter coordinates, or pick another point)** The abbreviation for this option is 3P.
RadTanTan (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image102.png})	Defines the circle by two points that are tangent to other entities with the radius. A **tangent** is where a circle touches another entity. The **Tangent (6.29.9)** entity snap is activated automatically during the first two prompts: 1. **Select first tangent point** - **(enter coordinates, or pick an entity)** 2. **Select second tangent point** - **(enter coordinates, or pick another entity)** 3. **Radius of circle** - **(specify a distance, or pick two points)** The abbreviation for this option is RTT.
Arc	Converts arcs to circles by closing them; prompts you: **Select arc to convert to circle** - **(Choose an arc)**
Multiple	Repeats the command until you press **Esc**.

Procedures

- General procedure to draw a circle (1.17.15)

- Editing a circle (1.17.15)

Grips Editing

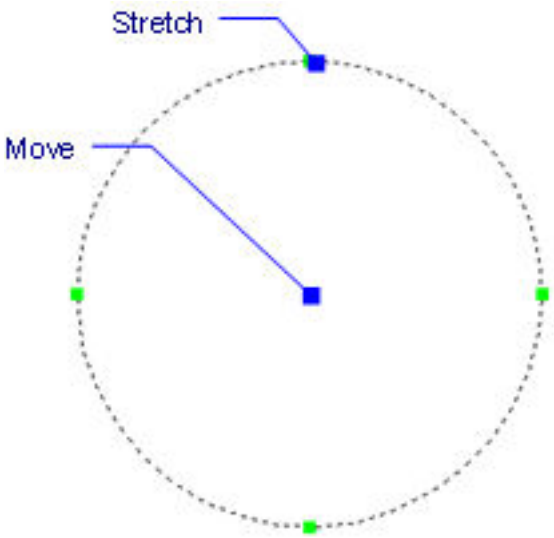
Circles can be edited directly through grips:

1. Select the circle. Notice that it has five grips.
2. Drag the grips to perform the following actions:

- Drag the quadrant grips to stretch the circle (increase and decrease the radius).
- Drag the center point grip to move the circle.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image463.jpg})

Figure 6.506



Related Commands

Ai_CircTan (6.5.1) - draws a circle tangent to three entities. Arc (6.4.12) - draws arcs. Change (6.9.10) - changes the diameter of circles. Ellipse (6.13.5) - draws ellipses. Join (6.18.1) - joins arcs to create circles. Donut (6.10.35) - draws fat circles from polyline arcs. DimRadius (6.11.17) and DimDiameter (6.11.10) - dimensions circles. Center (6.9.3) - snaps to the center of circles. Quadrant (6.25.7) - snaps to the 90-degree points on the circumference of circles. Tangent (6.29.9) - snaps to the tangency of circles.

Figure 6.507

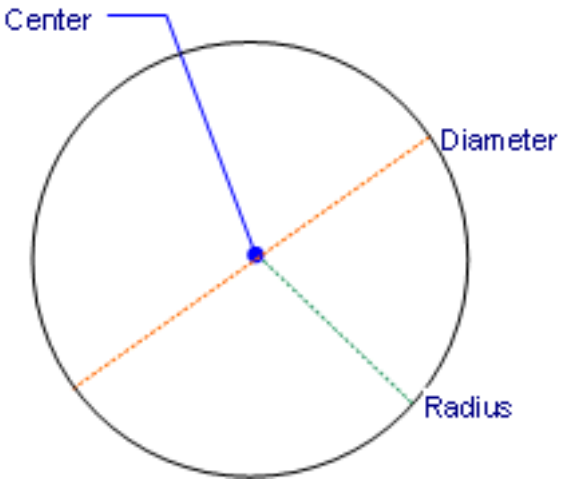


Figure 6.508

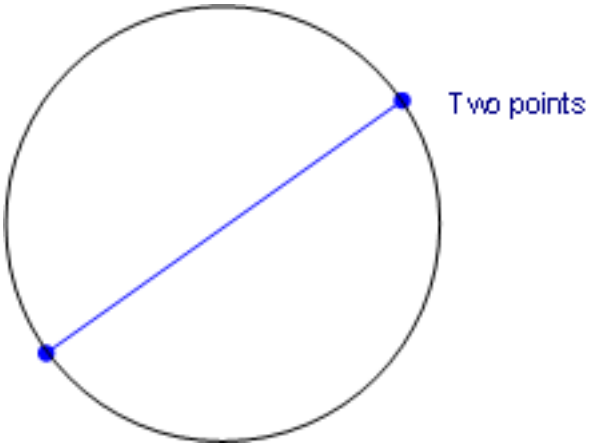


Figure 6.509

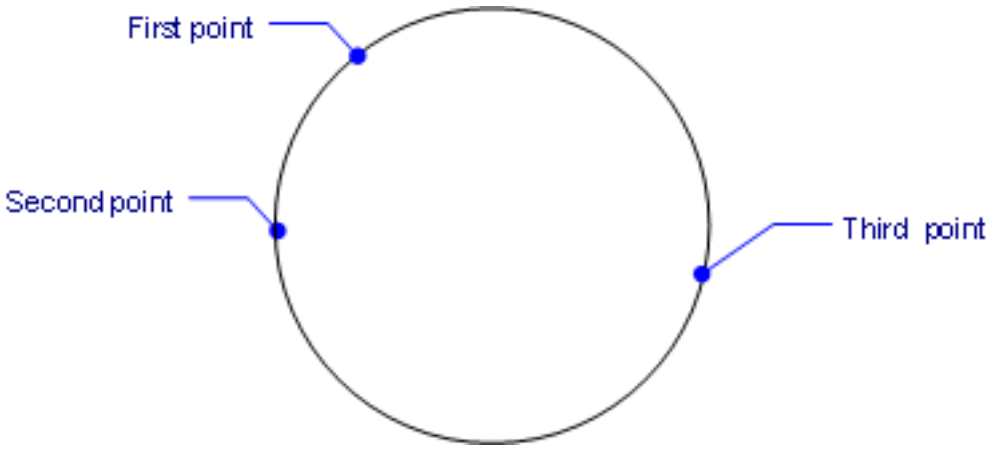


Figure 6.510

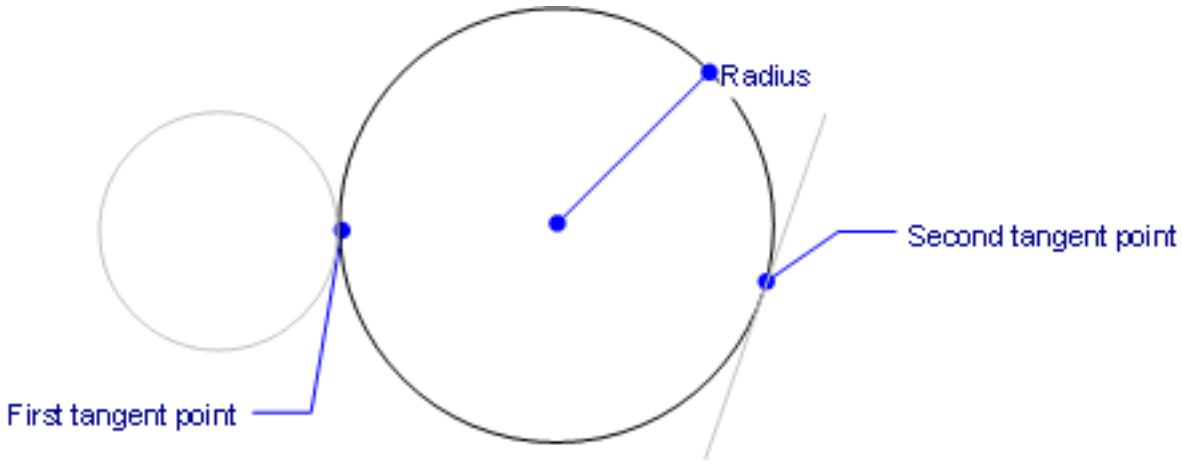


Figure 6.511



6.9.21 CleanScreenOff

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Displays user interface elements that were hidden by the **CleanScreenOn (6.9.22)** command.

Accessing the Command

 command bar: **cleanscreenoff** keyboard:Ctrl + 0(zero) menu bar:**View | Clean Screen** toolbar: **Standard** | quad: **General | Clean Screen** **Note:** Do not use the numeric keypad.On AZERTY-keyboards: do not press the **Shift**key when hitting the number key. : **cleanscreenoff** Displays no prompts in the command bar.

Related Commands

CleanScreenOn (6.9.22) - hides elements of the user interface. CommandLine (6.9.34) - displays the command bar. CommandLineHide (6.9.36) - hides the command bar. Ribbon (6.26.31) - displays the ribbon. RibbonClose (6.26.33) - hides the ribbon. StatBar (6.27.45) - toggles the display of the status bar.

Figure 6.512



6.9.22 CleanScreenOn

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Makes the drawing area larger by hiding elements of the user interface, as specified by the CLEANSCREENOPTIONS system variable.

Accessing the Command


command bar: **cleanscreenoff** keyboard:Ctrl + 0(zero) menu bar:**View | Clean Screen** toolbar: **Standard |** quad: **General | Clean Screen** Do not use the numeric keypad.On AZERTY-keyboards: do not press the Shift-key when hitting the number key. **: cleanscreenon** Displays no prompts in the command bar: Elements in the user interface are hidden according to the values of the Clean Screen Options section of the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/CleanScreenOptions.png})

Figure 6.513

Clean screen options	0x007F (127)
1	☑ Hide document tabs
2	☑ Hide dockable panels
4	☑ Hide toolbars
8	☑ Hide ribbon
16	☑ Hide command line panel
32	☑ Hide status bar
64	☑ Hide menu bar

CleanScreenOptions Variable Options

Option	Description
Hide document tabs	The display of document tabs is controlled by the ShowDocTabs user preference.See Working with document tabs (1.12.2) in the User Guide.
Hide dockable panels	Hides all dockable panels, such as the Command Bar, BIM Compositions, Content Browser, Layers, Mechanical Browser, Properties Bar, Render Materials, Ribbon, Sheet Sets, Structure, Tool Palettes. These can be toggled independently of the ClearScreenOn command by the +Toolbars command.See the Dockable Panels (1.12.7) topic in the User Guide.
Hide toolbars	Hides all toolbars.See the Toolbars (1.12.7) topic in the User Guide.
Hide ribbon	Hides the ribbon, which can be turned on and off independently by theRIBBON (6.26.31) and RIBBONCLOSE (6.26.33) commands.
Hide command line panel	Hides the command bar, which can be turned on and off independently by theCOMMANDLINE (6.9.34) and COMMANDLINEHIDE (6.9.36) commands.
Hide status bar	Hides the status bar, which can be turned on and off independently by theSTATBAR (6.27.45) command.The current status of the status bar is saved through the WNDLSTAT system variable.
Hide menu bar	Hides the menu bar, which can be turned on and off independently by the MENUBAR system variable.

Related Commands

CleanScreenOff (6.9.21) - displays user interface elements that were hidden by the CleanScreenOn (6.9.22) command. CommandLine (6.9.34) - displays the command bar. CommandLineHide (6.9.36) - hides the command bar. Ribbon (6.26.31) - displays the ribbon. RibbonClose (6.26.33) - hides the ribbon. StatBar (6.27.45) - toggles the display of the status bar.

Figure 6.514



6.9.23 CleanUnusedVariables

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Purges parametric variables not used by constraint expressions and not linked to dimensions.

Accessing the Command

 command bar: **cleanunusedvariables** menu bar:**Parametric | Delete Unused Parameters** toolbar: **Parameters |**  ribbon:**Solids | 3D Constraints | Delete Parameter** : **cleanunusedvariables** When used parametric variables are cleaned, reports: **** 9 unused variables were removed** When no unused variables exist in the drawing, this message is displayed: **** No unused variables detected**

Related Commands

Parameters (6.24.5) - manages dimensional constraint parameters and user variables in the current drawing. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. DcAligned (6.10.5) - constrains the distance between two points. DcAngular (6.10.6) - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. DcConvert (6.10.7) - converts an associative dimension to a dimensional constraint. DcDiameter (6.10.8) - constrains the diameter of a circle or an arc. DcHorizontal (6.10.10) - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. DcLinear (6.10.39) - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. DcRadius (6.10.11) - constrains the radius of a circle or an arc. DcVertical (6.10.12) - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.515



6.9.24 ClipDisplay

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Toggles the Clip Display property of section planes and BIM section entities.

Tutorial: 3D Modeling at a Clip Click here (1.6.31) to watch

Accessing the Command

 command bar: **clipdisplay** menu bar:**Model | Direct Modeling | Sections | Clipdisplay** toolbar: **Sections** |  ribbon:**Model | View | Clip Display** mouse: double-click a section entity **: clipdisplay You are prompted:** Select a section entity: **(Select a section plane or a BIM section entity)**

Command Options

Option	Description
Select a section entity	Toggles the display of the section plane: if on, then the clip is turned off; if off, then it is turned on.

Note: You can also toggle the display of the section entity through the Properties panel.

Procedures

- Using Clip Display (2.12.1)

Related Commands

BimSection (6.7.27) - creates a BIM section entity. LiveSection (6.20.34) - toggles the Live Section property of a section plane. SectionPlane (6.27.12) - creates a section entity that creates sections of 3D solids.

Figure 6.516



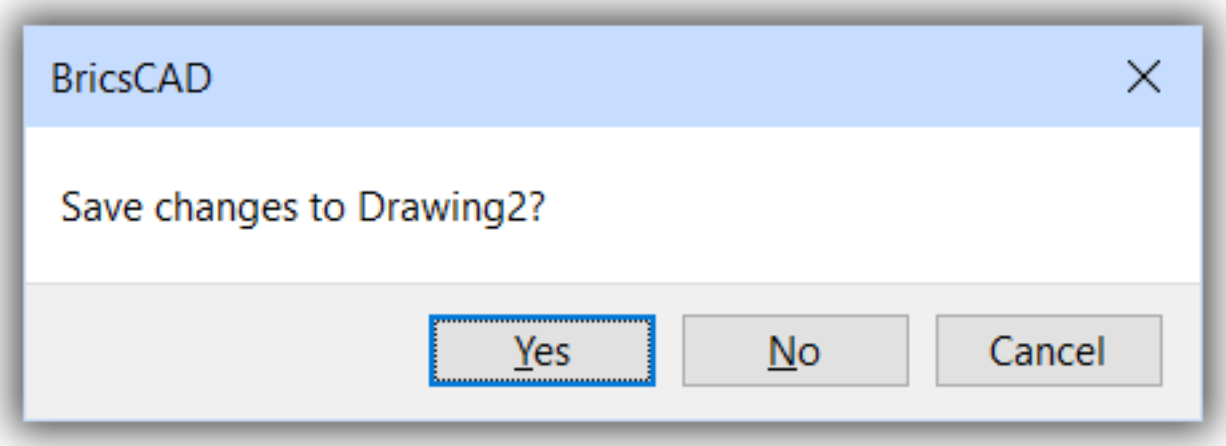
6.9.25 Close

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Closes the current drawing.

Accessing the Command

command bar: **close** keyboard shortcut: **Ctrl+F4** menu bar: **File | Close** : **close** If the drawing needs to be saved before closing, displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/closdlg.png}) Click the Yes, No, or Cancel button.

Figure 6.517



Command Options

Option	Description
Yes	Saves the drawing before closing it.
No	Does not save the drawing, and then closes it.
Cancel	Cancels the command and returns to the drawing.

Procedures

- Working with document tabs (1.12.2)

Related Commands

Open (6.23.8) - opens drawing files. WClose (6.33.4) - closes the current window. Quit (6.25.8) - exits BricsCAD after closing open drawings.

Figure 6.518



6.9.26 CloudAccount

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Reportsin the command bar whether the computer is logged into a Bricsys 24/7 account; formerly the ChapooAccount command.

Accessing the Command

 command bar: **cloudaccount** toolbar: **Bricsys 24/7 |**  ribbon:**Home | Bricsys 24/7 | Account** : **cloudaccount** Reports in the command bar, if logged in: **Server: https://my.bricsys247.com User name: your.name@email.com** If not logged into a Bricsys 24/7 account: **Bricsys 24/7: not logged in**

Related Commands

CloudDownload (6.9.27) - downloads a drawing from your Bricsys 24/7 project to a local folder. CloudLogoff (6.9.28) - logs off from Bricsys 24/7. CloudLogon (6.9.29) - logs on to Bricsys 24/7. CloudOpen (6.9.31) - downloads and opens a drawing from Bricsys 24/7. CloudProject (6.9.30) - opens your Bricsys 24/7 project in a browser. CloudUpload (6.9.32) - uploads the current drawing to Bricsys 24/7.

Figure 6.519



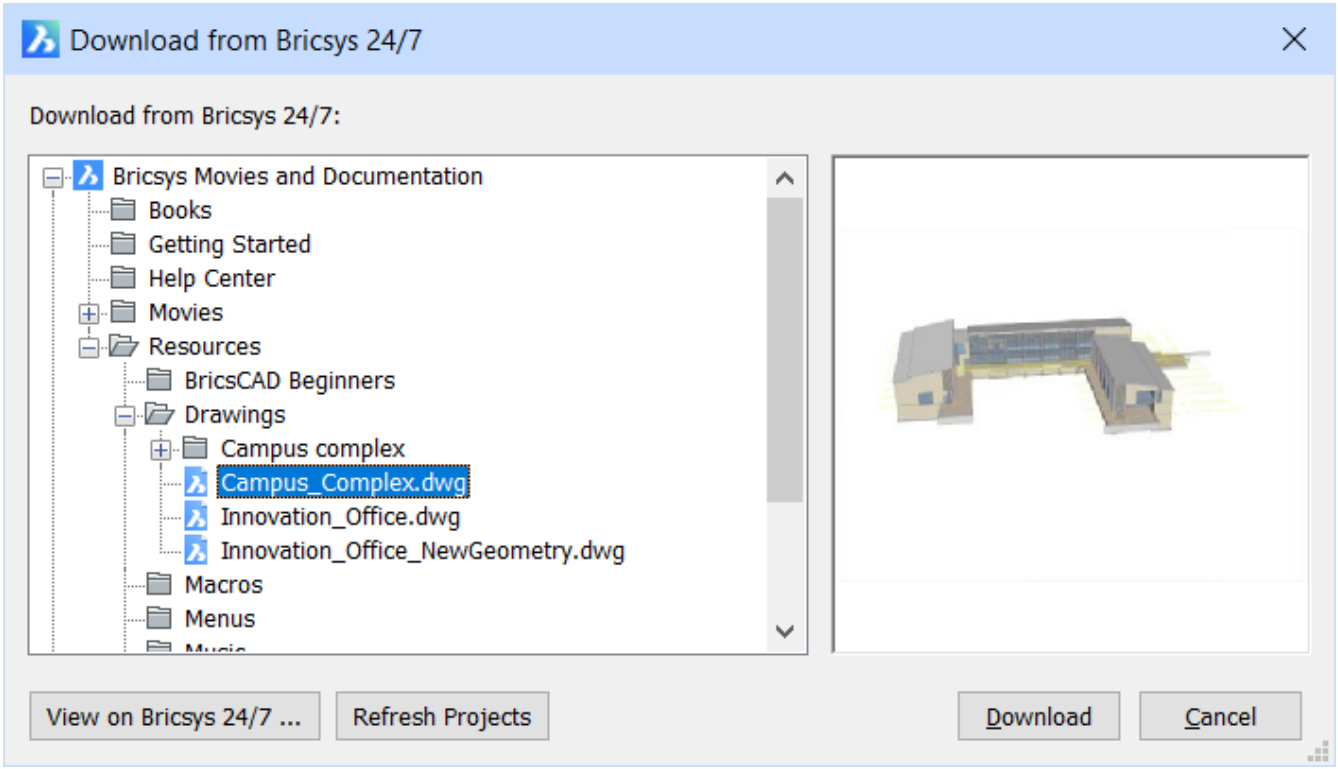
6.9.27 CloudDownload

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Opens a drawing from your Bricsys 24/7 project by downloading it to a local folder; replaces the ChapooDownload command.

Accessing the Command

 command bar: **clouddownload** menu bar: **File | Bricsys 24/7 | Download...** toolbar: **Bricsys 24/7** | ribbon:**Home | Bricsys 24/7 | File | Download** : **clouddownload** If you are not yet logged on to Cloud, the **CloudLogon (6.9.29)** command is launched first. Once you are logged into a Bricsys 24/7 account, displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/clouddownload-dlg.png})

Figure 6.520



Command Options

Option	Description
View on Bricsys 24/7...	Opens Bricsys 24/7 in your computers' default Web browser.
Refresh Projects	Reloads the project list.
Download	Downloads the selected document to a local folder; opens the File Save As dialog box.

Related Commands

CloudAccount (6.9.26) - reports the current status of your Bricsys 24/7 account in the command bar. CloudLogoff (6.9.28) - logs off from Bricsys 24/7. CloudLogon (6.9.29) - logs on to Bricsys 24/7. CloudOpen (6.9.31) - downloads and opens a drawing from Bricsys 24/7. CloudProject (6.9.30) - opens your Bricsys 24/7 project in a browser. CloudUpload (6.9.32) - uploads the current drawing to Bricsys 24/7.

Figure 6.521



6.9.28 CloudLogoff

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Logs you off from the Bricsys 24/7 account.

Accessing the Command

command bar: **cloudlogoff** menu bar:**File | Bricsys 24/7 | Logoff** : **cloudlogoff** Logged off.

Related Commands

CloudAccount (6.9.26) - reports the current status of your Bricsys 24/7 account in the command bar. CloudDownload (6.9.27) - downloads a drawing from your Bricsys 24/7 project to a local folder. CloudLogon (6.9.29) - logs on to Bricsys 24/7. CloudOpen (6.9.31) - downloads and opens a drawing from Bricsys 24/7. CloudProject (6.9.30) - opens your Bricsys 24/7 project in a browser. CloudUpload (6.9.32) - uploads the current drawing to Bricsys 24/7.

Figure 6.522



6.9.29 CloudLogon

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Logs your computer on to a Bricsys 24/7 account. Other commands, such as CloudUpload also prompt you to log-on, if necessary.

Accessing the Command

command bar: **cloudlogon** : **command** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
dlg.png}) When you click **Log In**, displays a second dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
login-2-dlg.png})

Figure 6.523

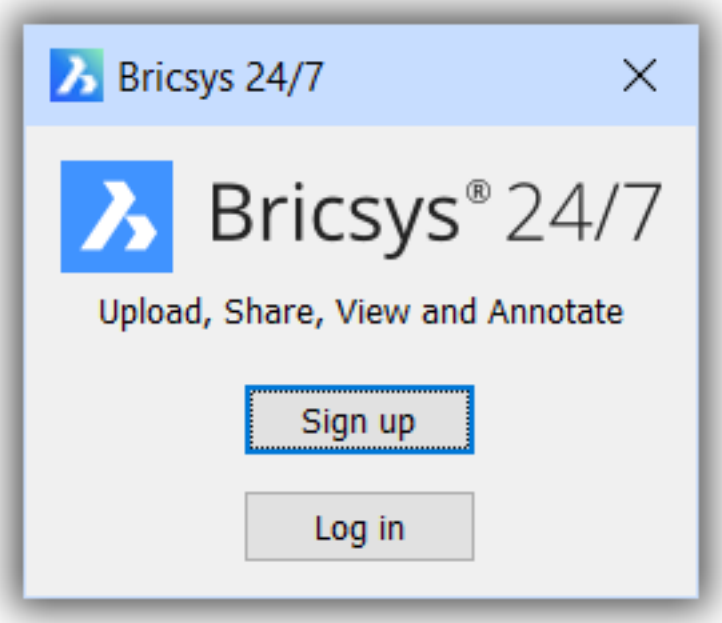
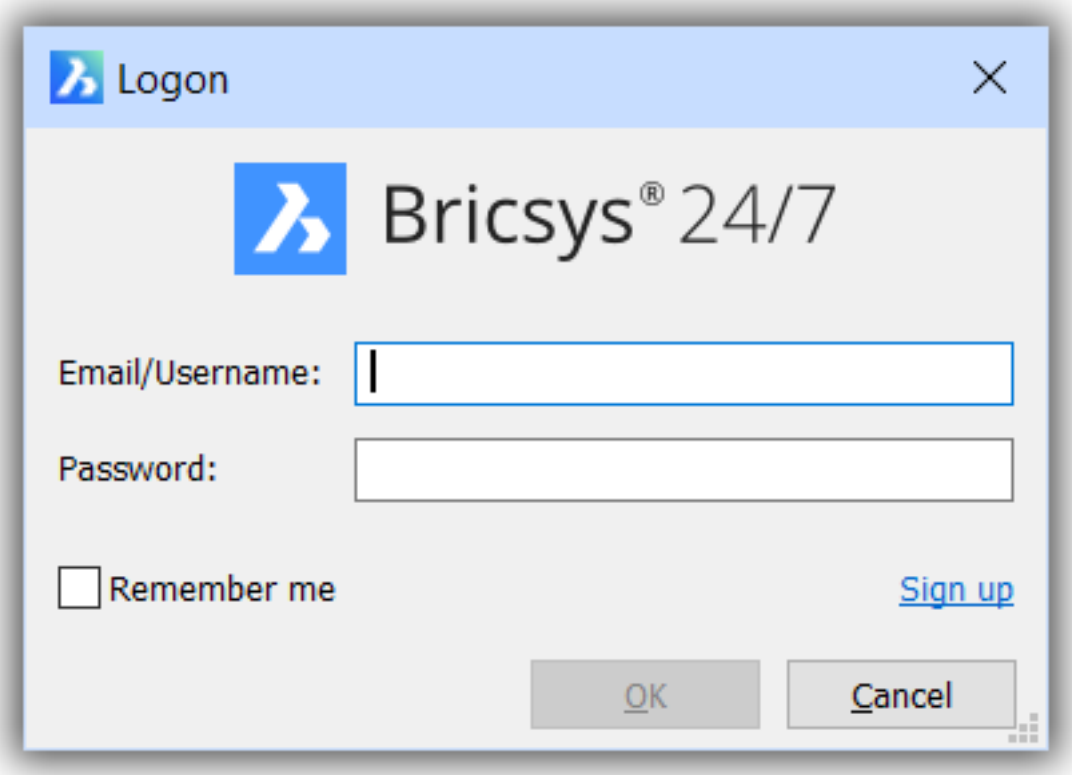


Figure 6.524



Command Options

Bricsys 24/7 Options	Description
Sign up	Opens your computer's default Web browser at thehttps://www.bricsys.com/en-intl/247/page
Log in	Displays the Login dialog box

Login Options	Description
Email/Username	Enter an email address attached to the Bricsys 24/7 account
Password	Enter the password attached to the email address
Remember Me	Skips this step the next time you use this command
Sign Up	Opens your computer's default Web browser at thehttps://www.bricsys.com/en-intl/247/page

Related Commands

CloudAccount (6.9.26) - reports the current status of your Bricsys 24/7 account in the command bar. CloudDownload (6.9.27) - downloads a drawing from your Bricsys 24/7 project to a local folder. CloudLogoff (6.9.28) - logs off from Bricsys 24/7. CloudOpen (6.9.31) - downloads and opens a drawing from Bricsys 24/7. CloudProject (6.9.30) - opens your Bricsys 24/7 project in a browser. CloudUpload (6.9.32) - uploads the current drawing to Bricsys 24/7.

Figure 6.525



6.9.30 CloudProject

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Opens a Bricsys 24/7 account in your computer's default Web browser; replaces the old ChapooProject command.

Accessing the Command

 command bar: **cloudproject** menu bar: **Bricsys 24/7 | My Bricsys 24/7...** toolbar: **Bricsys 24/7|**  ribbon:**Home | Bricsys 24/7 | My Bricsys** : **cloudproject** Displays the logon page to Bricsys 24/7 in a browser: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/cloud-project.png}) Once you logon, you are taken to your project area in Bricsys 24/7: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/project-2-dlg.png})

Figure 6.526

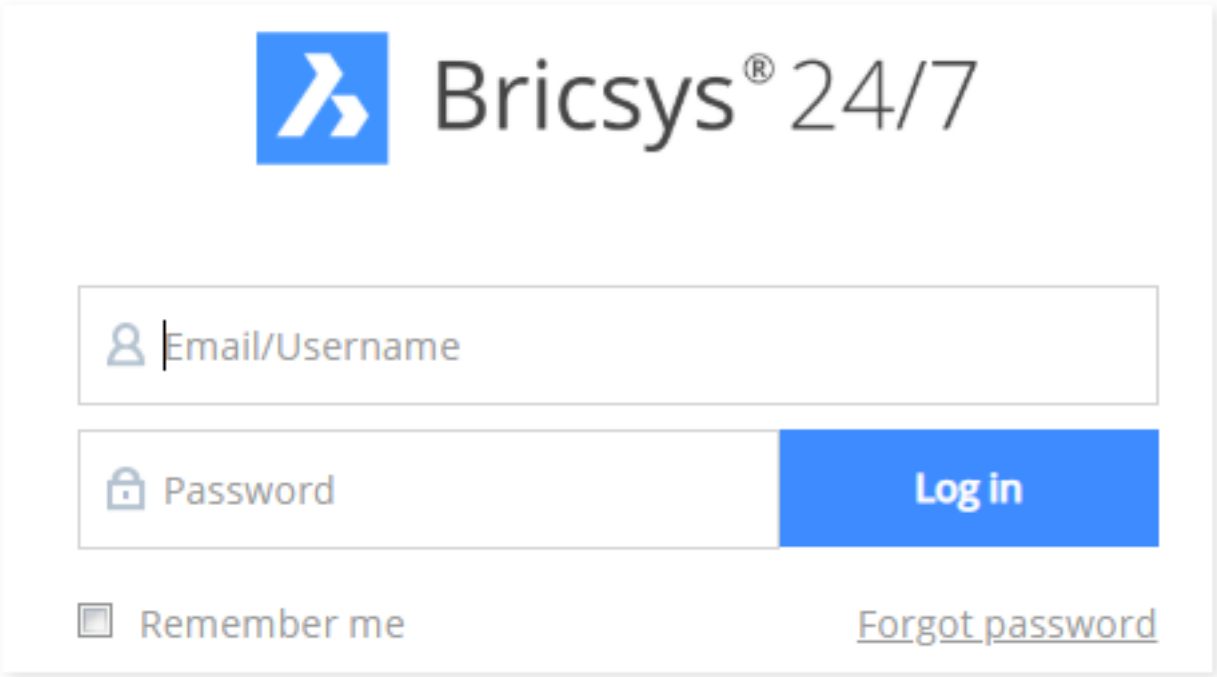
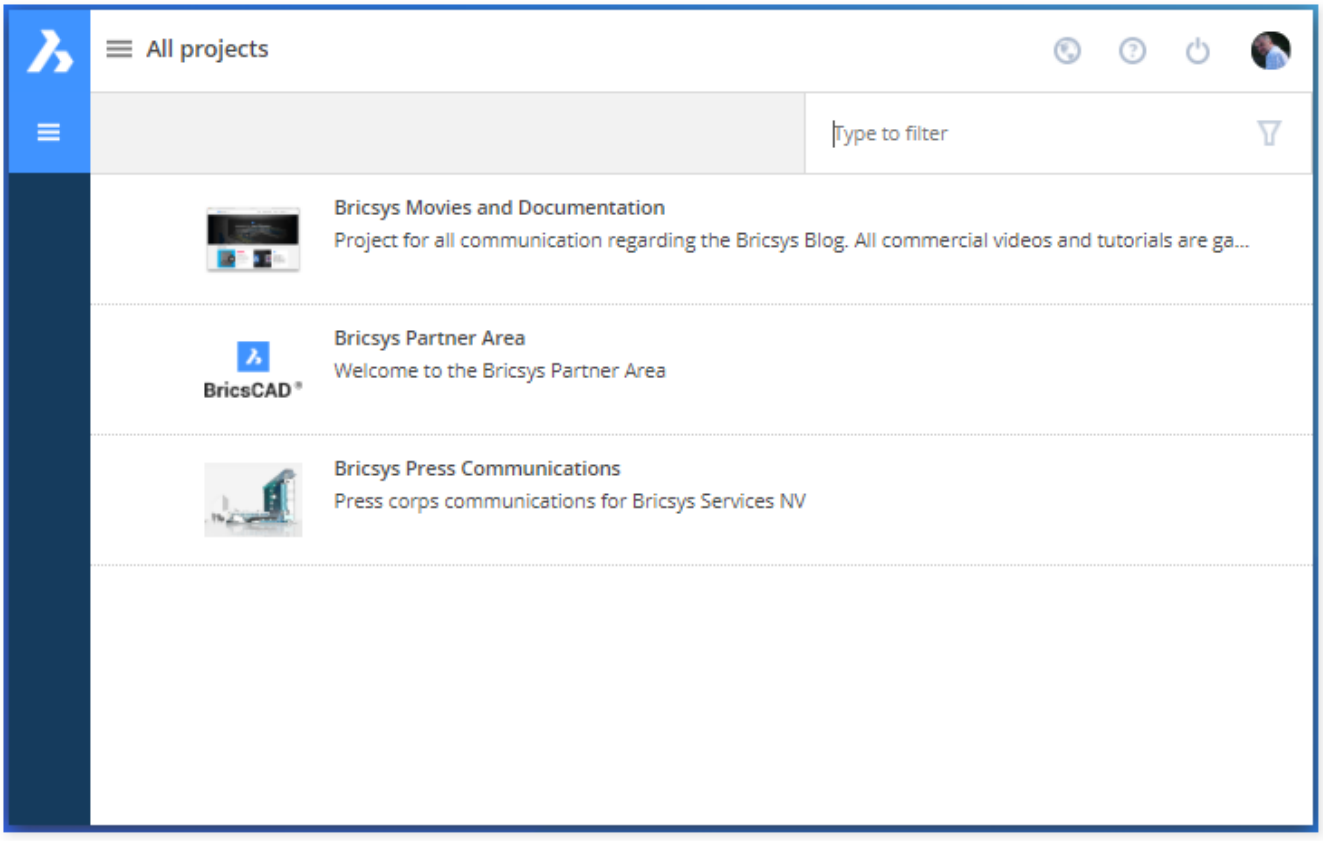


Figure 6.527



Command Options

Option	Description
Email/Username	Enter an email address attached to the Bricsys 24/7 account
Password	Enter the password attached to the email address
Remember Me	Skips this step the next time you use this command
Sign Up	Opens your computer's default Web browser at the https://www.bricsys.com/en-intl/247/ page

Related Commands

CloudAccount (6.9.26) - reports the current status of your Bricsys 24/7 account in the command bar. CloudDownload (6.9.27) - downloads a drawing from your Bricsys 24/7 project to a local folder. CloudLogoff (6.9.28) - logs off from Bricsys 24/7. CloudLogon (6.9.29) - logs on to Bricsys 24/7. CloudOpen (6.9.31) - downloads and opens a drawing from Bricsys 24/7. CloudUpload (6.9.32) - uploads the current drawing to Bricsys 24/7.

Figure 6.528



6.9.31 CloudOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Downloads and opens a drawing from a Bricsys 24/7 account; replaces the old ChapooOpen command.

Accessing the Command

 command bar: **cloudopen** menu bar: **File | Bricsys 24/7 | Open...** toolbar: **Bricsys 24/7** | ribbon:Home | Bricsys 24/7 | File | **Remote** :**cloudopen** If you are not yet logged on to Bricsys 24/7, then the program first launches theCloudLogon (6.9.29) command. When you are logged in, this command displays a dialog box listing the areas of Bricsys 24/7 to which you have access: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C 1-dlg.png}) Select a file, click **Open**, displays another dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e 2-dlg.png})

Figure 6.529

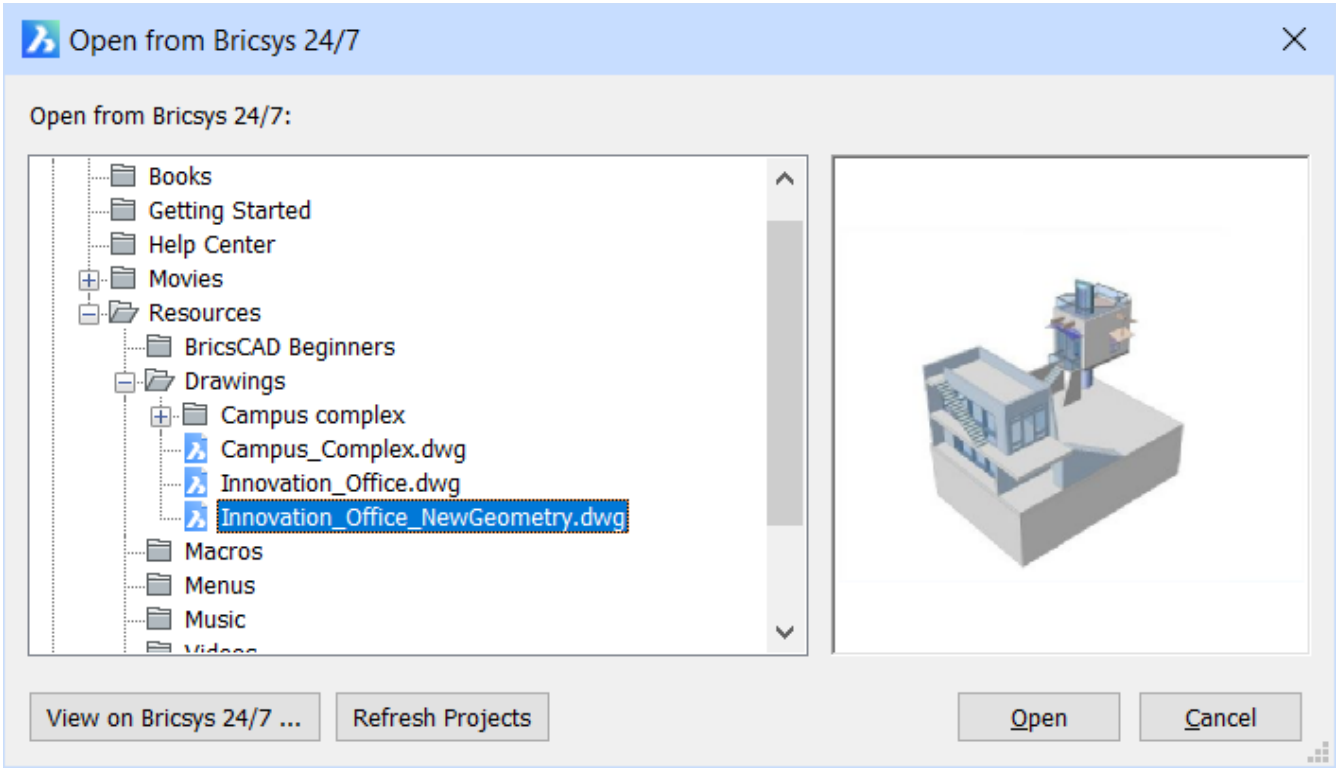
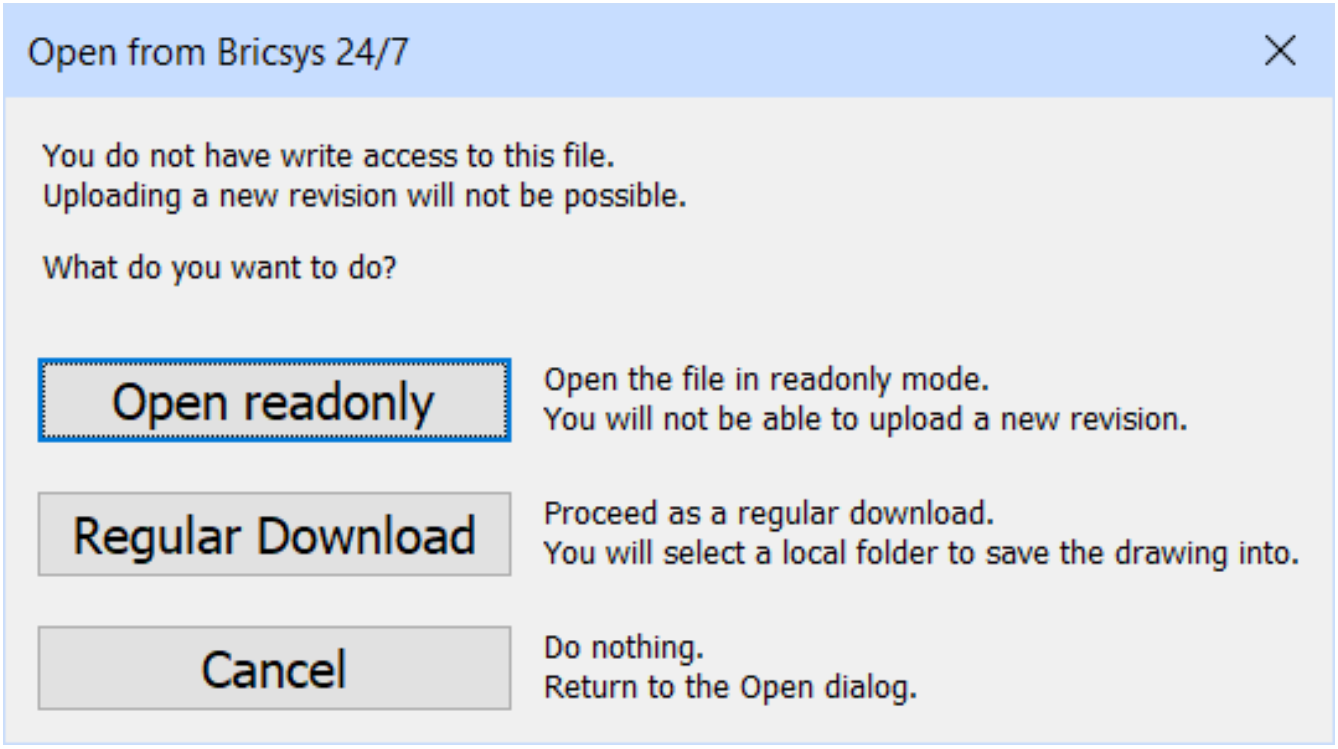


Figure 6.530



Command Options

Option	Description
View on Bricsys 24/7...	Opens Bricsys 24/7 in your computers' default Web browser.
Refresh Projects	Reloads the project list.
Open	Displays the second dialog box and opens the selected drawing.

Option	Description
Open Readonly	Downloads the file, and then opens it in BricsCAD as a read-only file, meaning that it cannot be saved by its filename. Use the SaveAs command to save it by another file name. The file is downloaded a temporary folder specified by the **CloudTempFolder**preference, which has an initial value of C:\Users\<login>\AppData\Local\Temp\Bricsys_24_7.
Regular Download	Opens the Save File As dialog box: 1. Choose a folder. 2. Optionally change the name of the drawing file. 3. Click **Save**. The program opens the drawing in BricsCAD.
Cancel	Cancels the download and exits the command.

Related Commands

CloudAccount (6.9.26) - reports the current status of your Bricsys 24/7 account in the command bar. CloudDownload (6.9.27) - downloads a drawing from your Bricsys 24/7 project to a local folder. CloudLogoff (6.9.28) - logs off from Bricsys 24/7. CloudLogon (6.9.29) - logs on to Bricsys 24/7. CloudProject (6.9.30) - opens your Bricsys 24/7 project in a browser. CloudUpload (6.9.32) - uploads the current drawing to Bricsys 24/7.

Figure 6.531



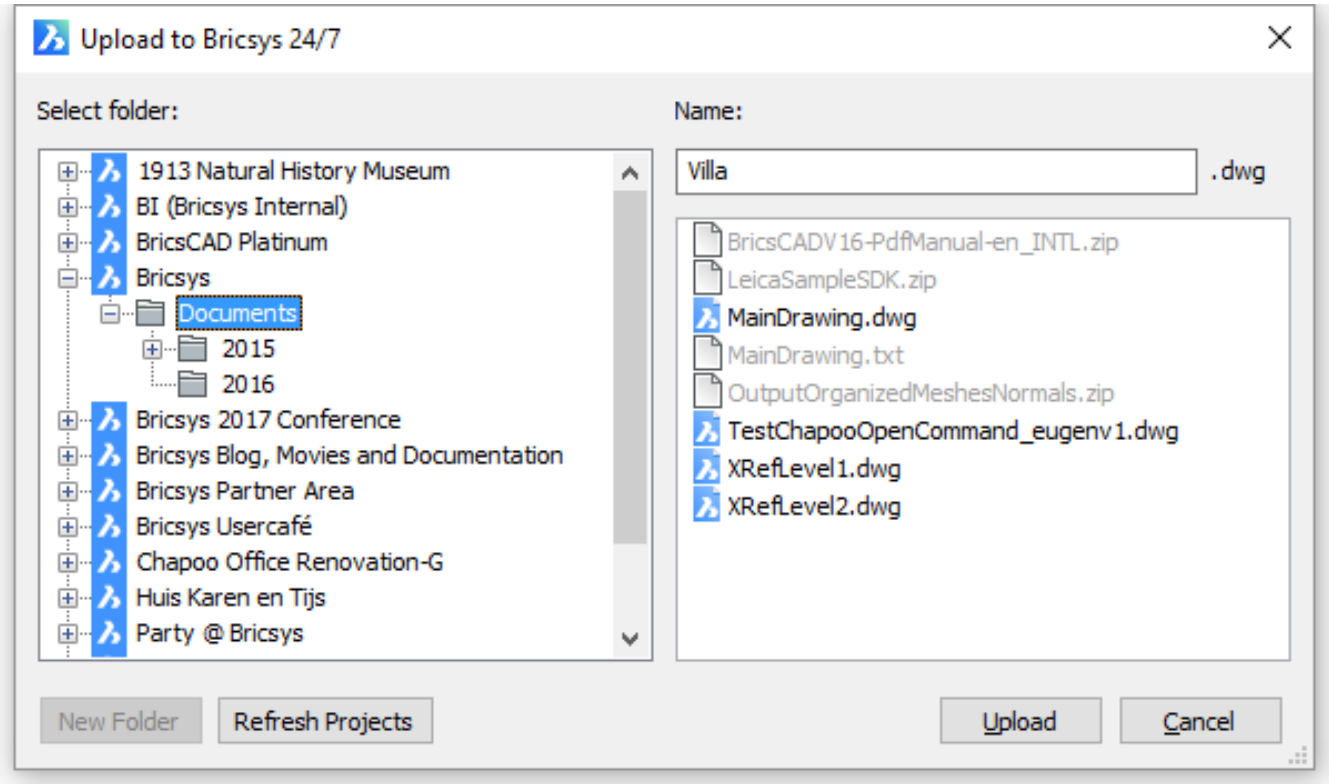
6.9.32 CloudUpload

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Uploads the current drawing to a project area on Bricsys 24/7.

Accessing the Command

 command bar: **cloudupload** menu bar: **File | Bricsys 24/7 | Upload...** toolbar: **Bricsys 24/7** |  ribbon:**Home | Bricsys 24/7 | File | Upload** : **cloudupload** If you are not yet logged on to Bricsys 24/7, the **CloudLogon (6.9.29)** command is launched first. Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
If you do not upload rights, then the folders are grayed out (unavailable to you). Choose a project, select a folder, and then click the **Upload** button.

Figure 6.532



Command Options

Option	Description
New Folder	Creates a new folder as a subfolder of the currently selected folder.
Refresh Projects	Reloads the project list.
Upload	Uploads the drawing to the selected folder. If a drawing of the same name already exists in the selected folder, the drawing is uploaded as a revision of the original drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C If the drawing is dependent from other files, such as external references or images, the **CloudUploadDependencies** user preference defines what to do: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C • 0: Upload the drawing only and ignore dependencies. • 1: Launch the eTransmit (6.13.9) command first to compose a zip-file if the drawing contains external references. • 2: Always launch eTransmit (6.13.9) first.

Related Commands

CloudAccount (6.9.26) - reports the current status of your Bricsys 24/7 account in the command bar. CloudDownload (6.9.27) - downloads a drawing from your Bricsys 24/7 project to a local folder. CloudLogoff (6.9.28) - logs off from Bricsys 24/7. CloudLogon (6.9.29) - logs on to Bricsys 24/7. CloudOpen (6.9.31) - downloads and opens a drawing from Bricsys 24/7. CloudProject (6.9.30) - opens your Bricsys 24/7 project in a browser. Etransmit (6.13.9) - creates a package of a drawing file and all its dependencies, such as: external references, images, font files, plot configuration files, plot style tables and font map files.

Figure 6.533

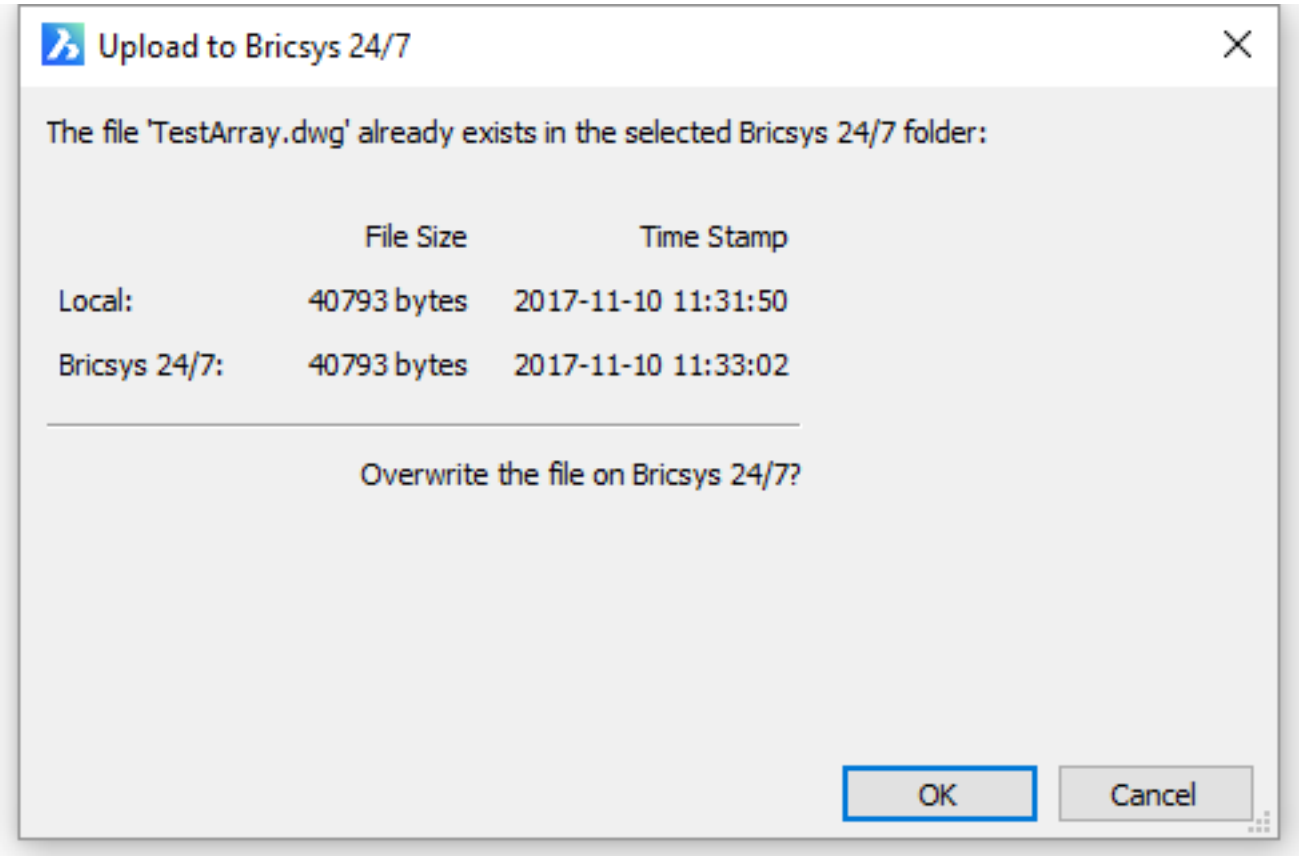


Figure 6.534

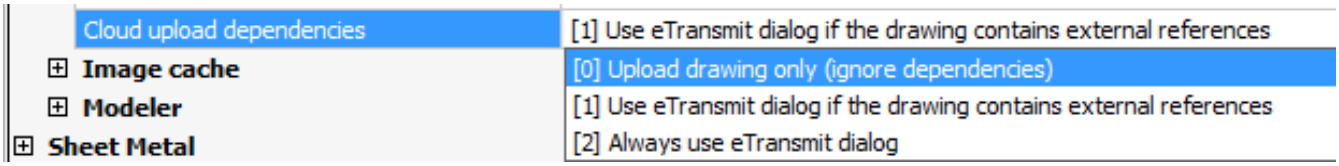


Figure 6.535



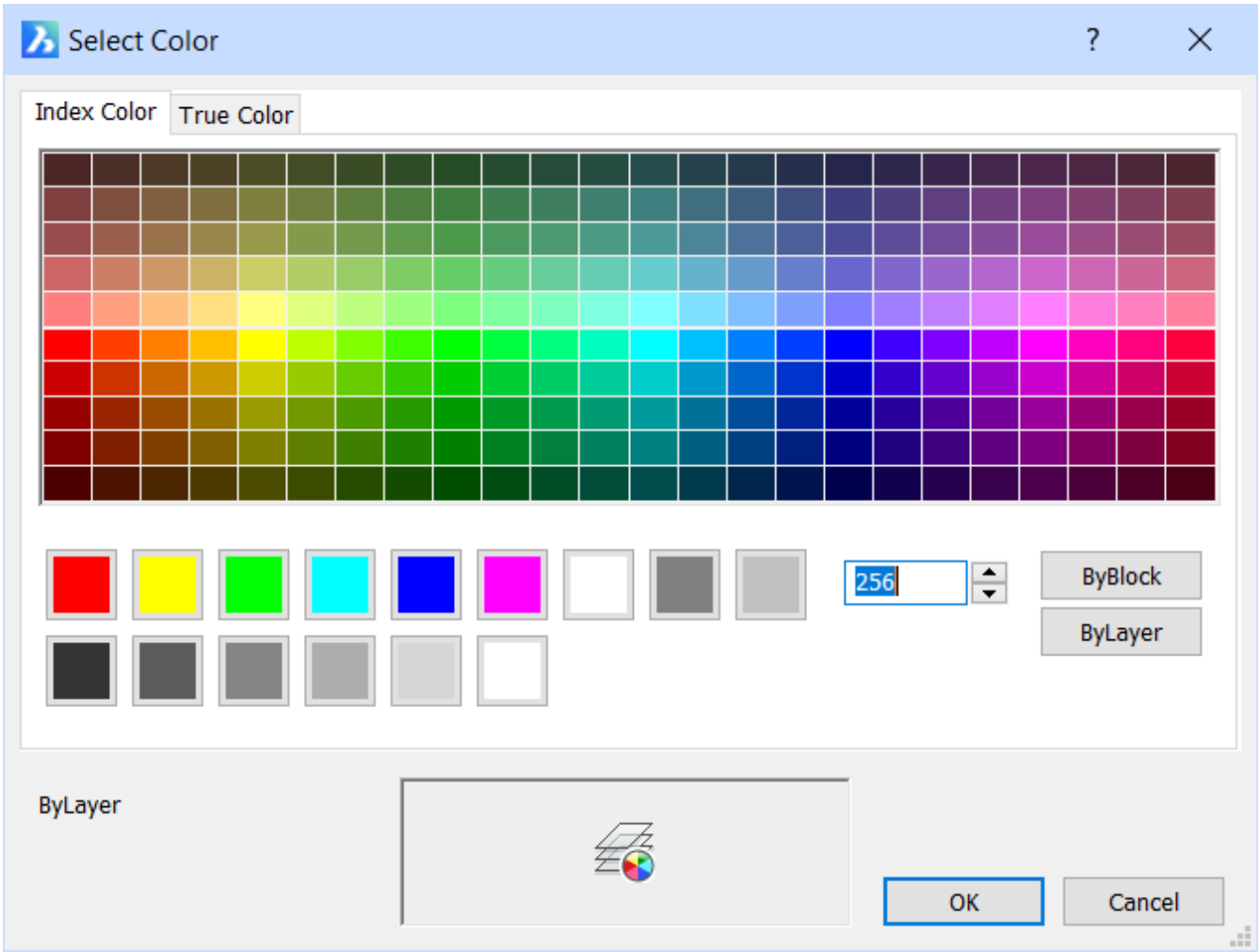
6.9.33 Color

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Specifies the current working color, through a dialog box.

Accessing the Command

 command bar: **color** transparent: **'color** aliases: **col, color, ddcolor, ddcolour, setcolor** menu bar: **Settings | Color | Select Color** toolbar: **Settings |**  status bar: **Color** field (**ByLayer or** ) ribbon: **Settings | Set Color** : **color** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/color-dlg.png}) Choose a color, and then click **OK**.

Figure 6.536


Select Color
Index Color
True Color
256
ByBlock
ByLayer
ByLayer
OK
Cancel

Command Options

Option	Description
Index Color tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
ByBlock	Sets the color to ByBlock, in which entities take on the color defined by their block.
ByLayer	Sets the color the ByLayer, in which entities take on the color defined by their layer's property.

Option	Description
True Color tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Hue	Specifies the color's hue; range is from 0 to 240. (Hue is the color, ranging from red through green, blue, and pink.)
Saturation	Specifies the intensity of the color; range is from 0 to 240. (Saturation is the intensity of the color, where 0 = gray and 240 = full color.)
Luminosity	Specifies the brightness of the color; range is from 0 to 240. (Luminosity is the lightness or darkness, where 0 = black and 240 = white. Hue, saturation, and luminosity work together, while red, green, and blue work separately.)
Red	Specifies the amount of red in the color; range is from 0 (no red) to 255 (full red).
Green	Specifies the amount of green in the color; range is from 0 (no green) to 255 (full green).
Blue	Specifies the amount of blue in the color; range is from 0 (no blue) to 255 (full blue).

Status Bar Menu

Right-click ByLayer (or other color name) on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbr
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image526.jpg})

Option	Description
ByLayer, ByBlock, Red, et cetera	Chooses the default color.
Properties	Opens the Select Color dialog box.

Figure 6.537

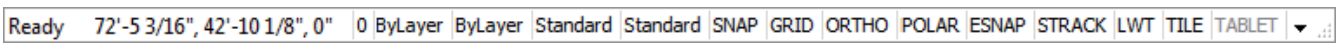


Figure 6.538



Related Commands

-Color (6.9.35) - sets the working color through the command bar. DimStyle (6.11.20) and -DimStyle (6.11.21) - specify colors for dimensions. Layer (6.20.2) and -Layer (6.20.3) - specify colors for layers. PlotStyle (6.24.29) - specifies how colors control plotted output. Settings (6.27.23) - specifies color-related parameters, such as the colors of grips, background, UCS and cursor axes, snap markers, and so on.

Figure 6.539

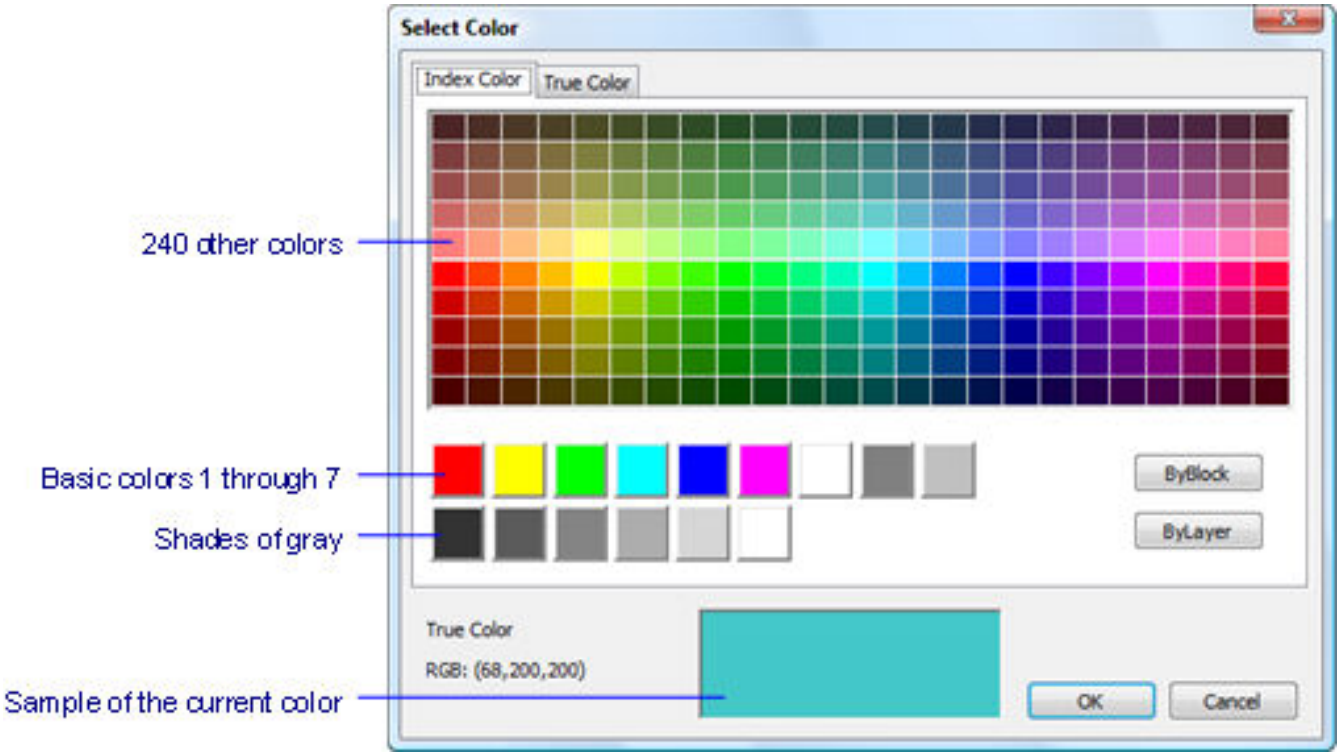


Figure 6.540

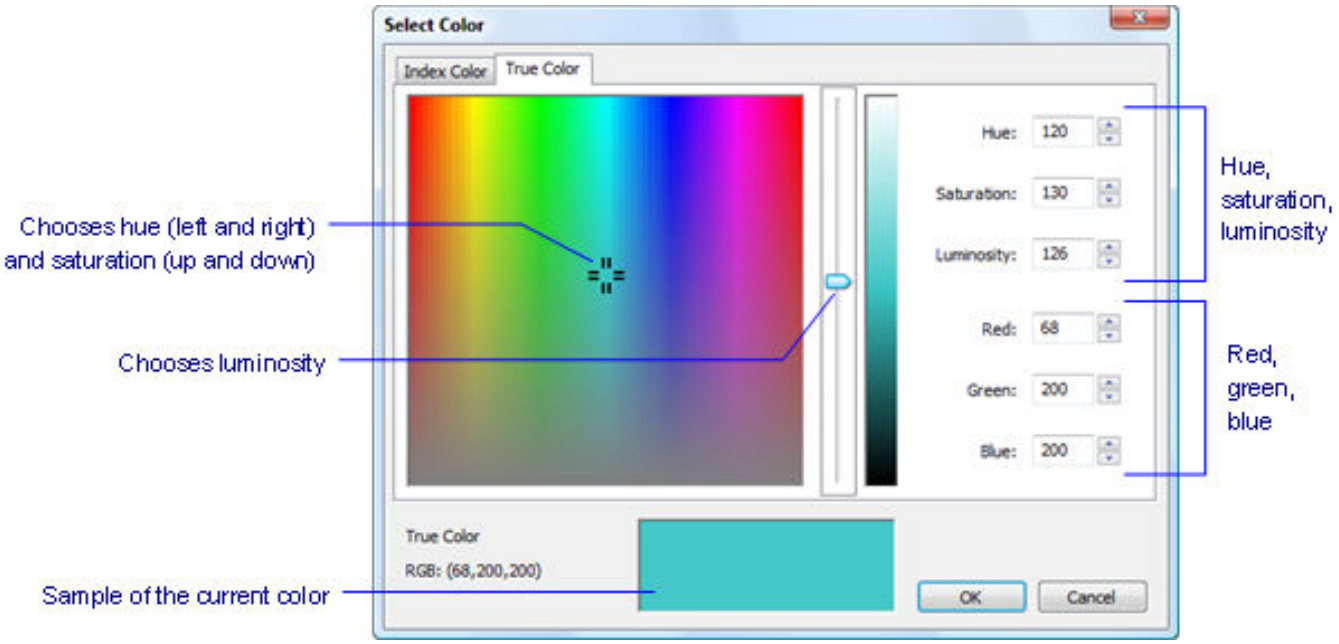


Figure 6.541



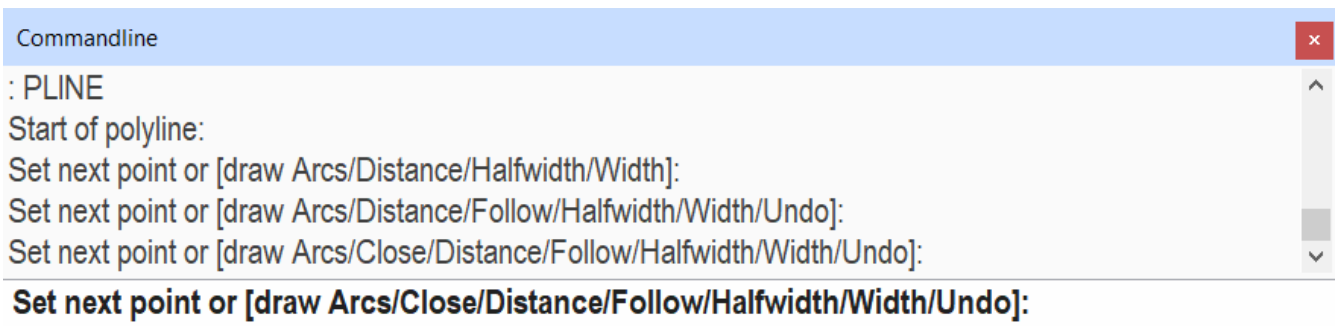
6.9.34 CommandLine

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Displays the command bar.

Accessing the Command

command bar: **commandline** menu bar: **View | Command Bar** keyboard shortcuts: **Shift+F2** and **Ctrl+9** mouse: double-click the left half of the status bar to toggle the display of the Command Bar : **commandline** Displays the command bar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/commandbar-panel.png})

Figure 6.542

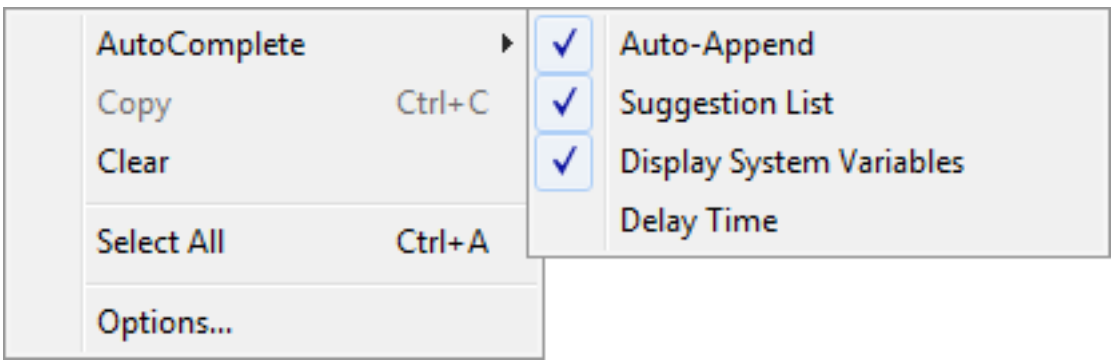


Shortcut Menu Options

Right-click to display the following context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cmd

Option	Description
AutoComplete	Auto-Append: If checked, automatically completes the entry if only one possible command is left. Suggestion List: If checked, displays a list of possible commands. Display System Variables: If checked, includes system variables in the suggestion list. Delay Time: Sets the time in seconds before the automated keyboard features display.
Copy	Copies the selected text to the Clipboard.
Clear	Clears the complete prompt history
Select All	Selects all the complete content of the prompt history
Options	Displays the **Settings** dialog box: Command line.

Figure 6.543



Related Commands

CleanScreenOff (6.9.21) - displays user interface elements that were hidden by te CleanScreenOn (6.9.22) command. CleanScreenOn (6.9.22) - hides elements of the user interface. CommandLineHide (6.9.36) - hides the command line. GraphScr (6.15.20) - brings the graphics window to the foreground. Scrollbar (6.27.10) - toggles the display of the scroll bars. StatBar (6.27.45) - toggles the display of the status bar. TextScr (6.29.13) - brings the text window to the foreground. Toolbar (6.29.18) and -Toolpanel (6.29.21) - toggle the display of toolbars.

Figure 6.544



6.9.35 -Color

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Sets the current working color, through the command bar.

Accessing the Command

command bar: **-Color** aliases: **-col, -colour** status bar: **ByLayer : -color You are prompted:** Color for drawing new entities <BYLAYER>: **(Enter a color name, number, or RGB value)**

Command Option

Option	Description
Color for new drawing entities	Enter the name, number, or RGB value of a color: • Names - specifies red (color #1) , yellow (2), green (3), cyan (4), blue (5), magenta (6), white (7), ByLayer (256), or ByBlock (0). • Numbers - specifies a number from 0 to 256. • RGB - specifies a value for each of red, green, and blue ranging from 0 to 255. For example, white is 255,255,255 and gray is 128,128,128.

Status Bar Menu

Right-click ByLayer (or other color name) on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbr (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image526.jpg})

Option	Description
ByLayer,ByBlock,Red,et cetera	Chooses the default color.
Properties	Opens the Select Color dialog box. See **Color (6.9.33)** command.

Figure 6.545

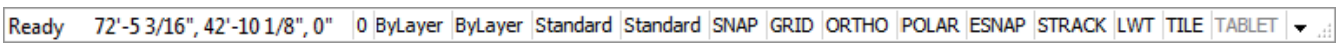


Figure 6.546



Related Commands

Color (6.9.33) - sets the working color through a dialog box. DimStyle (6.11.20) and -DimStyle (6.11.21) - specify colors for dimensions. Layer (6.20.2) and -Layer (6.20.3) - specify colors for layers. PlotStyle (6.24.29) - specifies how colors control plotted output. Settings (6.27.23) - specifies color-related parameters, such as the colors of grips, background, UCS and cursor axes, snap markers, and so on.

Figure 6.547



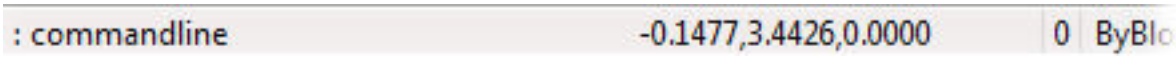
6.9.36 CommandLineHide

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Hides the command bar.

Accessing the Command

command bar: **commandlinehide** menu bar: **View | Command Bar** keyboard shortcuts: **Shift+F2** and **Ctrl+9** mouse: double-click the left half of the status bar to toggle the display of the Command Bar : **commandlinehide** Hides the command bar. Prompts are displayed on the status bar. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1

Figure 6.548



Related Commands

CleanScreenOff (6.9.21) - displays user interface elements that were hidden by te CleanScreenOn (6.9.22) command. CleanScreenOn (6.9.22) - hides elements of the user interface. CommandLine (6.9.34) - shows the command line. GraphScr (6.15.20) - brings the graphics window to the foreground. Scrollbar (6.27.10) - toggles the display of the scroll bars. StatBar (6.27.45) - toggles the display of the status bar. TextScr (6.29.13) - brings the text window to the foreground. Toolbar (6.29.18) and -Toolpanel (6.29.21) - toggle the display of toolbars.

Figure 6.549



6.9.37 Commands

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Lists the names of commands available in the program, first the name in English followed by the localized command name.

Tutorial: Unlock BricsCAD's extra features Click here (1.9.1) to watch

Accessing the Command

command bar: **commands : commands You are prompted:** List commands [Internal/External/Both] <Internal>: **(Press Enter to list internal commands only, or enter an option)**

Command Options

Option	Description
Internal	Lists command names native to BricsCAD.
External	Lists command names from add-on applications.
Both	List both internal and external command names.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. CuiLoad (6.9.56) and CuiUnload (6.9.57) - loads and unloads menu files. Customize (6.9.58) - customizes the user interface. MenuLoad (6.21.14) and MenuUnload (6.21.15) - loads and unloads menu files. Netload (6.22.3) - loads a .NET application. VbaLoad (6.32.2) and -VbaLoad (6.32.3) - loads VBA projects. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaMan (6.32.4) - manages VBA projects. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.550



6.9.38 ComponentsPanelClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes the Components panel.

Accessing the Command

command bar: **componentspanelclose** context menu: right-click any toolbar or ribbon, and then choose **Components**

Related Commands

ComponentsPanelOpen (6.9.61) -Opens the Components panel as a docked panel.

Figure 6.551



6.9.39 CommunicatorInfo

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Checks whether the Communicator (4.1.1) add-on is correctly installed and provides a diagnostic report. (The command is available on Windows only. The Communicator version must match the major version of BricsCAD.)

Accessing the Command

command bar: **communicatorinfo** : **communicatorinfo** Switches to thePrompt History (6.29.13) window, and then provides a report like this one:(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/commundlg.png}) When Communicator is not installed, then the command reports: **Brief check result: Communicator installation: FAILED**

Figure 6.552

```
: COMMUNICATORINFO

Brief check result: No problems found.

A BricsCAD license level (Pro or Platinum): OK
Communicator installation: OK
Communicator license: OK
Communicator module loading: OK
UseCommunicator setting (1 or 2): OK
   (Change in the settings affects only the next launch
    The value depends on license level)
MSVC runtime version 10: OK.
MSVC runtime version 11: OK.
MSVC runtime version 12: OK.
BricsCAD path: 'C:\Program Files\Bricsys\BricsCAD V18 en_US\'.
ACIS: OK.
ACIS: dll product version '20181.0.1'.
Communicator installation path: 'C:\Program Files\Bricsys\BricsCAD Communicator V18\'.
Communicator InterOp check: OK.
Communicator Connect license check: OK.
Communicator UConnect license check: OK.
Using PS Direct translation mode.
```

Related Commands

LicenceManager (6.20.22) - displays the status of all licenses associated with the program. Import (6.17.10) - Communicator adds file formats to import into the current drawing. Export (6.13.15) - Communicator adds file formats in which to export the current drawings.

Figure 6.553



6.9.40 ContentBrowserClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes the Content Browser panel.

Accessing the Command

command bar: **contentbrowserclose : contentbrowserclose**

Related Commands

ContentBrowserOpen (6.9.41) - opens the Content Browser panel.

Figure 6.554



6.9.41 ContentBrowserOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Content Browser panel.

Accessing the Command

command bar: **contentbrowseropen** context menu*: **Content Browser** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/
* Place the cursor over a toolbar or a ribbon panel, then right click. : **contentbrowseropen** Displays the Content Browser panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ContentBrowserPane

Figure 6.555

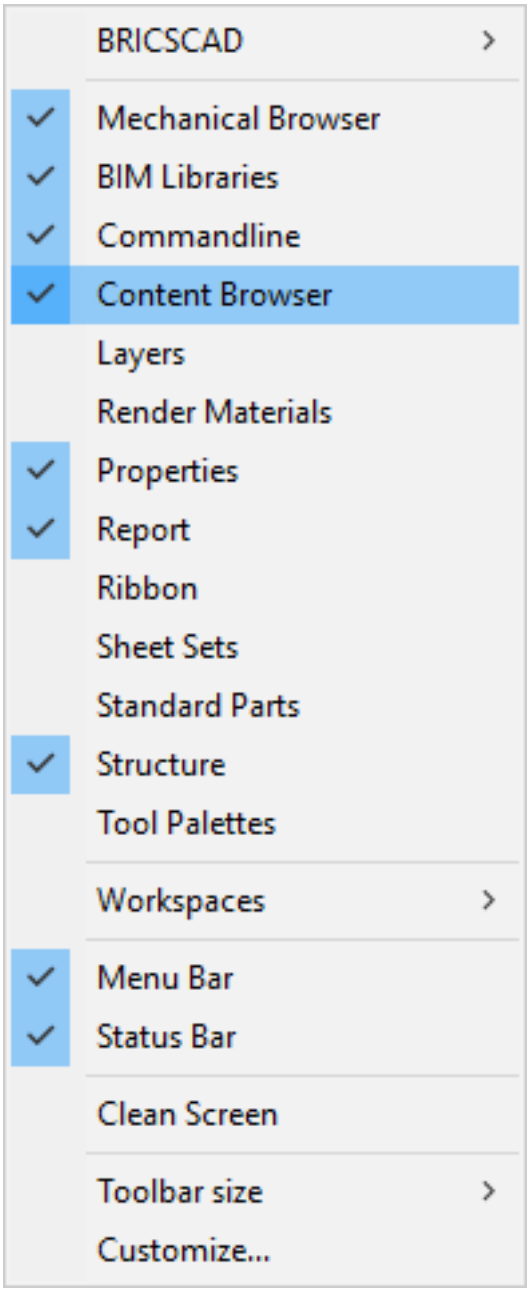
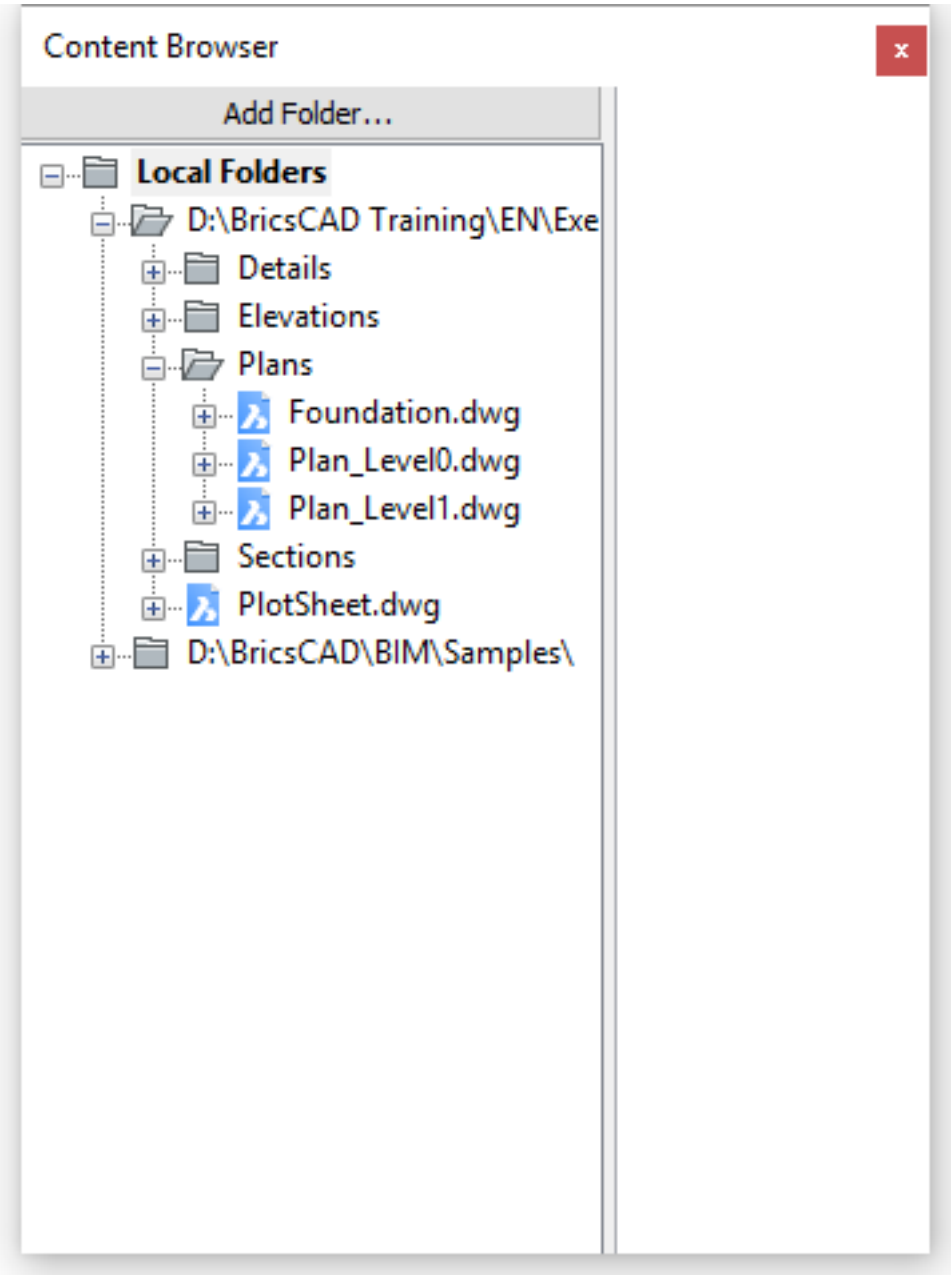

BRICSCAD
Mechanical Browser
BIM Libraries
Commandline
Content Browser
Layers
Render Materials
Properties
Report
Ribbon
Sheet Sets
Standard Parts
Structure
Tool Palettes
Workspaces
Menu Bar
Status Bar
Clean Screen
Toolbar size
Customize...

Figure 6.556


Content Browser
Add Folder...
Local Folders
D:\BricsCAD Training\EN\Exe
Details
Elevations
Plans
Foundation.dwg
Plan_Level0.dwg
Plan_Level1.dwg
Sections
PlotSheet.dwg
D:\BricsCAD\BIM\Samples\

Command Options

Option	Description
Add Folder...	Adds a folder to the **Local Folders** tree. Do one of the following: • Click the **Add Folder...** button. • Right click on the **Local Folders** node, then choose **Add folder** in the context menu.
Remove Folder	Removes a folder from the **Local Folders** tree. Right click the folder you want to remove and choose **Remove folder** in the context menu.
Open Drawing	Opens a drawing in one of the folders in the **Local Folders** tree. Do one of the following: • Double click the drawing name. • Right click the drawing name and choose **Open drawing** in the context menu.
Insert Block	Executes the -Insert (6.17.14) command. Do the following: • Expand the source drawing in the **Local Folders** tree. • Click **Blocks** . • Select the block, then do one of the following: • Drag the block onto the current drawing. • Right click and choose **Insert Block** in the context menu.
Copy Dimension Style	Copies the selected dimension style(s) to the current drawing. Do the following: • Expand the source drawing in the **Local Folders** tree. • Click **Dimension Styles** . • Select the style(s)*, then do one of the following: • Drag the selected style(s) onto the current drawing. • Right click and choose **Add Dimstyle(s)** in the context menu. * Press and hold the Ctrl-key to select multiple dimension styles.

Place View	Executes the PlaceView (6.24.24) command. Places a model view selected from a source drawing into a paper space layout of the current drawing. Do the following: • Switch to the layout where you want to place the model view in the target drawing. • Expand the source drawing in the **Local Folders** tree. • Click **Model Views** . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • Drag a model view from the **Content Browser** panel onto the paper space layout of the target drawing.* Prompts you: **Enter file name:(D:\BricsCAD Training\Bricscad\TrainingCD\EN\Exercises\House.dwg) Enter view name to place or [?]:(Garage (Option) right click to select scale.) Enter view origin point:** • (option) Right click and choose a scale in the Scale List • Click a point to place the view. The source drawing is inserted as an Xref drawing in the model space of the current drawing, on a layer with the same name as the source drawing A paper space viewport is added looking onto the xref according to a newly added view that matches the source view. In this new viewport, only the layers of the xref are set visible, so the view is not disturbed by other drawing content. If the current drawing belongs to a sheetset, a matching sheetset view is created and a view label block is added. * If the current layout does not belong to an open sheet set an alert box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/

Procedures

To open or close a panel (1.12.7) To dock a panel (1.12.7) To dock a panel with respect to another docked panel (1.12.7) To stack panels in a tabbed panel (1.12.7)

Related Commands

ContentBrowserclose (6.9.40) - closes the Content Browser panel. -Insert (6.17.14) - inserts blocks, by prompting in the command bar and prompts for attribute values through the command bar or a dialog box. MView (6.21.37) - creates one or more viewports in layout tabs. PlaceView (6.24.24) - places a model view selected from a source drawing into a paper space layout of the current drawing.

Figure 6.557

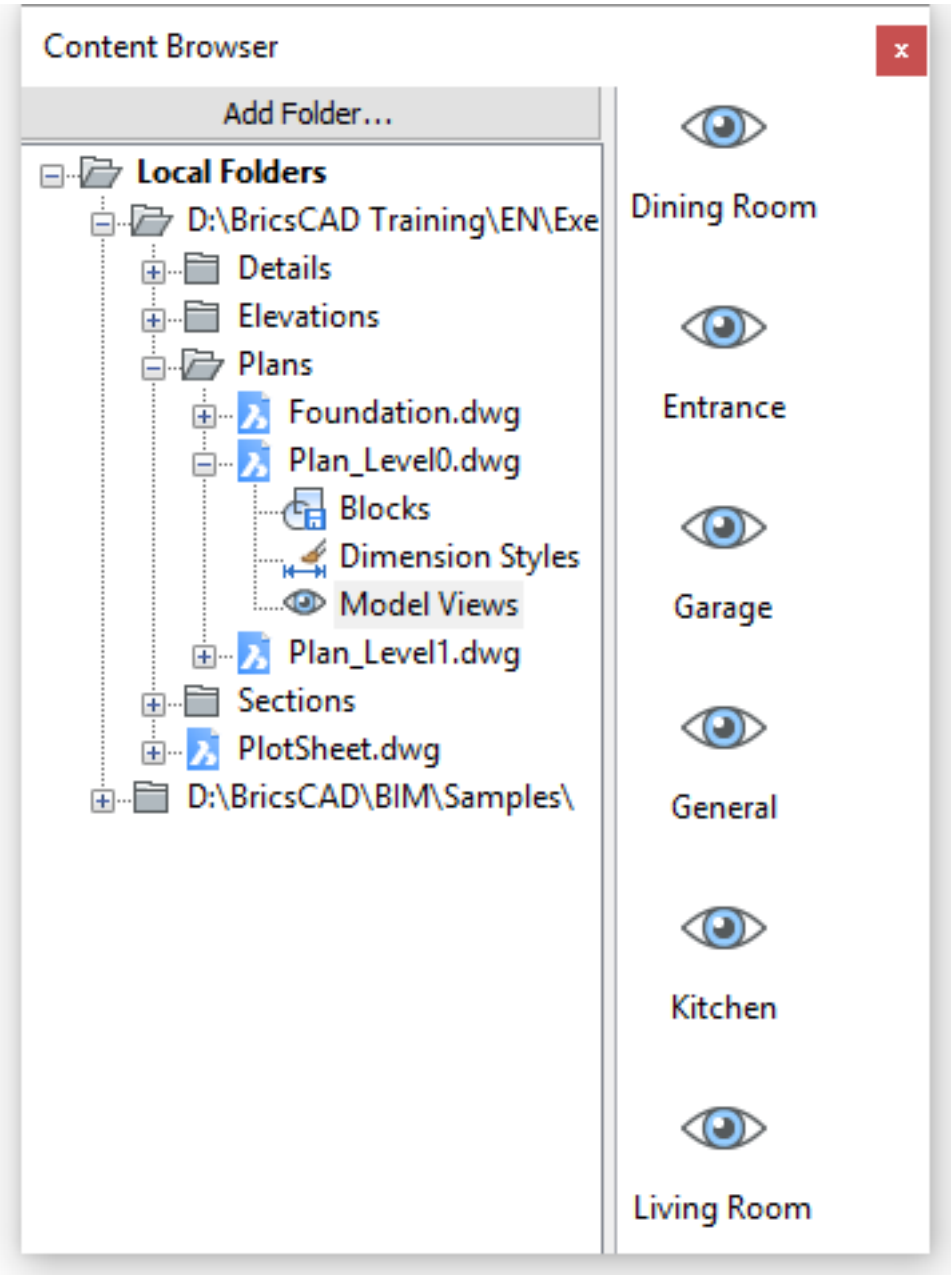


Figure 6.558

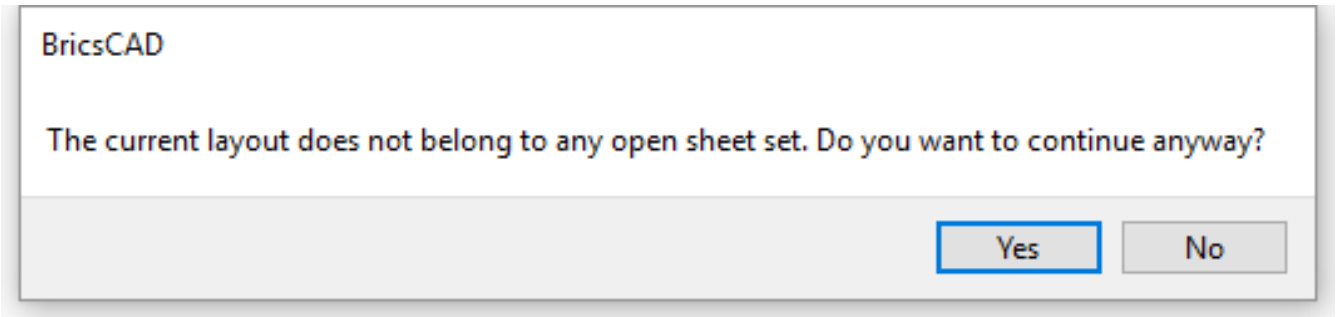


Figure 6.559



6.9.42 ConvertCTB

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Converts CTB (color-based plot tables) files to STB (style-based plot tables) files.

Accessing the Command

command bar: **convertctb** : **convertctb** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU 1-dlg.png}) Choose a CTB file, and then click **Open**. Displays the next dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbric 2-dlg.png}) Enter a name for the STB file, and then click **Save**. Displays the next dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/convertctb-3-dlg.png}) Read the warnings, and then click **OK**.

Figure 6.560

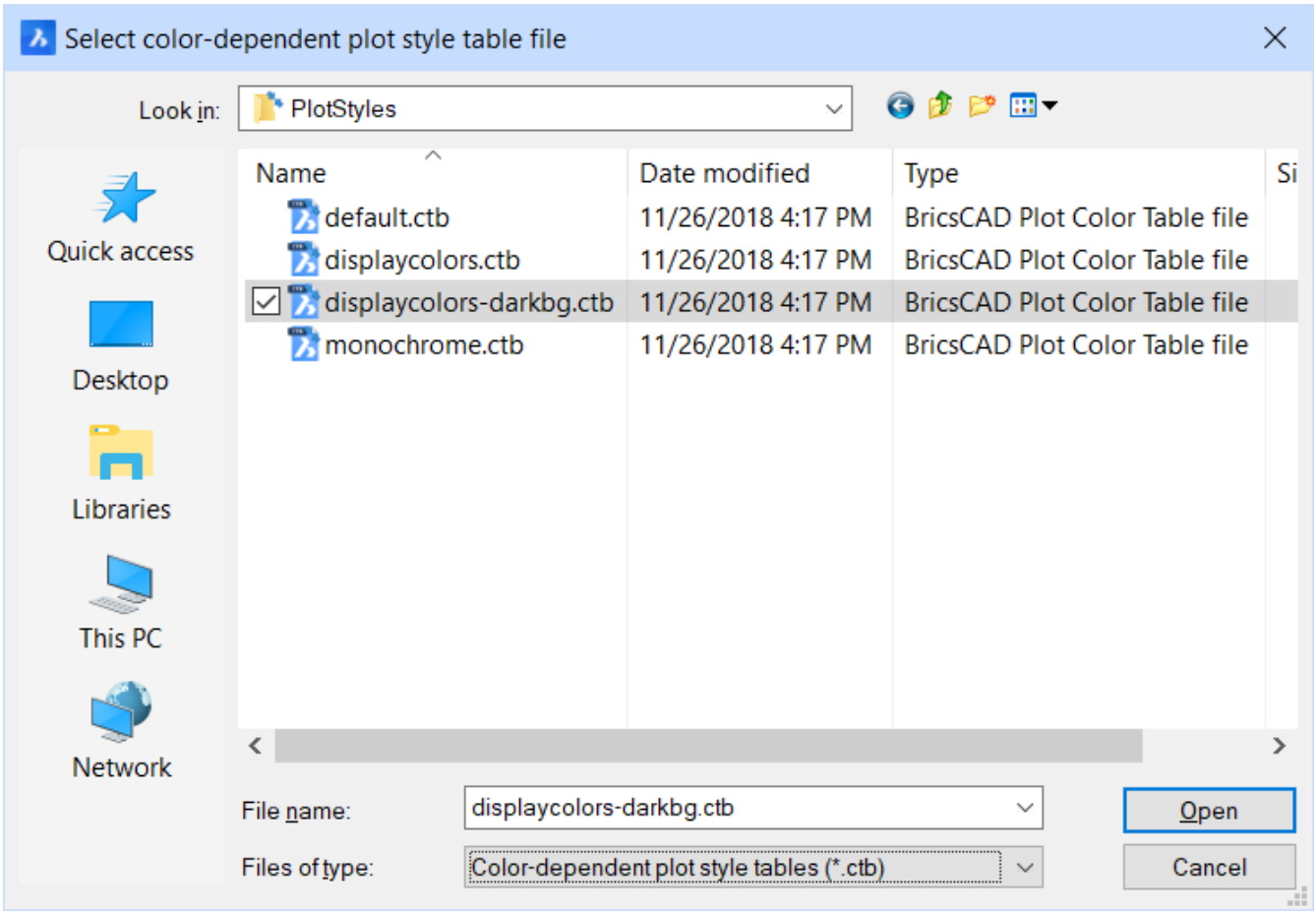


Figure 6.561

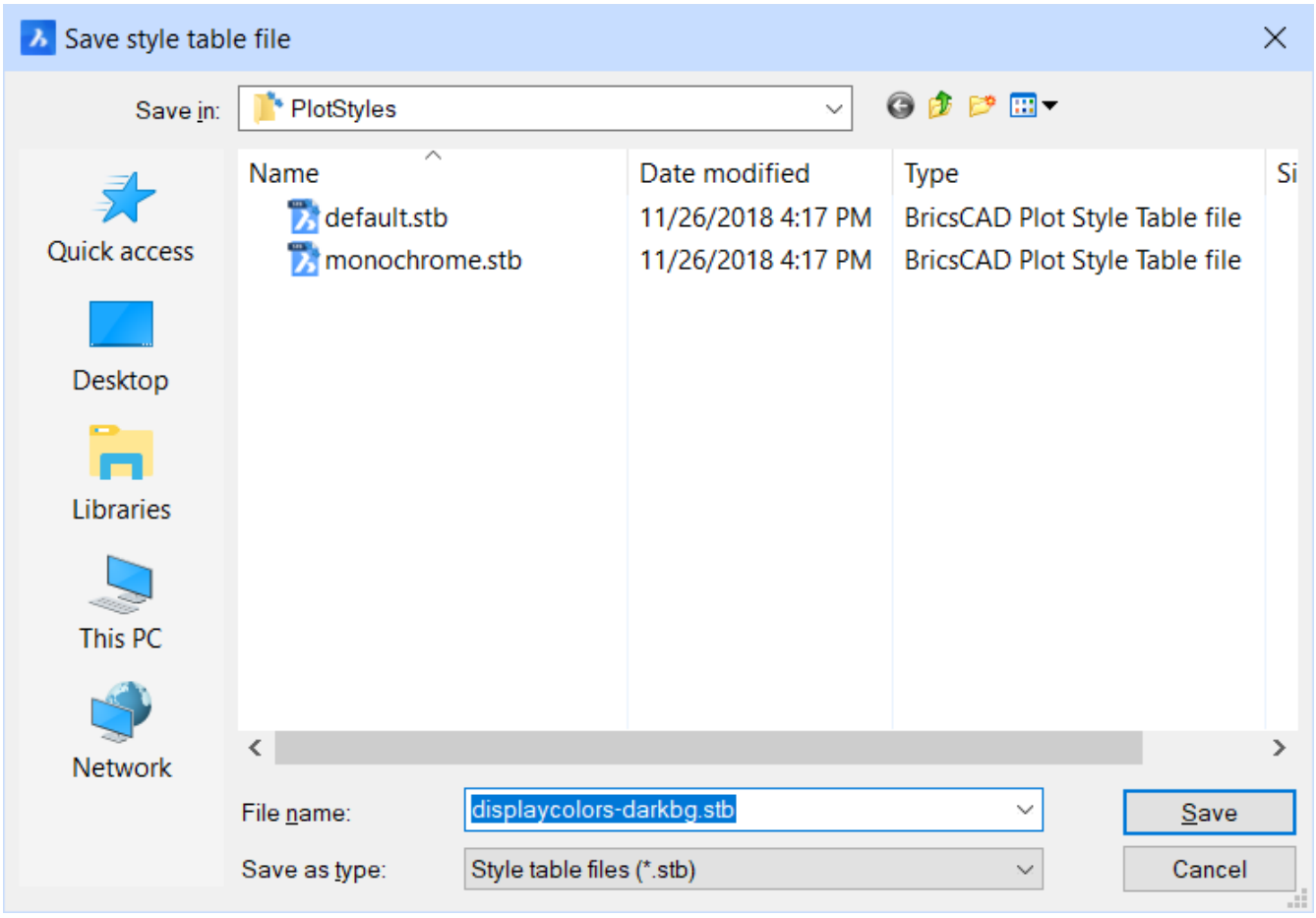
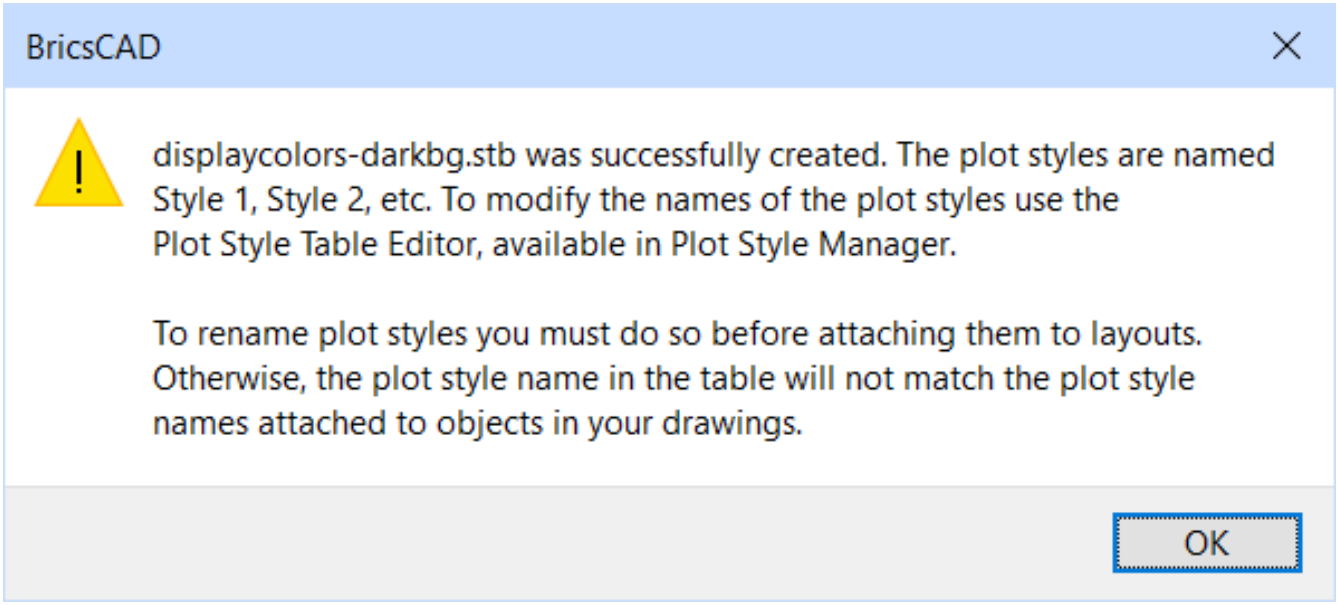


Figure 6.562



Procedures

- To edit plot style tables (1.25.3)
- To manage plot styles (1.25.5)

Related Commands

ConvertPStyles (6.9.46) - converts drawings from CTB to STB plot styles. Layer (6.20.2) - assigns plot styles to layers. PageSetup (6.24.1) - assigns plot style to drawings. PlotStyle (6.24.29) - sets the current plot style. Print (6.25.3) - plots drawings with plot styles. Properties (6.24.49) - changes plot styles assigned to entities. StylesManager (6.27.55) - creates and attaches plot styles to drawings.

Figure 6.563



6.9.43 ConvertOldLights

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Converts the definitionsold lights to that of the current format.

Accessing the Command

command bar: **convertoldlights : convertoldlights** The drawing is searched for old light definitions, which are then converted to the current format automatically.It may be necessary to check their properties (in case some may have changed) by editing the converted lights with the **LightList (6.20.29)** command.

Related Commands

Light (6.20.28) - creates a new light in the drawing. LightList (6.20.29) - displays the Drawing Explorer - Lights dialog window.

Figure 6.564



6.9.44 ConvertOldMaterials

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Converts old definitions of material to the that of the current format.

Accessing the Command

command bar: **convertoldmaterials : convertoldmaterials** The drawing is searched for old material definitions, which are then converted to the current format automatically. It might be necessary to edit the converted materials to check their settings with the **Materials (6.21.11)** command.

Related commands

Materials (6.21.11) - creates materials and edits their properties through the Drawing Explorer.

Figure 6.565



6.9.45 ConvertPoly

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Converts 2D and 3D polylines between modern lightweight and classic polylines (heavyweight) definitions.

Accessing the Command

command bar: **convertpoly** transparent: **'convertpoly** menu bar:**Modify | Convert Polyline : convertpoly You are prompted:** Enter polyline conversion option [Heavy/Light/3dPoly] <Light>**: (Enter an option)** Select entities: **(Select one or more polylines in the drawing)** Entities in set: 2 Select entities: **(Select more polylines, or press else Enter to convert the selection)** 2 polyline objects converted.

Command Options

Option	Description
Light	Converts lightweight 2D polylines to classic, heavyweight polylines. This may be needed to make drawings compatible with certain software.
Heavy	Converts classic 2D polylines to lightweight ones.
3dPoly	Converts 3D polylines (The abbreviation for this option is 3P.)
Select entities	Select one or more polylines. The program ignores ineligible entities.

The primary benefit to using this command is to reduce the drawing file size by using lightweight polyline definitions. This command does not convert the following kinds of polylines:

- Curve-fit
- Splined polylines
- Polylines with extended object data stored on their vertices .

You may not need to use this command in the following situations:

- Editing commands apply to both classic and lightweight polylines.
- The PlineType system variable specifies whether classic polylines are converted to lightweight polylines automatically when an older drawing is opened. You can use this system variable to determine which style of polyline is created in new drawings.

Related Commands

PLine (6.24.27) - draws polyline lines, arcs, and splines. PEdit (6.24.20) - edits polylines, 3D polylines, and 3D meshes.

Figure 6.566



6.9.46 ConvertPStyles

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Converts the plotting method in drawings to from CTB (using color tables) to STB (using named plot styles).

Accessing the Command

command bar: **convertpstyles** : **convertpstyles** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/
1-dlg.png}) Click **OK**.The next dialog box appears: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Con
2-dlg.png}) Choose an STB file, and then click **Open**.**BricsCAD reports in the command bar:** Drawing converted from Color Dependent mode to Named plot style mode. **(To convert CTB files to STB format, use the ConvertCtb (6.9.42) command.)**

Figure 6.567

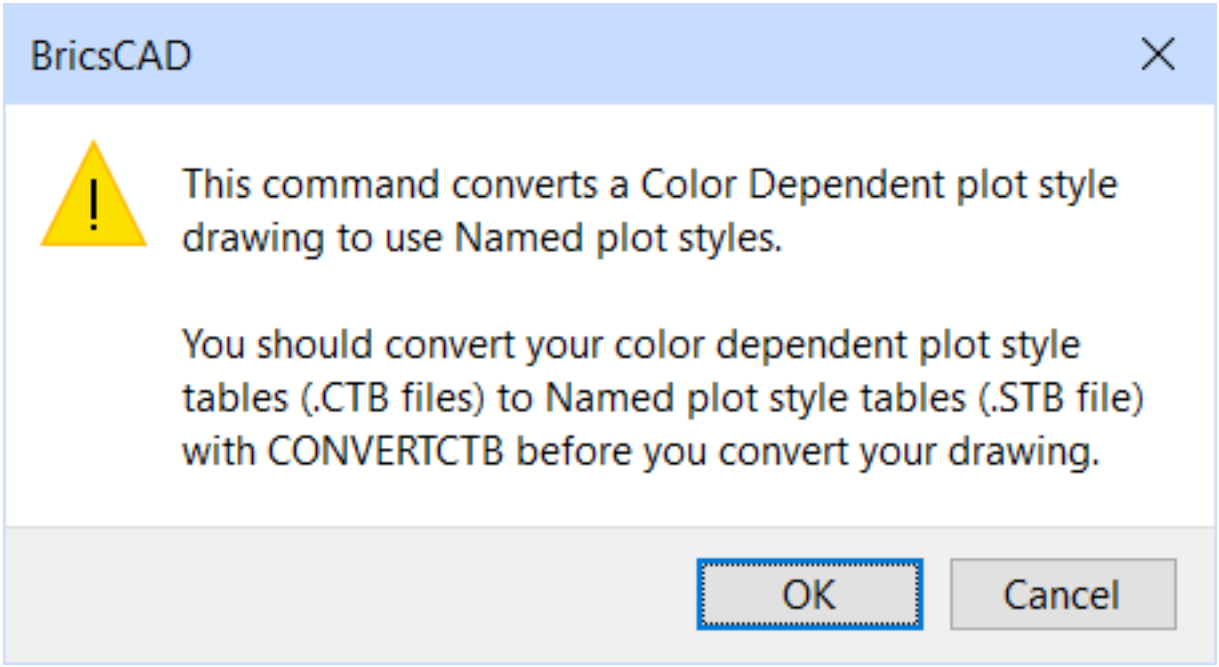
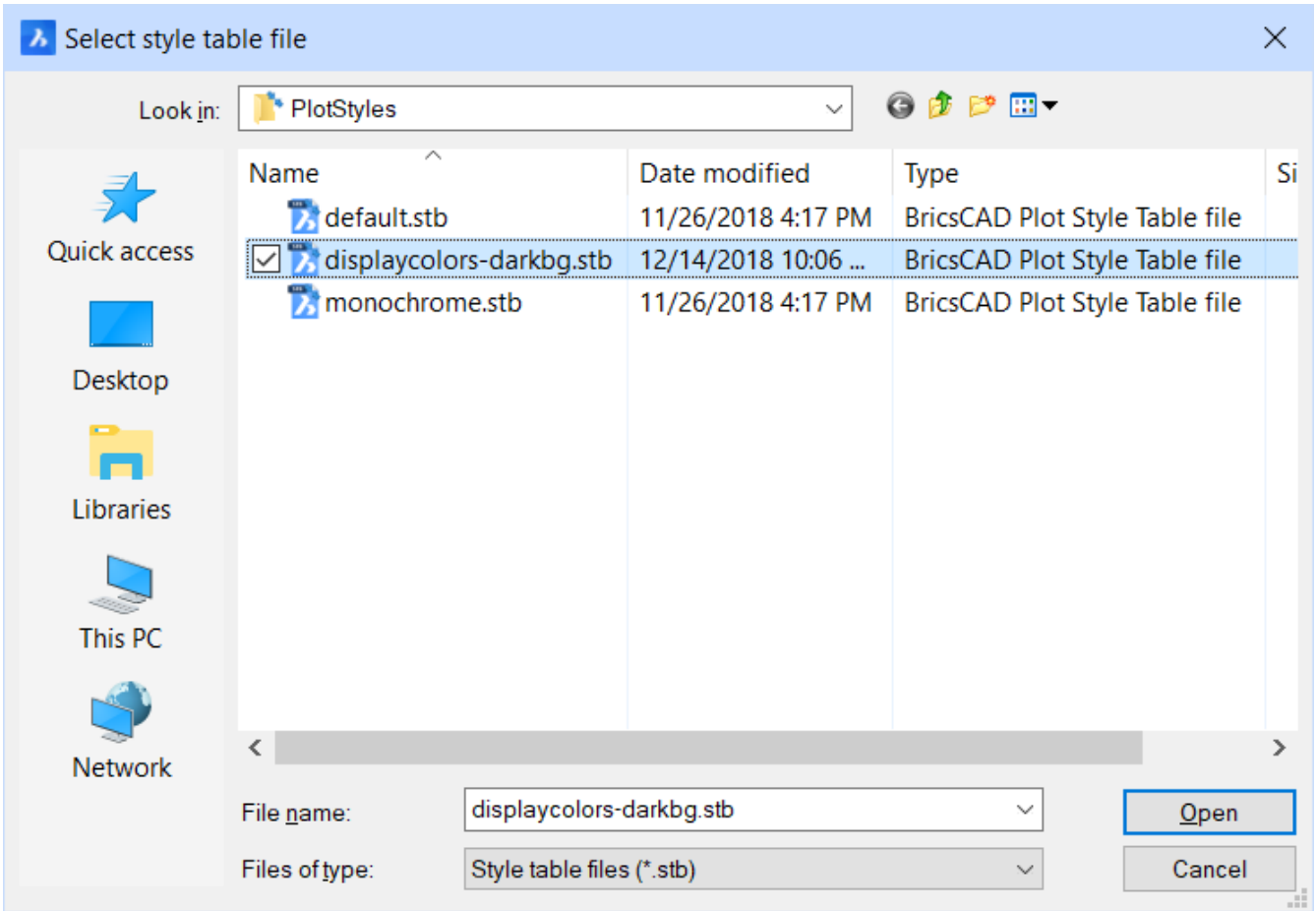


Figure 6.568



Procedures

- To edit plot style tables (1.25.3)
- To manage plot styles (1.25.5)

Related Commands

ConvertCtb (6.9.42) - converts CTB plot style files to STB files. Layer (6.20.2) - assigns plot styles to layers. PageSetup (6.24.1) - assigns plot style to drawings. PlotStyle (6.24.29) - sets the current plot style. Print (6.25.3) - plots drawings with plot styles. Properties (6.24.49) - changes plot styles assigned to entities. StylesManager (6.27.55) - creates and attaches plot styles to drawings.

Figure 6.569



6.9.47 ConvToMesh

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Converts the following 2D and 3D entities to 3D meshes:3D solids, 3D surfaces, 3D faces, polyface and polygon meshes (legacy), regions, and closed polylines.

Accessing the Command

Keyboard: **convtomesh :ConvToMesh You are prompted:** Select entities: **(Select one or more valid entities; see list below)** Entities in set: 1 Select entities: **(Press Enter to finish selecting entities and execute command)** BricsCAD converts the entities into meshes.

Command Options

Option	Description
Select entities:	Select one or more of the following entities: • 3D solids • 3D surfaces • 3D faces • Polyface and polygon meshes (legacy) • Regions • Closed polylines

Related Commands

ConvToSolid (6.9.48) - converts entities to 3D solids. ConvToSurface (6.9.49) - converts entities to 3D surfaces

Figure 6.570



6.9.48 ConvToSolid

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Converts the following 2D and 3D entities to 3D solids:watertight 3D meshes and 3D surfaces; 3D polygon and polyface meshes; circles with thickness; and closed polylines of non-zero thickness and uniform width.

Accessing the Command

Keyboard: convtosolid :ConvToSolid **You are prompted:** Select entities: **(Select one or more valid entities, as listed below)** Entities in set: 1 Select entities: **(Press Enter to finish selecting entities and execute command)** BricsCAD converts the entities into 3D solids.

Command Options

Option	Description
Select entities:	Select one or more of the following entities: • Watertight 3D meshes and 3D surfaces • 3D polygon and polyface meshes • Circles with thickness • Closed polylines of non-zero thickness and uniform width ("Watertight" means a mesh or surface that is completely enclosed, with no gaps or other openings.)

Related Commands

ConvToMesh (6.9.47) - converts entities to 3D meshes. ConvToSurface (6.9.49) - converts entities to 3D surfaces.

Figure 6.571



6.9.49 ConvToSurface

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Converts the following 2D and 3D entities to 3D surfaces: 2D solids and 3D solids, regions, zero-width open polylines with thickness, lines and arcs with thickness, meshes and planar 3D faces.

Accessing the Command

Keyboard: convtosurface :ConvToSurface **You are prompted:** Select entities: **(Select one or more valid entities)** Entities in set: 1 Select entities: **(Press Enter to finish selecting entities and execute command)** BricsCAD converts the entities to surfaces.

Command Options

Option	Description
Select entities:	Select one or more of the following entities: • 2D and 3D solids • Regions • Zero-width open polylines with thickness • Arcs and lines and arcs with thickness • Meshes and planar 3D faces

Related Commands

ConvToMesh (6.9.47) - converts entities to 3D meshes. ConvToSolid (6.9.48) - converts entities to 3D solids.

Figure 6.572



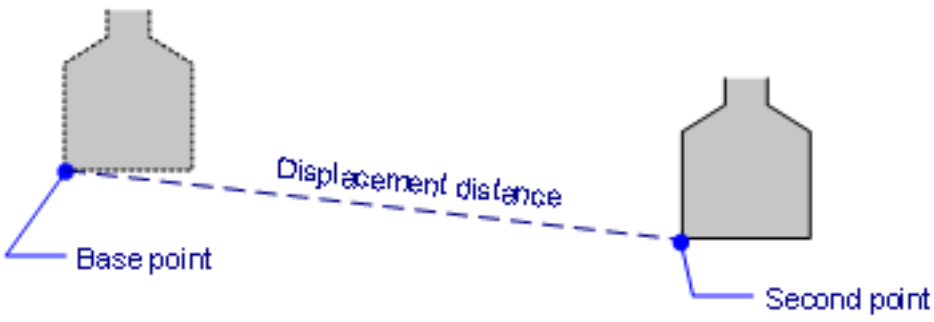
6.9.50 Copy

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Makes copies entities, including repeating ones.

Accessing the Command

command bar: **copy** aliases: **co, cp** menu bar: **Modify | Copy** toolbar: **Modify |** ribbon:**Change | Arrange | Copy** Quad cursor menu: **Custom | : copy You are prompted:** Select entities to copy: **(Select one or more entities)** Select entities to copy: **(Select more entities, or else press Enter to end entity selection)** Current copy mode: Multiple Enter base point [Displacement/mOde] <Displacement>: **(Pick a point, or enter an option.)** Enter second point <Use base point as displacement>: **(Pick another point, or press Enter)** Enter second point [Undo/Repeat/Exit]: **(Pick another point, or enter an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS

Figure 6.573



Command Options

Option	Description
Select entities to copy	Use any selection mode to choose the entities to be copied.
Enter base point	Specifies the point from which to copy.Use entity snaps to pick the starting point accurately.
mOde	Toggles between single and multiple copying modes; prompts you: **Enter a copy mode [Single/Multiple] <Multiple>** - **(enter S or M)**: • **Single** - makes one copy, and then exits the command • **Multiple** - repeats the 'Enter second point' prompt until you enter the Exit option
Displacement	Specifies the displacement (the distance at which to position the copy); prompts you: **Enter displacement point <0.0000,0.0000,0.0000>** - **(enter x,y,z coordinates, or pick a point)**
Enter second point	Specifies the destination of the copied entities by entering coordinates or picking a point When you use x,y,z coordinates, you copy objects in 3D space.Hold down the **Shift** key to copy entities in a horizontal or vertical direction
Use Base Point As Displacement	Determines the displacement distance from the coordinates of the base point.
Multiple	In single-copy mode only: Switches to multiple-copy mode.
Undo	In multiple-copy mode only: Undoes the last copy operation.
Repeat	Repeats the copy using the same displacement; prompts you: **Select end point (<ENTER> to key in number of repeats)** - **(specify a point or press Enter.)** • **Specify a point** - move the cursor in the displacement direction. The further you move away from the source entity, the more copies are created. Click when the desired number of copies displays. • **Enter number of repeats <1>** - press Enter to accept the previous number of repeats or type a number, then press Enter.
Exit	In multiple-copy mode only: Exits this command.

Procedures

- Using Copy and Paste to copy entities between drawings (1.23.17)
- Using Copy and Pasteorig to copy entities between drawings (1.23.17)
- Using Copybase and Paste to copy entities between drawings (1.23.17)
- Paste a selection as a block in the target drawing (1.23.17)
- To copy entities using grips (1.23.4)

Related Commands

CopyClip (6.9.52) - copies selected entities to the Clipboard, which can then be pasted in the same or other drawings. CopyBase (6.9.51)- copies selected entities to the Clipboard with a base point, for pasting at an insertion point. Array (6.4.16) - creates arrayed copies of entities in 2D linear, rectangular, and polar patterns. 3dArray (6.3.2) - creates arrayed copies of entities in 3D rectangular and polar patterns. MInsert (6.21.17) - inserts arrayed copies of blocks. Offset (6.23.3) - creates parallel copies of entities.

Figure 6.574



6.9.51 CopyBase

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Copies entities to the Clipboard with a base point for pasting with the PasteBlock command.

Accessing the Command

 command bar: **copybase** keyboard shortcut: **Ctrl+Shift+C** menu bar: **Edit | Copy with Base Point** ribbon:**Home | Edit | Copy | Copy with Base Point** : copybase **You are prompted:** Select base point: **(Pick a point, or enter x,y coordinates)** Select entities to copy to the clipboard: **(Choose one or more entities)** Select entities to copy to the clipboard: **(Choose more entities, or else press Enter to continue)** Use the PasteBlock (6.24.9) command to paste the copied entities into the same or another drawing as a block.

Command Options

Option	Description
Select base point	Specifies the base point for the entities being copied to the Clipboard.Pick a point, or enter x,y coordinates. The PasteBlock command uses these coordinates to paste the entities from the Clipboard into a drawing.
Select entities to copy to the clipboard	Chooses the entities to copy to the Clipboard usingany entity selection method.

Procedures

- Using Copybase and Paste to copy entities between drawings (1.23.17)
- Paste a selection as a block in the target drawing (1.23.17)

Related Commands

CopyClip (6.9.52) - copies selected entities to the Clipboard, which can then be pasted in the same or other drawings. CopyHist (6.9.54) - copies all text from the Text window to the Clipboard. CutClip (6.9.59) - cuts (moves) selected entities to the Clipboard. PasteBlock (6.24.9) - pasted entities from the Clipboard as a block at an insertion point. PasteClip (6.24.10) - pastes entities from the Clipboard into the current drawing. PasteOrig (6.24.41) - pastes entities from the Clipboard at the original coordinates. PasteSpec (6.24.11) - displays the Paste Special dialog box for controlling the format of pasted entities.

Figure 6.575



6.9.52 CopyClip

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Copies entities to the Clipboard for pasting into drawings and other documents. 

Accessing the Command

command bar: **copyclip** keyboard shortcut: **Ctrl+C** menu bar: **Edit | Copy** toolbar: **Standard |** ribbon: **Home | Edit | Copy : copyclip You are prompted:** Select entities to copy to the clipboard:**(Choose one or more entities, or press Ctrl+A to sleetall of them)** Select entities to copy to the clipboard:**(Choose more entities, or press else Enter to exit entity selection)**

Command Options

Option	Description
Select entities to copy to the clipboard	Choose the entities to be copied; use any entity selection method. Once in the Clipboard, the entities can be pasted **Ctrl+V** into the same drawing, into other drawings, and into other documents.

The **PictureExportScale** user preference sets the resolution of the image when the exported geometry is pasted in bitmap format, such as in a Word document.

Procedures

- Using Copy and Paste to copy entities between drawings (1.23.17)
- Using Copy and Pasteorig to copy entities between drawings (1.23.17)

Related Commands

ChSpace (6.9.19) - moves entities from paper space to model space and vice versa. CopyBase (6.9.51)- copies selected entities to the Clipboard with a base point. CopyHist (6.9.54) - copies all text from the Text window to the Clipboard. CutClip (6.9.59) - cuts (moves) selected entities to the Clipboard. PasteBlock (6.24.9) - pasted entities from the Clipboard as a block at an insertion point. PasteClip (6.24.10) - pastes entities from the Clipboard into the current drawing. PasteOrig (6.24.41) - pastes entities from the Clipboard at the original coordinates. PasteSpec (6.24.11) - displays the Paste Special dialog box for controlling the format of pasted entities.

Figure 6.576



6.9.53 CopyEData

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Copies extended entity data from one entity to others.

Accessing the Command



command bar: **copyedata** transparent: **'Copyedata** menu bar: **Tools | Entity Data | Copy Entity Data** toolbar: **Entity Data |** **: copyedata You are prompted:** Copy Entity Data: Name of application or [?]: **(Enter the name of the application to which the entity data belongs, or else type ? to list the names of application)** Select entity from which to copy Entity Data: **(Choose a single entity.)** Select entity to which to copy Entity Data: **(Choose another entity)** Select entity to which to copy Entity Data: **(Choose another entity, or else press Enter to exit the command)**

Command Options

Option	Description
Name of Application	Specifies the name of the application that generated the extended entity data.
?	Lists the names of registered applications.
Select Entity From Which to Copy Entity Data	Selects the entity from which to copy edata. The entity must contain edata, which can be created with the EditE-Data command.
Select Entity To Which to Copy Entity Data	Specifies the entity to receive the edata.

Related Commands

DelEData (6.10.25) - erases extended entity data from entities. EditEData (6.13.3) - creates and edits extended entity data. MoveEData (6.21.32) - moves extended entity data to other entities. ReAssocApp (6.26.2) - associates applications with entities.

Figure 6.577



6.9.54 CopyHist

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Copies the last 254 lines of command history text to the Clipboard.

Accessing the Command

command bar: **copyhist** : **copyhist** Displays no prompt in the command bar.All text in the command bar history is copied to the Clipboard; you can paste the text into another application with Ctrl+V . The **ScrlHist** variable determines how many lines of command history the Pormpt History window retains; its initial value is 256 lines.

Related Commands

CopyClip (6.9.52) - copies selected entities to the Clipboard. CutClip (6.9.59) - cuts (moves) selected entities to the Clipboard. PasteClip (6.24.10) - pastes entities from the Clipboard into the current drawing. PasteOrig (6.24.41) - pastes entities from the Clipboard at the original coordinates. PasteSpec (6.24.11) - displays the Paste Special dialog box for controlling the format of pasted entities.

Figure 6.578



6.9.55 CPageSetup

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits the page setup of the current layout or model space in a dialog box.

Accessing the Command

command bar: **cpagesetup** context menu: right-click the model tab or a layout tab (\vref {fig:C:/Dev/Task-2448/images/staticb
: **cpagesetup** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/cp
dlg.png})

Figure 6.579

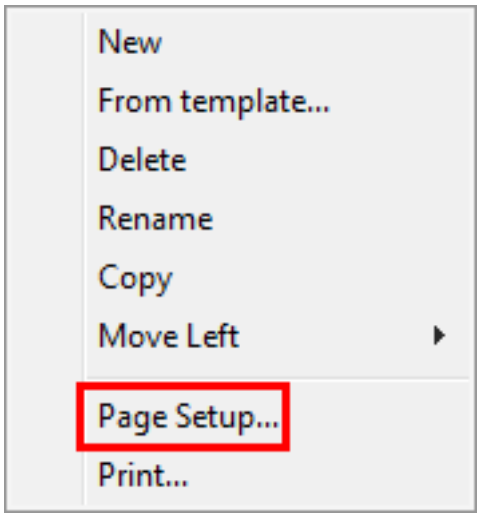
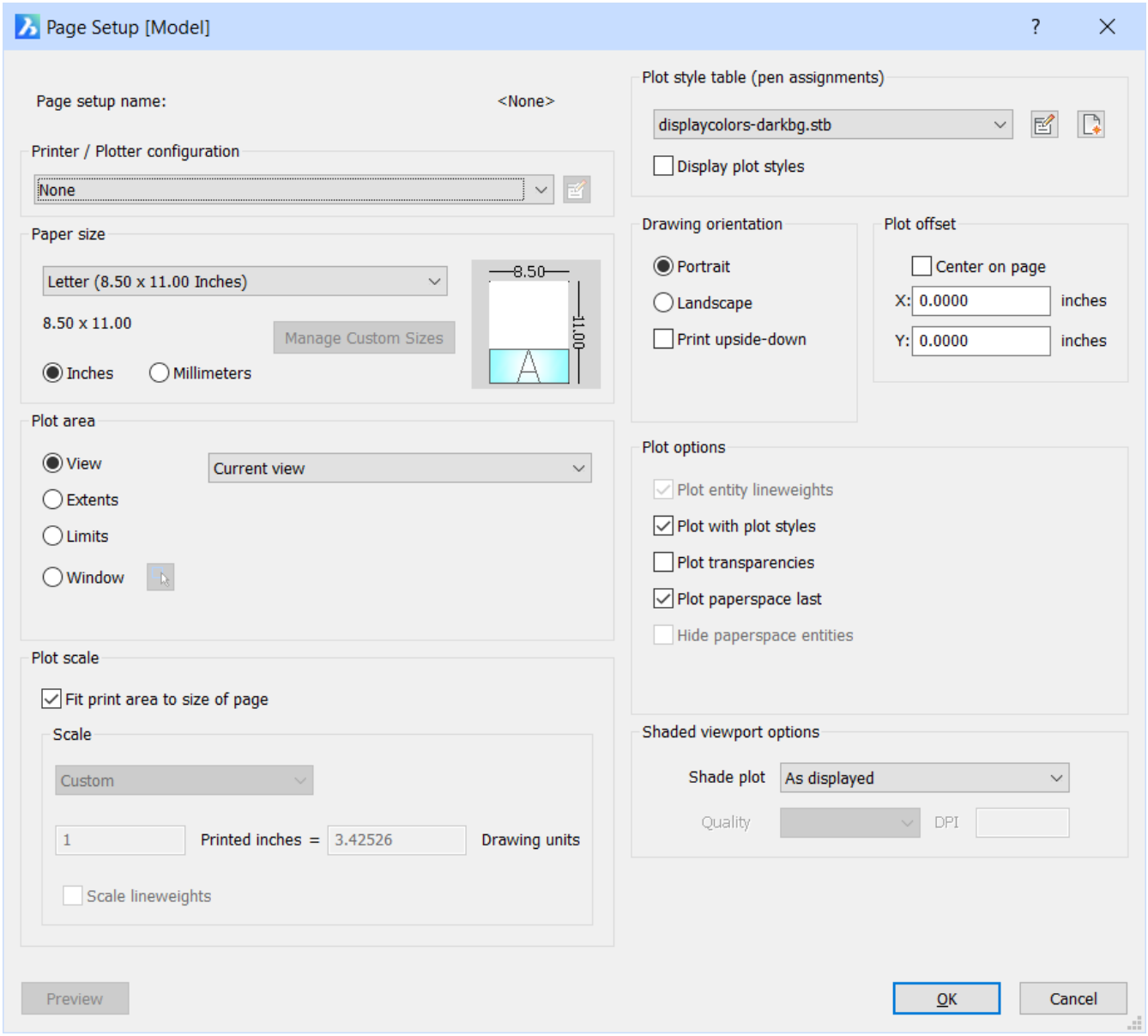


Figure 6.580


Page Setup [Model]
Page setup name:
<None>
Printer / Plotter configuration
None
Paper size
Letter (8.50 x 11.00 Inches)
8.50 x 11.00
Manage Custom Sizes
Inches
Millimeters
8.50
11.00
Plot area
View
Current view
Extents
Limits
Window
Plot scale
Fit print area to size of page
Scale
Custom
1
Printed inches =
3.42526
Drawing units
Scale lineweights
Plot style table (pen assignments)
displaycolors-darkbg.stb
Display plot styles
Drawing orientation
Portrait
Landscape
Print upside-down
Plot offset
Center on page
X:
0.0000
inches
Y:
0.0000
inches
Plot options
Plot entity lineweights
Plot with plot styles
Plot transparencies
Plot paperspace last
Hide paperspace entities
Shaded viewport options
Shade plot
As displayed
Quality
DPI
Preview
OK
Cancel

Command Options

See the PageSetup (6.24.1) command.

Related Commands

PageSetup (6.24.1) - creates and edits page setups for plotting drawings in the Drawing Explorer. Print (6.25.3) - prints drawings to plotters and files through a dialog box.

Figure 6.581



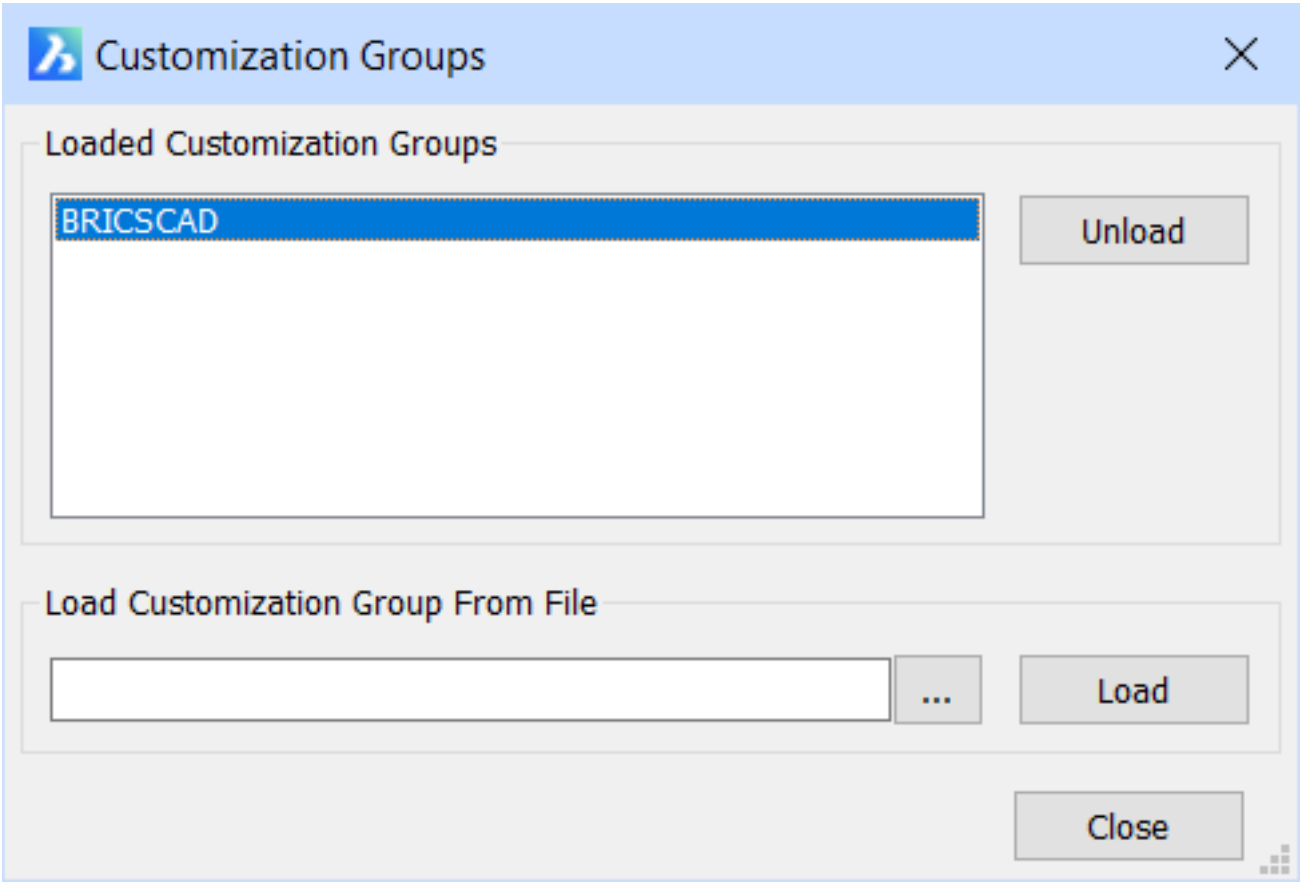
6.9.56 CuiLoad

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads CUIX and CUI (user interface customization), MNU (menu), MNS (LISP code), and ICM (IntelliCAD menu) files into the program.

Accessing the Command

command bar: **cuiload** transparent:'**cuiload** aliases: **menuload (6.21.14), cuiunload** : **cuiload** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/cuiload-dlg.png})

Figure 6.582



Command Options

Option	Description
Loaded Customization Groups	Lists the names of customization files loaded in BricsCAD.
Unload	Unloads the selected customization group.**Caution**: unloading 'BricsCAD' causes most user interface elements to disappear.
...	Displays the Choose a Customization File dialog box.Choose a CUIX, CUI, MNU, MNS, or ICM file, and then click **Open**.
Load	Loads the opened customization file. **Caution**: loading a customization file changes the program's user interface.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Commands (6.9.37) - lists the names of internal and external commands. CuiUnload (6.9.57) - unloads menu files. Customize (6.9.58) - customizes the user interface. MenuLoad (6.21.14) and MenuUnload (6.21.15) - loads and unloads menu files. Netload (6.22.3) - loads a .NET application. VbaLoad (6.32.2) and -VbaLoad (6.32.3) - loads VBA projects. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaMan (6.32.4) - manages VBA projects. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.583



6.9.57 CuiUnload

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads and unloads customization groups from the user interface; operates identically to the **CuiLoad (6.9.56)** command.

Accessing the Command

command bar:**cuiUnload** transparent:**'cuiUnload** aliases:**menuload (6.21.14), cuiunload** :**cuiUnload** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/cuiload-dlg.png})

Figure 6.584

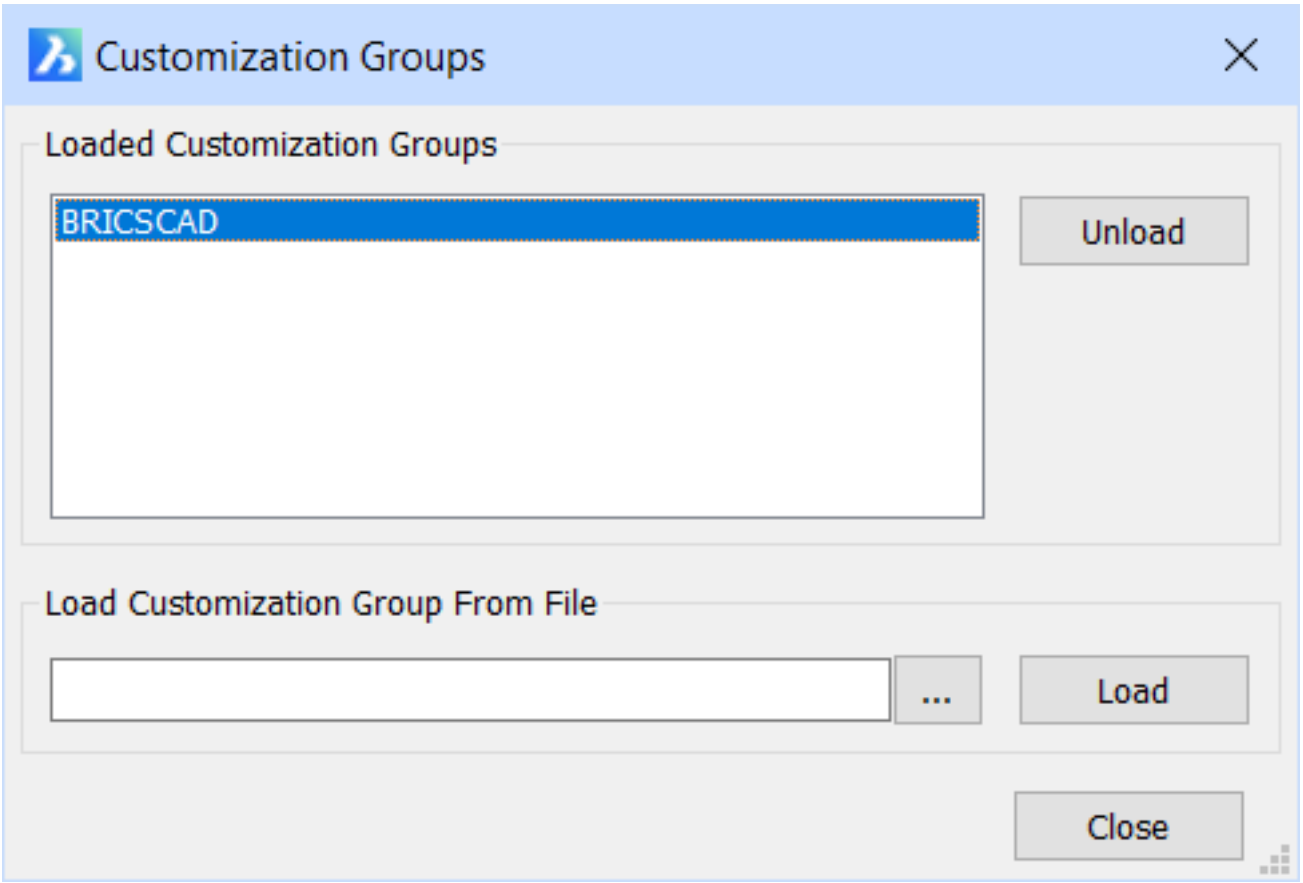

Customization Groups
Loaded Customization Groups
BRICSCAD
Unload
Load Customization Group From File
...
Load
Close

Figure 6.585



6.9.58 Customize

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Customizes the user interface of BricsCAD: menus, toolbars, keyboard shortcuts, mouse actions, tablet buttons and menus, Quad settings, properties, workspaces, aliases, and shell commands. For details on using this command, see also ”Customizing BricsCAD” in Help Center.

Accessing the Command

command bar: **customize** alias: **cui** menu bar: **Tools | Customize** context menu: right-click any toolbar or ribbon, and then choose **Customize** : **customize** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/er
Create or modify a user interface elements, and then click **OK**.For details on using this dialog box, see the Customizing BricsCADebook.

Command Options

Options	Description
File	Displays a menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu • **Load main CUI file...**: See the CuiLoad (6.9.56) command. • **Save main CUI file as...**: Saves the current main CUI file under a different name, thus copying the main CUI file. • **Load Partial CUI file...**: See To load a partial CUI file (1.13.4). • **Create new partial CUI file...**: Creates an empty CUI file. • **Import workspace(s)...**: Loads one or more workspaces from another CUI file. – Displays a **File** dialog to let you select a CUI file. Select CUI, then click the **Open** button. – Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comm Select the workspaces to be imported (press an hold the Ctrl key to select multiple workspaces), then click the **OK** button.
Main Customization File	Specifies the CUI file that defines customization of the menu, toolbar, and other elements of the user interface. Press the **Browse** (...) button to load a different main CUI file. Aliases and shell commands are defined in a PGP file.
Search	Searches the Available Tools list for command names.

Manage Your Customizations	Displays the Manage Customizations dialog box for confirming and reversing changes made to the user interface. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu Lists all customizeations: • **Added**: Items you have created display in green. • **Modified**: Items you have modified display in blue. • **Removed**: Items you have removed display in red Do one of the following: • Uncheck the checkbox of a customized item(s) to remove the customization(s). • Click the **Retain customizations** button to keep all customized items • Click the **Revert to defaults** button to remove all customizations. Click the **OK** button to confirm.

Menus Tab Options	**Description**
Menus Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu To add a new menu item: 1. Select a command in the right hand pane 2. Drag the command to the menu in the left hand pane To move a menu item: • Drag the item to its new location in the menu structure
Shortcut Menus	Right click a menu (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu Right-click a submenu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu Right-click a menu item: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu Choose an option: • **Insert main menu** - inserts a main menu. • **Append ...**- adds a submenu, menu item or separator below the last item. • **Insert ...**- adds an submenu, item or separator above the selected item. • **Delete ...**- deletes the menu, submenu, item or separator.
Menu Item	
Title	Specifies the name displayed by the menu item. Prefix a letter with & to create an Alt-key shortcut, such as &Line.
Diesel	Specifies the Diesel code to be employed by the menu item.
Tool ID	Identifies the menu item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD.)
Help	Specifies the help string displayed on the status bar.
Command	Specifies the command(s) or macros.

Image	Specifies the image to display next to the menu item; click ... to access the file dialog box.

Toolbars Tab Options	**Description**
Toolbars Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu • To add a new toolbar item: Select a command in the right and pane, then drag the command to the toolbar in the left hand pane. • To move a toolbar item: Drag the item to its new location.
Shortcut Menu	Right-click a toolbar item: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C • **Insert Tool**: Inserts a new button in the selected toolbar. • **Insert Control**: Inserts a control (droplist). • **Insert Flyout**: Inserts a flyout (sub-toolbar). • **Insert Separator**: Inserts a separator line. • **Delete**: Erases the selected toolbar button.
Toolbar Button	
Title	Names the toolbar button; displayed by the tooltip.
Diesel	Specifies the Diesel code to be employed with the button.
Tool ID	Identifies the toolbar item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD.)
Help	Specifies the help string displayed on the status bar.
Command	Specifies the command(s) or macros.
Image	Specifies the image to use for the toolbar button; click ... to access the file dialog box.

Ribbon Tab Options	**Description**
Ribbon Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu
Ribbon tabs	To add a new ribbon tab: • Right click the **Ribbon tabs** node and click **Append ribbon tab** in the context menu to add a new tab at the bottom of the list. • Right click an existing tab and choose **Insert ribbon tab** in the context menu to add a new tab above the selected tab. Drag a tab to modify the order. To add a panel to a tab: • Right click the tab name and choose **Append panel** in the context menu to add a panel at the bottom of the list. • Right click an existing panel and choose **Insert panel** in the context menu to ad a panel above the selected panel. Drag a panel to modify the order.
Ribbon panels	See the Ribbon (1.13.6) item in the BricsCAD User Guide.

Keyboard Tab Options	Description
Keyboard Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu To assign a different command to a keyboard shortcut, select a command in the right and pane, then drag the command to the keyboard shortcut in the left hand pane.
Shortcut Menu	Right-click a keyboard shortcut: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C
Insert Shortcut	Adds a new keyboard shortcut to the list; displays dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C • **Select Available Tool**: Assigns an existing command to the keyboard shortcut. • **Create New Tool**: Assigns a new command/macro to the shortcut. • **Toolbox**: Specifies the toolbox to which to add the new command; click the droplist, and then choose a toolbox. • **Title**: Names the new tool. • **Help**: Specifies the help text. • **Command**: Specifies the command(s) or macros. • **Image**: Specifies the name of the image file; click ... to access the file dialog box. • **Delete Shortcut**: Erases the selected shortcut.
Keyboard Shortcut	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu • **Key**:Specifies the keyboard shortcut. • **Tool ID**:Identifies the keyboard shortcut to other elements in the CUI file. (The Tool ID is assigned by BricsCAD.) • **Help**:Specifies the text of the help string, even though help strings are not displayed by keyboard shortcuts. • **Command**:Specifies the command(s) to associate with the keyboard shortcut. • **Image**:Specifies the image to associate with the command, even though images are not used with keyboard shortcuts.

Mouse Tab Options	Description
Mouse Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu

Tablet Tab Options	Description
Tablet Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu

Quad Tab Options	Description

Quad Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu • To add a command to a Quad tab: Select a command in the right and pane, then drag the command to the Quad tab in the left hand pane. • To move a command: Drag the command to its new location.
Shortcut Menus	Right click the CUI name (BRICSCAD) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu • **Append quad tab**: Adds a new Quad tab at the bottom of the tab list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/ Specify the **Label** and **Title** for the new Quad tab and click the OK button. Right-click a Quad tab: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu • **Insert quad tab**: Inserts a new Quad tab above the selected Quad tab (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/ Specify the **Label** and **Title** for the new Quad tab and click the **OK** button. • **Insert copy**: Inserts a copy of the selected Quad tab. Specify the **Label** and **Title** for the new Quad tab and click the **OK** button. • **Delete quad tab**: deletes the selected Quad tab. • **Append quad button**:Adds a new Quad button at the bottom of the list. Specify the **Label** and **Title** for the new Quad tab and click the **OK** button Right-click a Quad button (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu • **Insert**: Inserts a new Quad button above the selected button. • **Delete**: Deletes the selected Quad button.
Insert quad tab	Inserts a new quad tab: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V20/Commands/C/Ad • **Label**: The label name displays in the Quad. • **Title**: The title name identifies the Quad tab and therefore must be unique.
Delete quad tab	Select the Quad Tab you want to delete.
Append quad button	Adds a button in a tab.

Properties Tab Options	**Description**
Properties Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu To specify which properties show when the ROLLOVERTIPS system variable is ON, select an entity in the left hand pane, then check the desired properties in the right hand pane.

Workspace Tab Options	Description
Workspaces Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu
Menus	Controls the display of menus in the menu bar. To modify the order: drag the menu. • To control the display of a menu: click the **State** field and select **On** or **Off** .
Toolbars	Lists the toolbars which are currently open in the workspace.
Palettes	Controls which tool panels are open in the workspace. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C
Ribbon	Controls which ribbon tabs are available in the workspace. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C Drag a tab to modify the order. • Right click a tab in the list, then choose **Insert tab** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou
Quad Groups	Controls the display and the order of the Quad item groups. • To modify the order: drag the Quad item groups. • To control the display of a Quad item group: click the **State** field and select **On** or **Off** .
On Switch	Sets the commands to be executed when switching to the workspace. To ad a command, do one of the following: • Right click, then choose **Append Tool**in the context menu. • Drag a command from the right hand pane to the **On Switch** command list.
Context Menu	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C • **Insert workspace**: Creates a new workspace. • **Insert separator**: Inserts a separator before the selected workspace. • **Delete**: Deletes the selected workspace.

Command Aliases Tab Options	Description
Command Aliases Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu
Add	Adds an alias; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C
Alias	Specifies the name of the alias, a command abbreviation.
Command	Specifies the command to which the alias is assigned.

Edit	Edits the selected alias.
Delete	Erases the selected alias.

Shell Commands Tab Options	**Description**
Shell Commands Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Cu
Add	Adds a shell command; displays the following dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C Shell commands run programs external to BricsCAD.
Alias	Specifies the name of the shell command; this is the command you enter at the ' : ' prompt.
Shell Command	Specifies the operating system command to execute, or program to run. To run a program, use the Start prefix, as in Start Notepad.exe.
Prompt	Specifies the prompt to display in the command bar, such as "Enter file to edit:"
Flags	Determines how the shell command operates: • **Don't Wait** - BricsCAD immediately returns to the command prompt without waiting for the application to finish. • **Minimized** - the application runs minimized. • **Hidden** - the application does not appear on the screen. • **Quotes** - the shell command uses quotations when the command string uses spaces,such as "C:\cad programs\file name.exe".
Edit	Edits the selected shell command.
Delete	Removes the selected shell command.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Commands (6.9.37) - lists the names of internal and external commands. CuiLoad (6.9.56) and CuiUnload (6.9.57) - loads and unloads menu files. MenuLoad (6.21.14) and MenuUnload (6.21.15) - loads and unloads menu files. Netload (6.22.3) - loads a .NET application. ProfileManager (6.24.47) - allows to set current, create, copy, delete, import and export user profiles through a dialog box. ToolPalettes (6.29.19) - opens the Tool Palettes bar. VbaLoad (6.32.2) and -VbaLoad (6.32.3) - loads VBA projects. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaMan (6.32.4) - manages VBA projects. VbaUnload (6.32.9) - unloads VBA projects. Workspace (6.33.15) - sets the current workspace and lets you create, modify and save workspaces. WsSettings (6.33.18) - opens the Customize Workspaces dialog box.

Figure 6.586

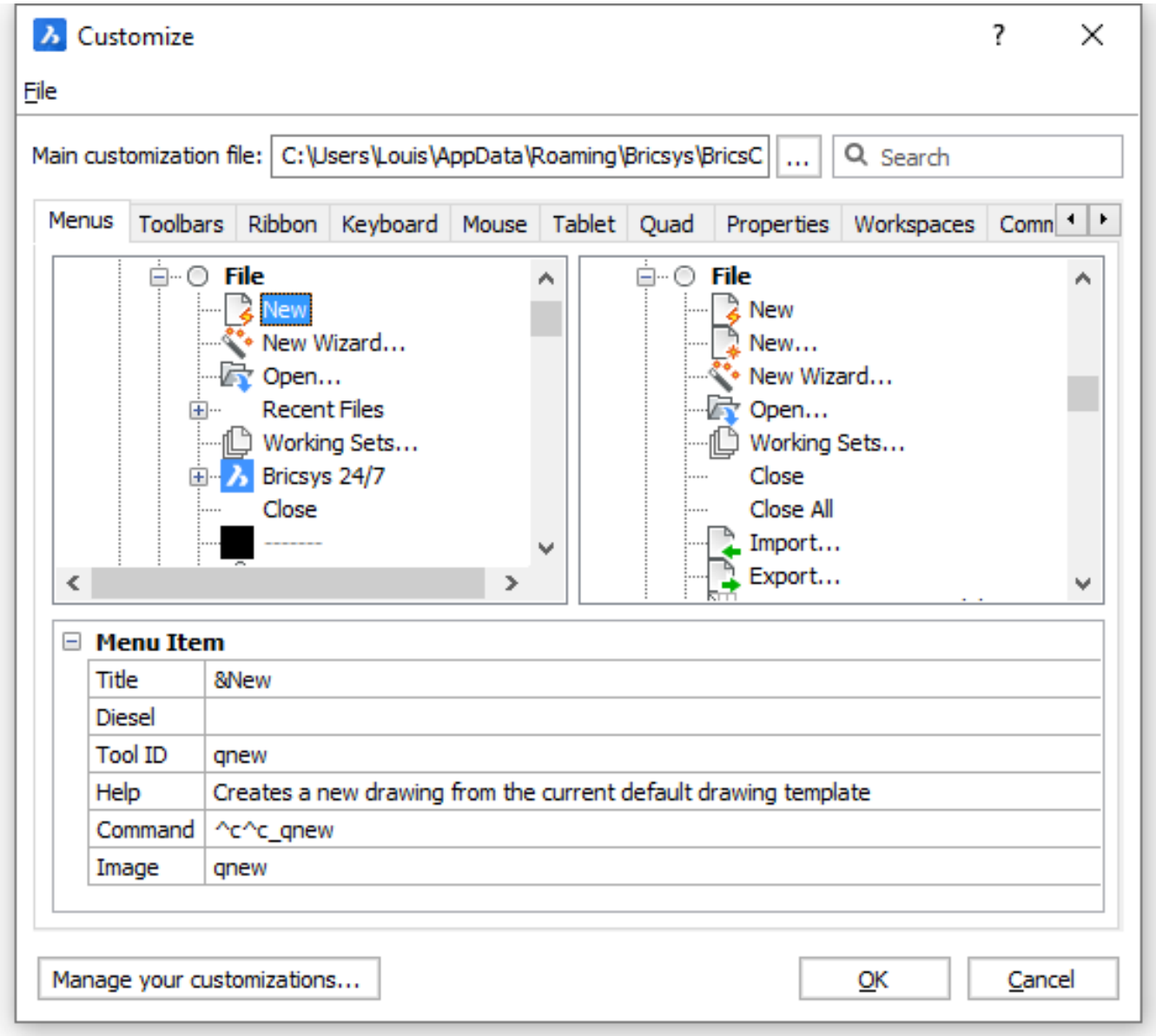


Figure 6.587

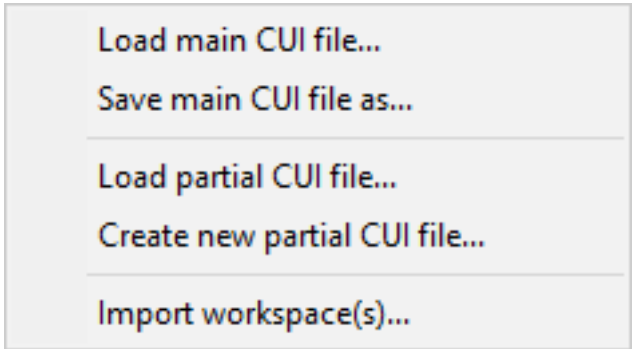


Figure 6.588

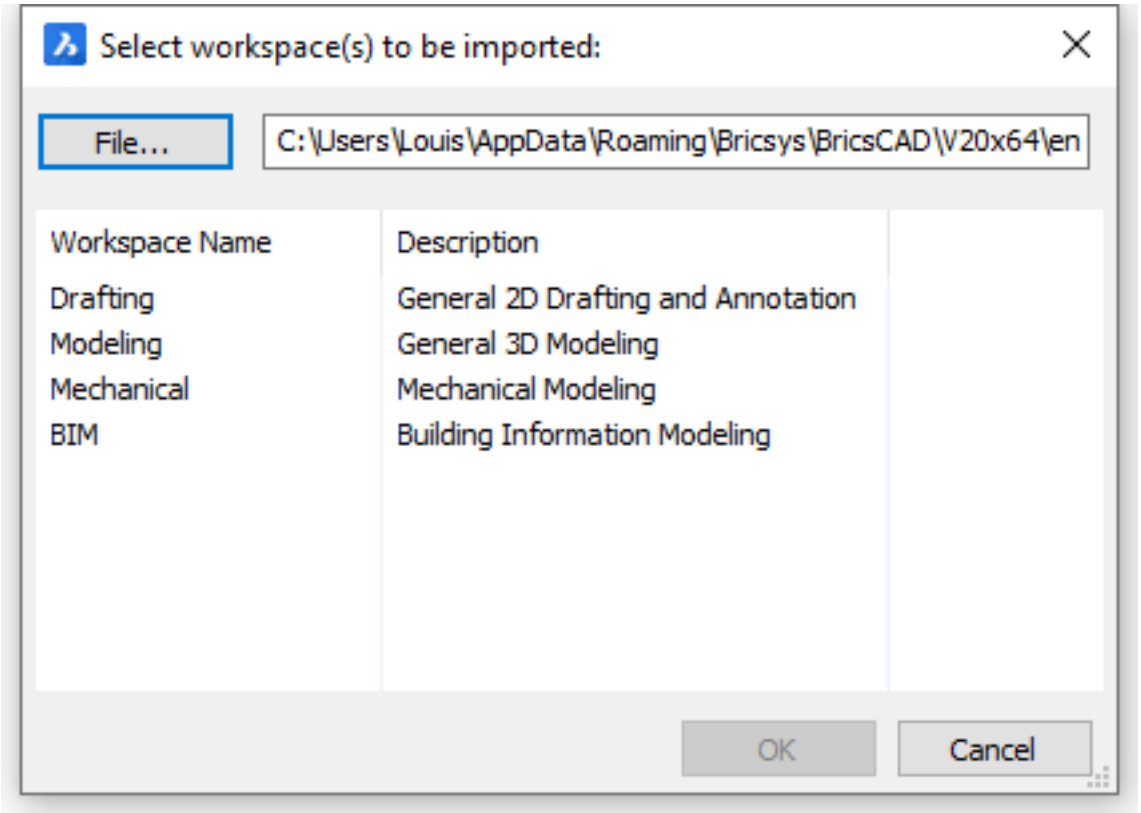

Select workspace(s) to be imported:
File...
C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V20x64\en
Workspace Name
Description
Drafting
General 2D Drafting and Annotation
Modeling
General 3D Modeling
Mechanical
Mechanical Modeling
BIM
Building Information Modeling
OK
Cancel

Figure 6.589

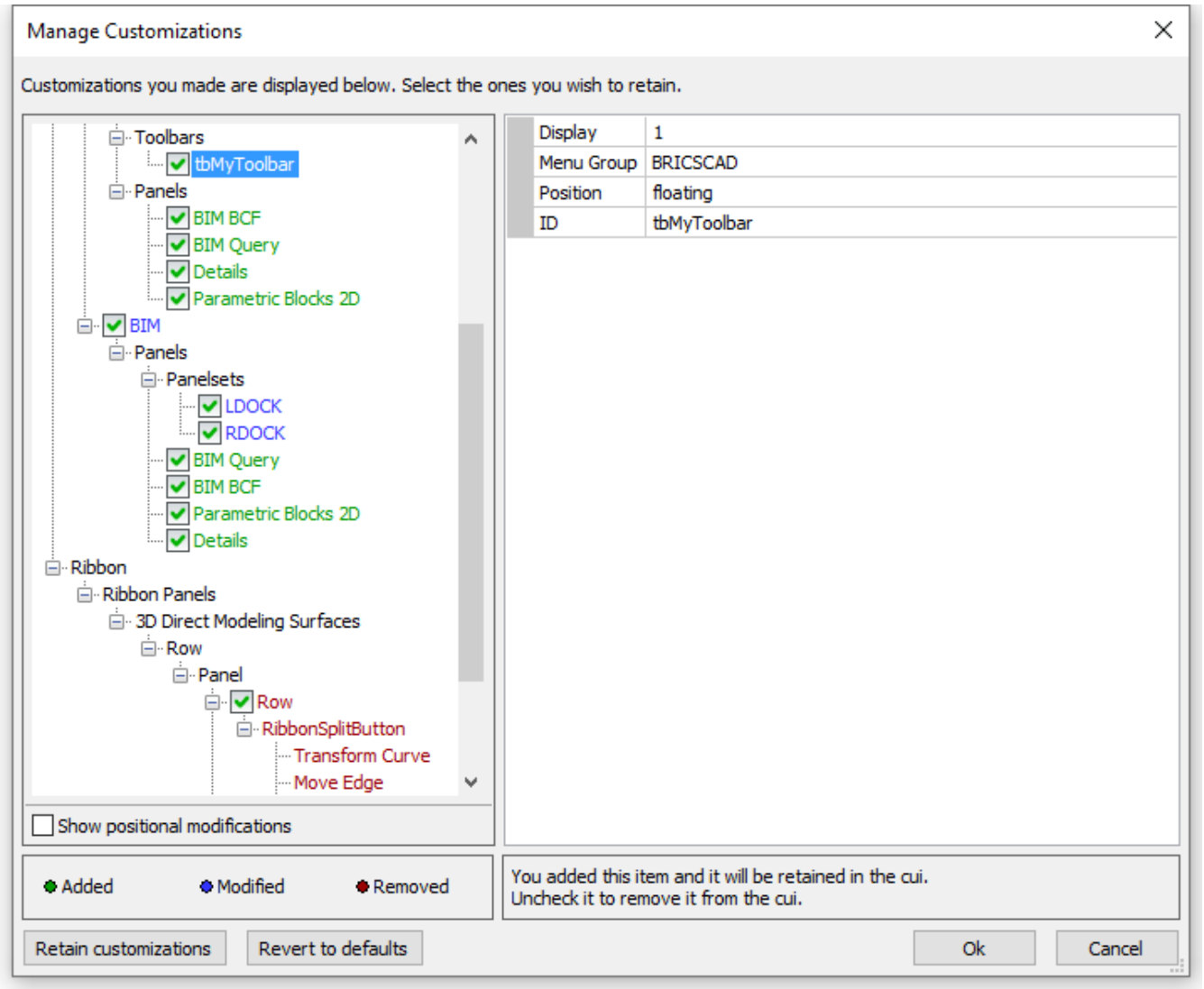

Manage Customizations
Customizations you made are displayed below. Select the ones you wish to retain.
Toolbars
tbMyToolbar
Panels
BIM BCF
BIM Query
Details
Parametric Blocks 2D
BIM
Panels
Panelsets
LDOCK
RDOCK
BIM Query
BIM BCF
Parametric Blocks 2D
Details
Ribbon
Ribbon Panels
3D Direct Modeling Surfaces
Row
Panel
Row
RibbonSplitButton
Transform Curve
Move Edge
Show positional modifications
Display 1
Menu Group BRICSCAD
Position floating
ID tbMyToolbar
Added
Modified
Removed
You added this item and it will be retained in the cui.
Uncheck it to remove it from the cui.
Retain customizations
Revert to defaults
Ok
Cancel

Figure 6.590

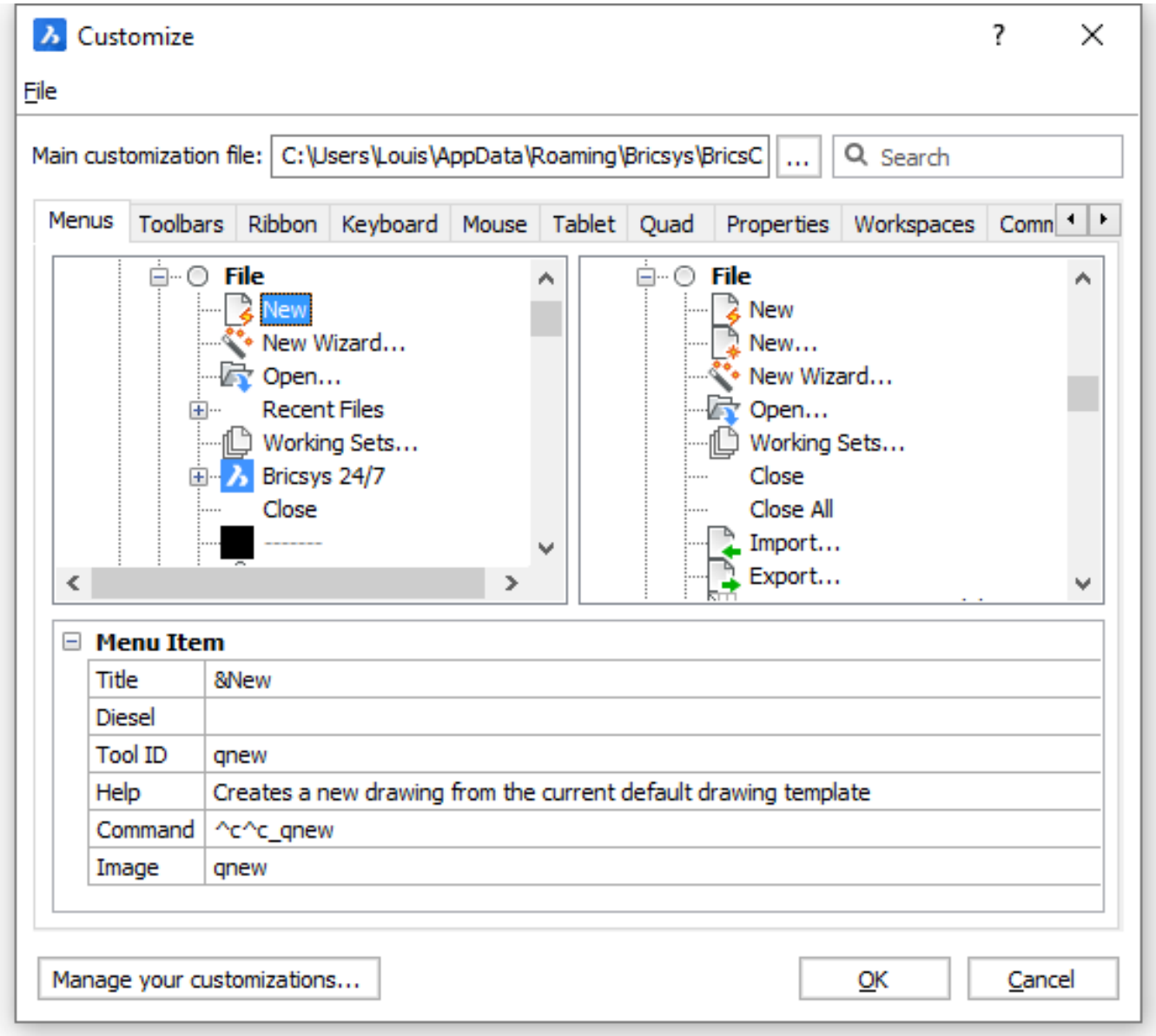


Figure 6.591

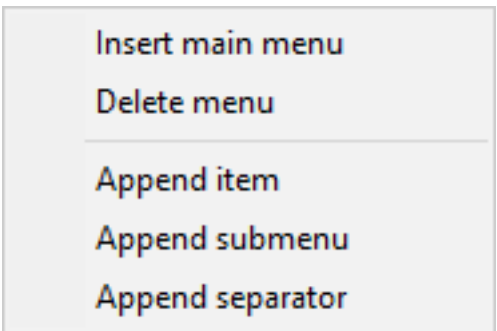


Figure 6.592

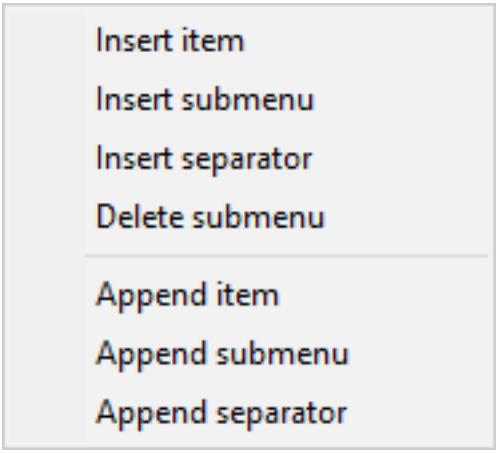


Figure 6.593

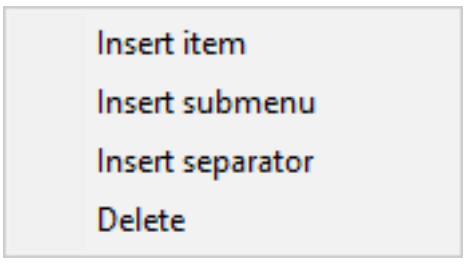


Figure 6.594

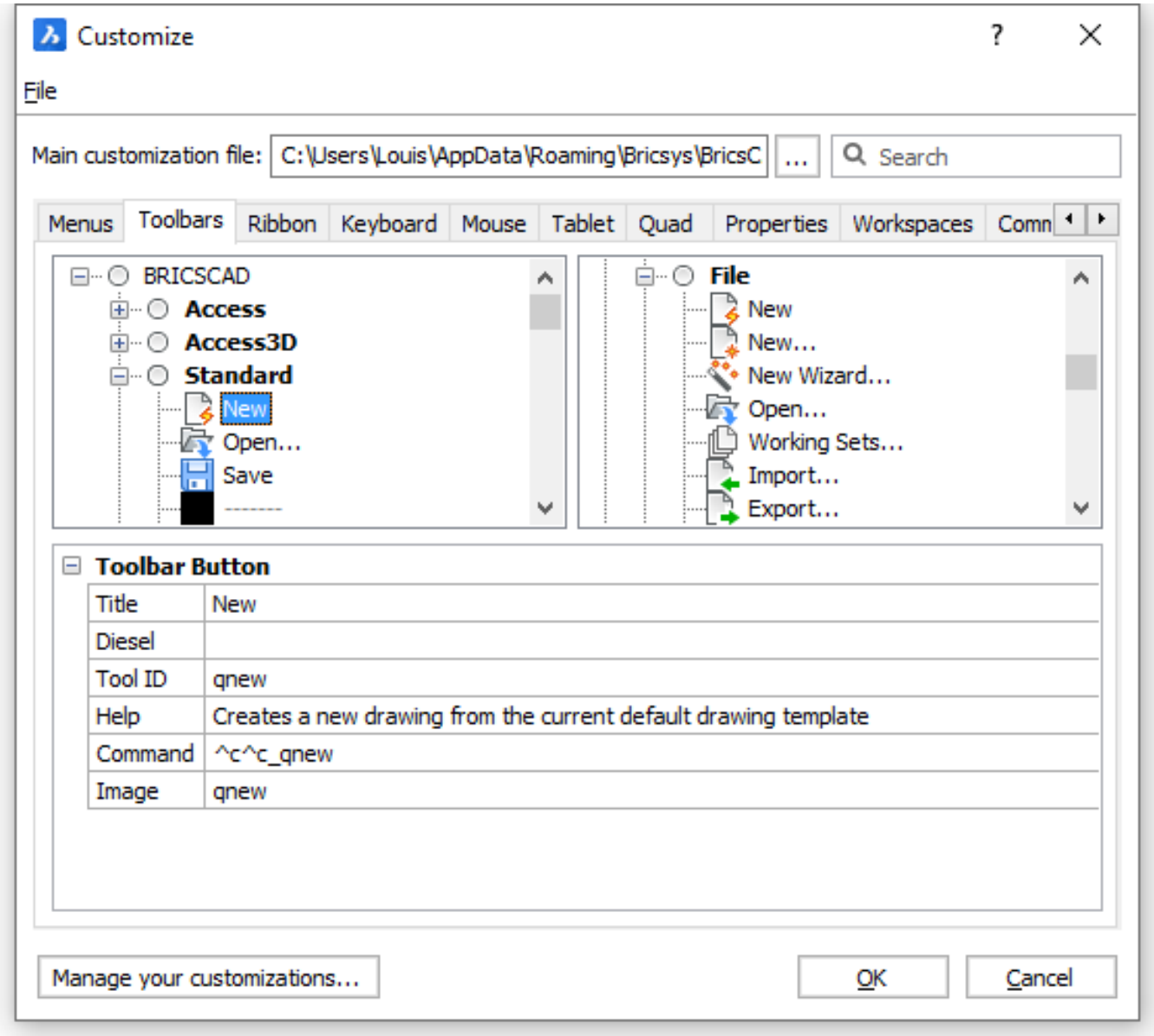

Customize
File
Main customization file:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsC
...
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Properties
Workspaces
Comn
BRICSCAD
Access
Access3D
Standard
New
Open...
Save
File
New
New...
New Wizard...
Open...
Working Sets...
Import...
Export...
Toolbar Button
Title
New
Diesel
Tool ID
qnew
Help
Creates a new drawing from the current default drawing template
Command
^c^c_qnew
Image
qnew
Manage your customizations...
OK
Cancel

Figure 6.595

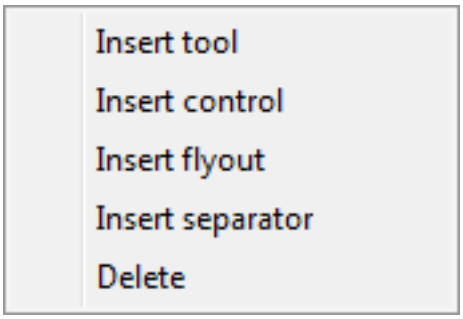


Figure 6.596

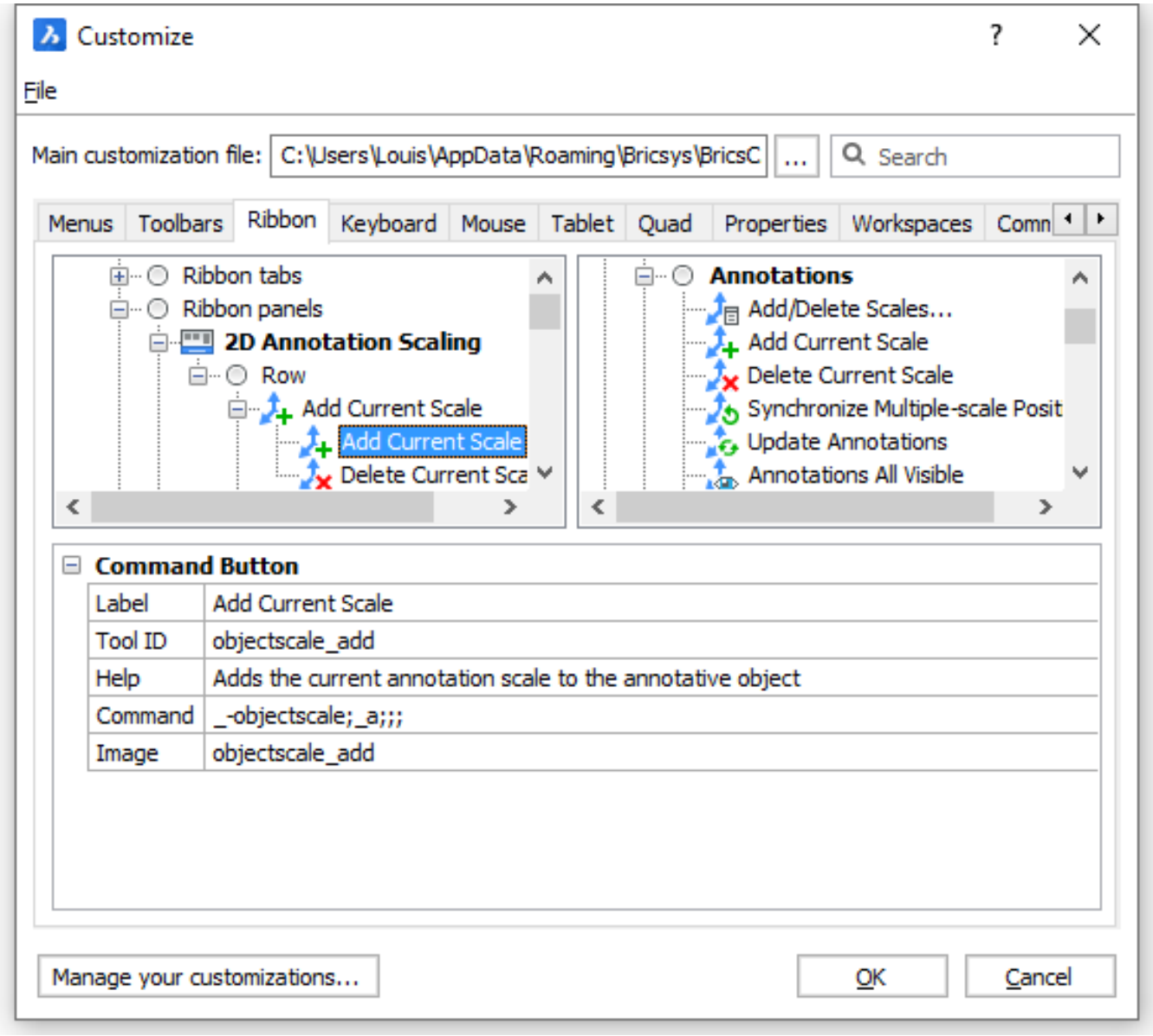


Figure 6.597

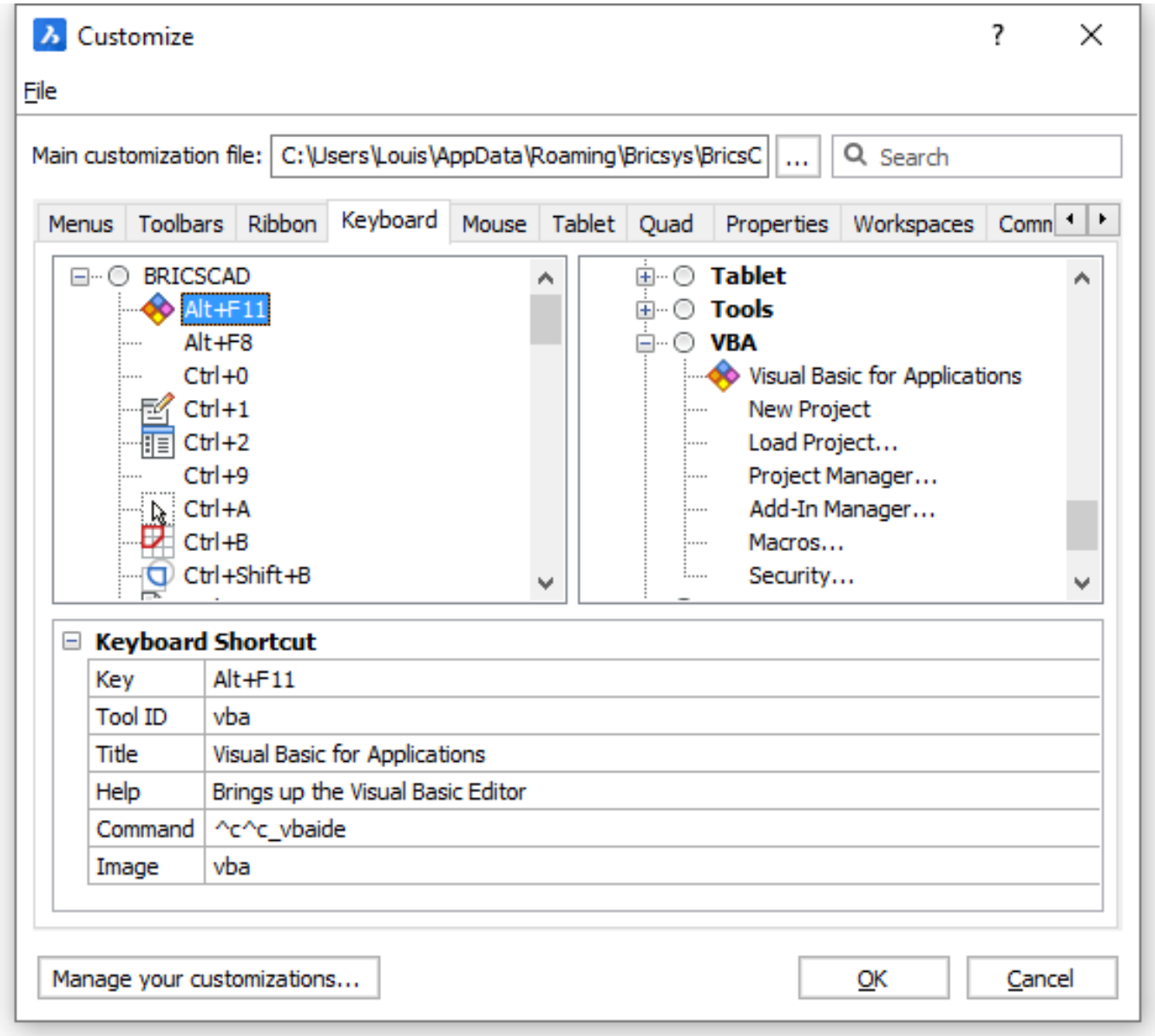


Figure 6.598

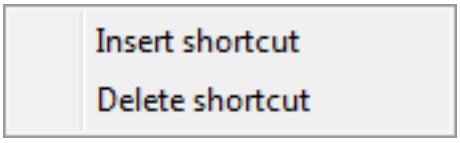


Figure 6.599

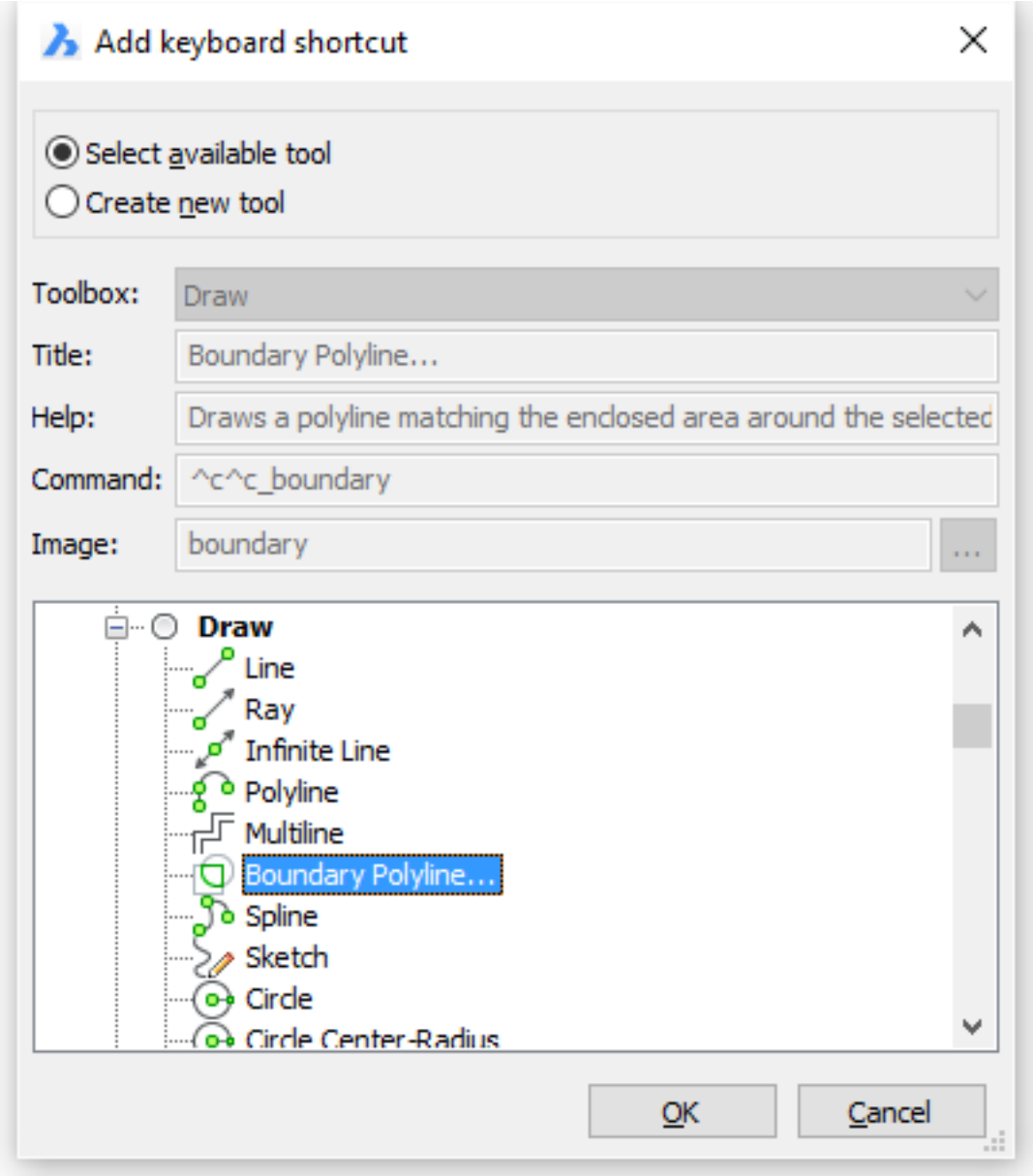


Figure 6.600

Keyboard Shortcut

Key	Ctrl+A
Tool ID	select_all
Title	Select All
Help	Selects all entities in the drawing
Command	^c^c_selgrips;_all;;
Image	select_all

Figure 6.601

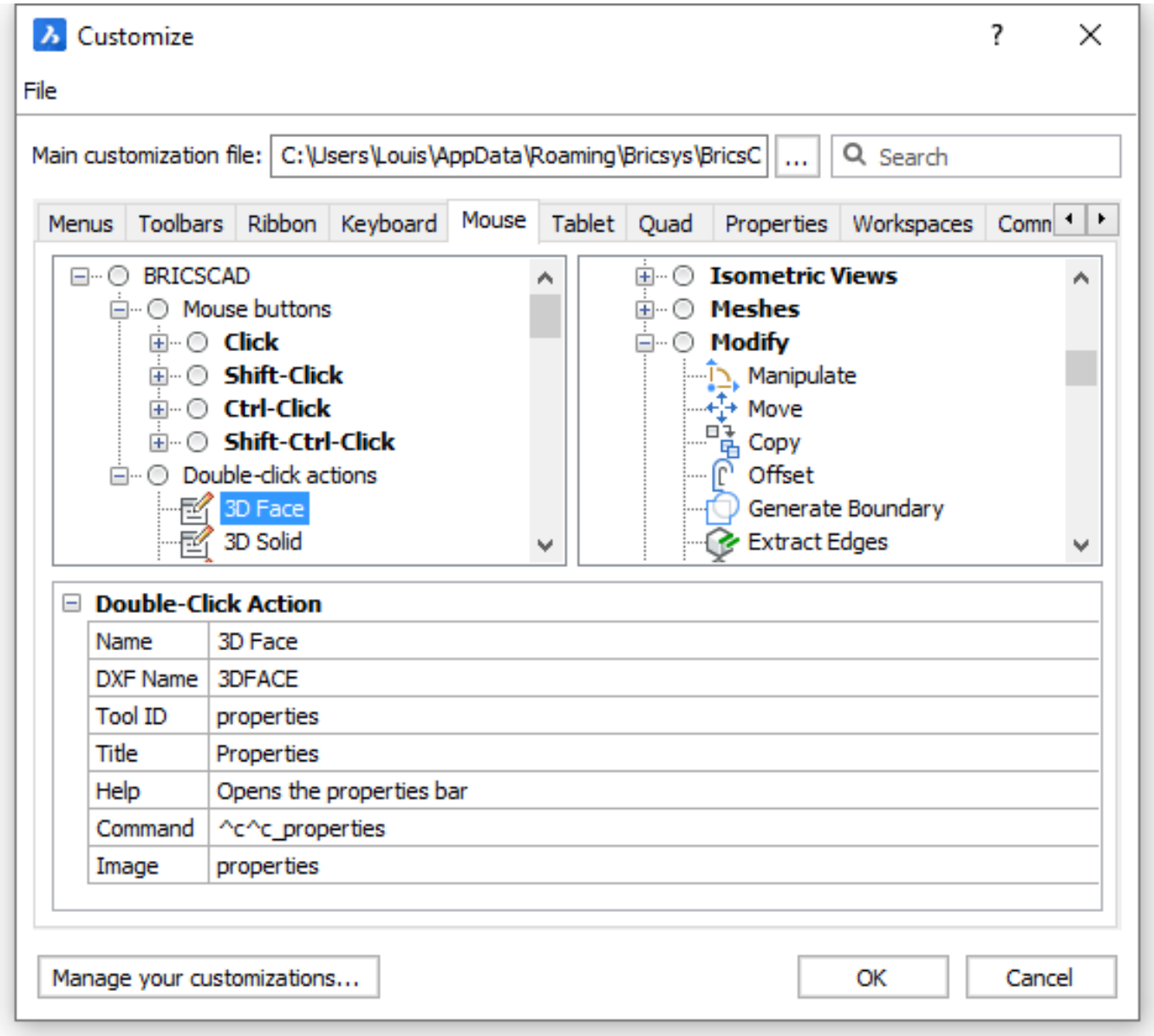

Customize
File
Main customization file:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsC
...
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Properties
Workspaces
Comn
BRICSCAD
Mouse buttons
Click
Shift-Click
Ctrl-Click
Shift-Ctrl-Click
Double-click actions
3D Face
3D Solid
Isometric Views
Meshes
Modify
Manipulate
Move
Copy
Offset
Generate Boundary
Extract Edges
Double-Click Action
Name 3D Face
DXF Name 3DFACE
Tool ID properties
Title Properties
Help Opens the properties bar
Command ^c^c_properties
Image properties
Manage your customizations...
OK
Cancel

Figure 6.602

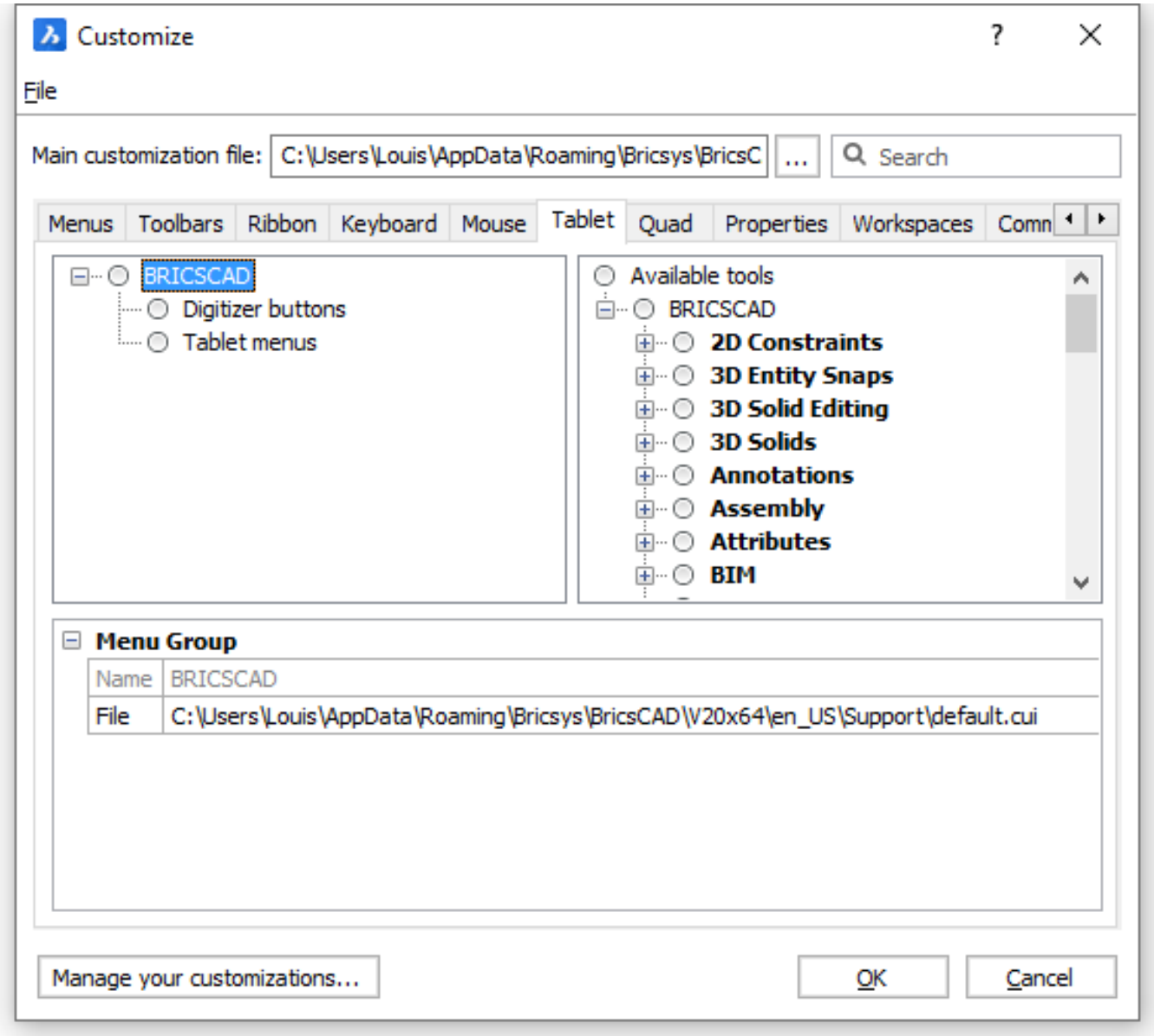

Customize
File
Main customization file:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsC
...
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Properties
Workspaces
Comn
BRICSCAD
Digitizer buttons
Tablet menus
Available tools
BRICSCAD
2D Constraints
3D Entity Snaps
3D Solid Editing
3D Solids
Annotations
Assembly
Attributes
BIM
Menu Group
Name
BRICSCAD
File
C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\Support\default.cui
Manage your customizations...
OK
Cancel

Figure 6.603

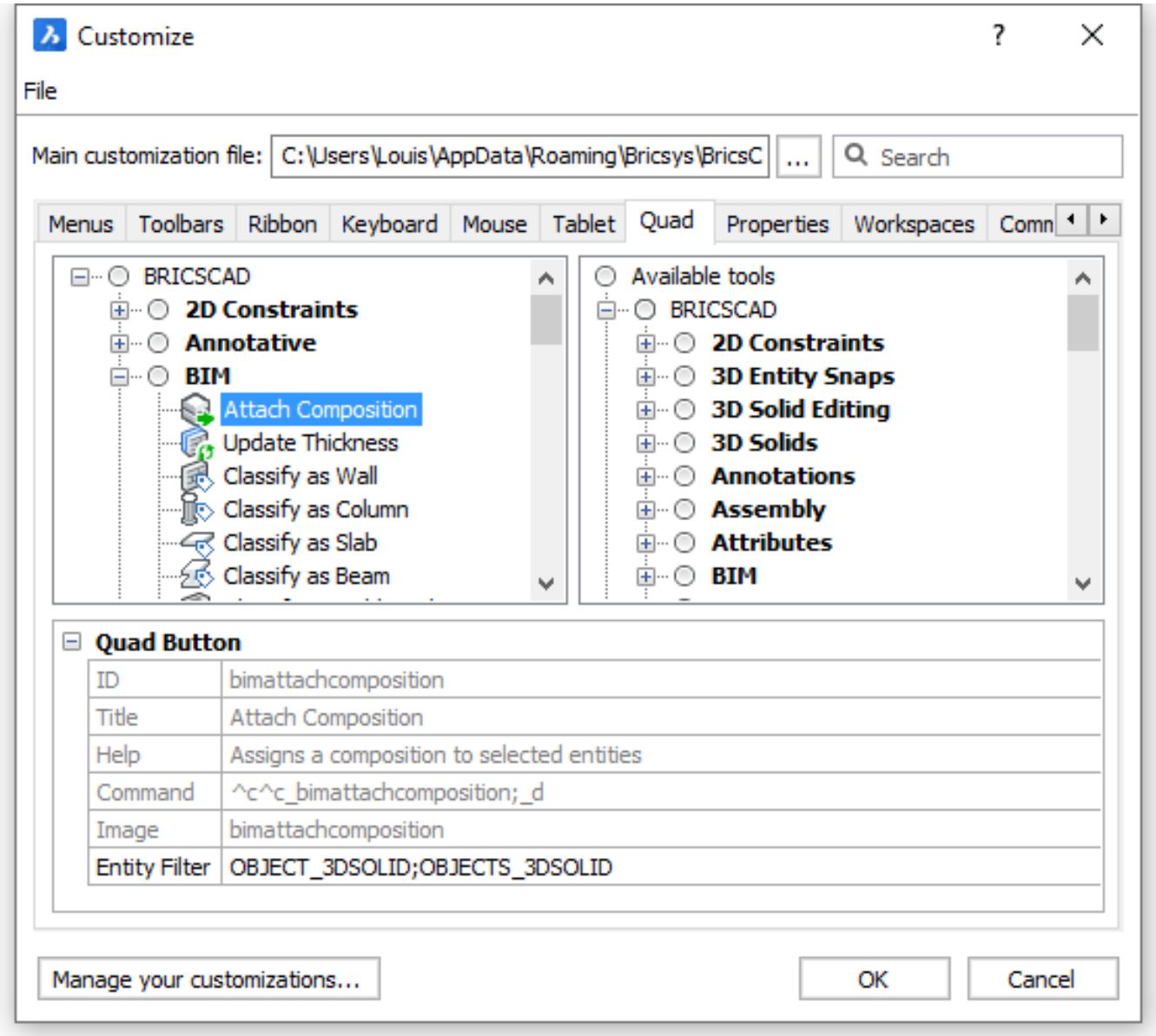


Figure 6.604

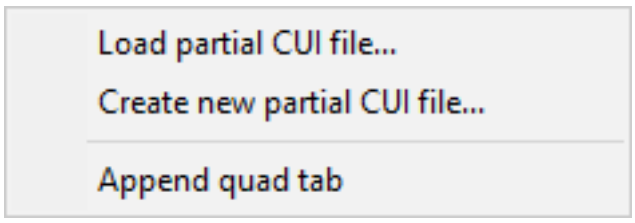


Figure 6.605

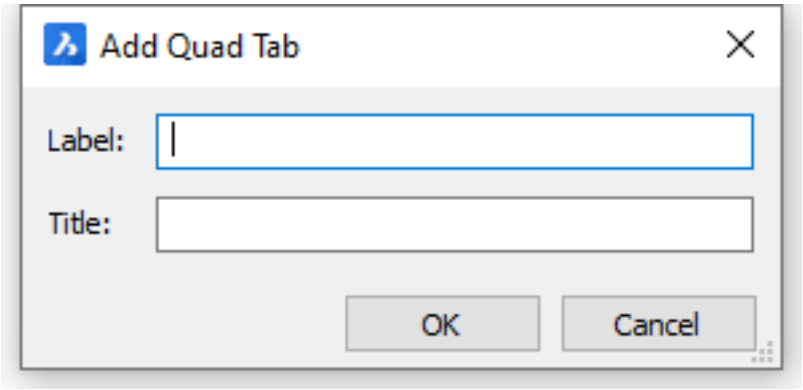


Figure 6.606

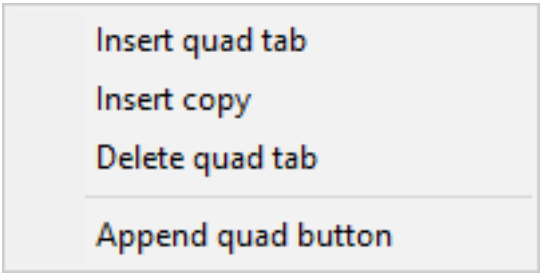


Figure 6.607

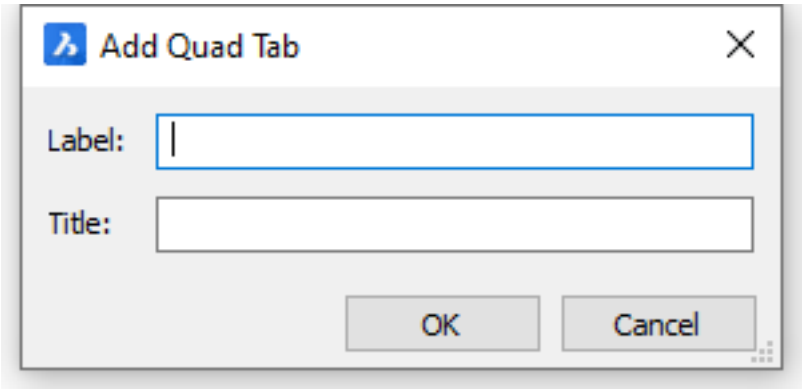


Figure 6.608

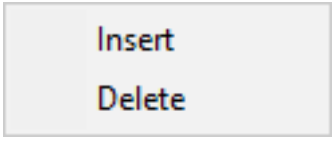


Figure 6.609

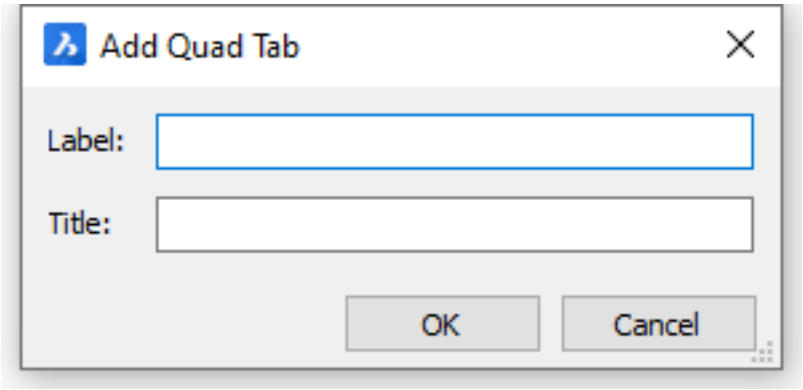


Figure 6.610

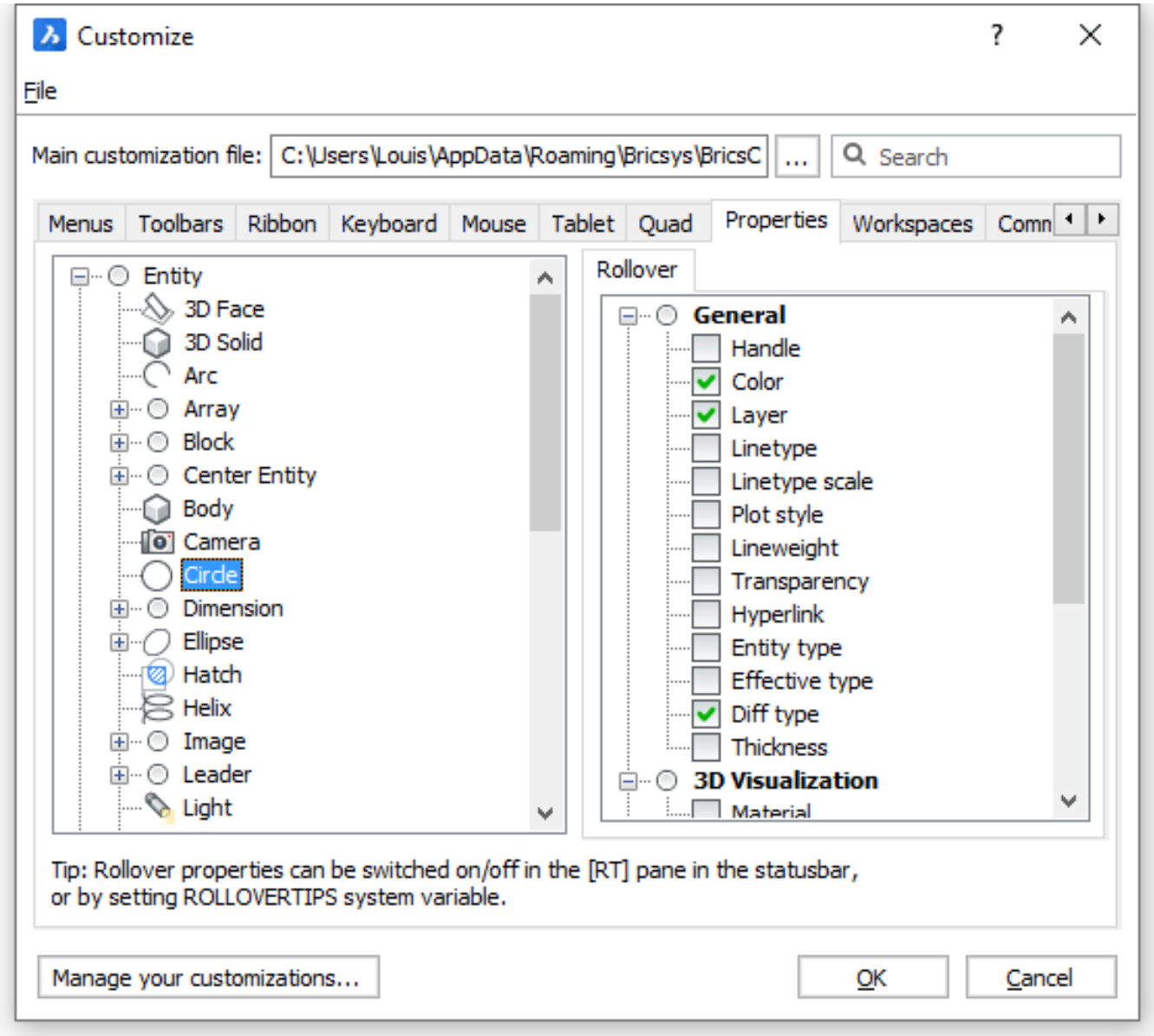

Customize
File
Main customization file:
C:\Users\Louis\AppData\Roaming\Bricsys\BricsC
...
Search
Menus
Toolbars
Ribbon
Keyboard
Mouse
Tablet
Quad
Properties
Workspaces
Comn
Entity
3D Face
3D Solid
Arc
Array
Block
Center Entity
Body
Camera
Circle
Dimension
Ellipse
Hatch
Helix
Image
Leader
Light
Rollover
General
Handle
Color
Layer
Linetype
Linetype scale
Plot style
Lineweight
Transparency
Hyperlink
Entity type
Effective type
Diff type
Thickness
3D Visualization
Material
Tip: Rollover properties can be switched on/off in the [RT] pane in the statusbar,
or by setting ROLLOVERTIPS system variable.
Manage your customizations...
OK
Cancel

Figure 6.611

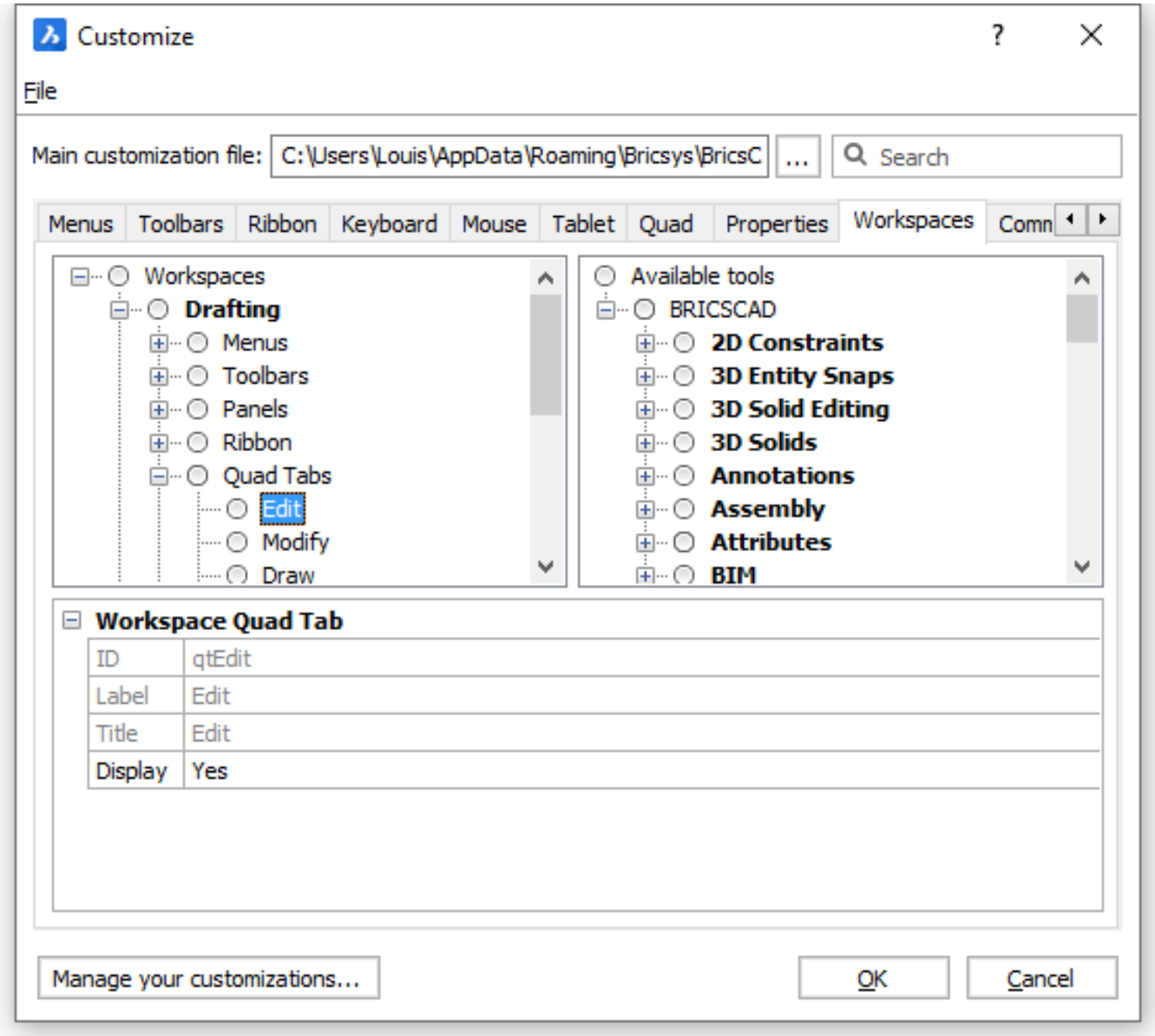


Figure 6.612

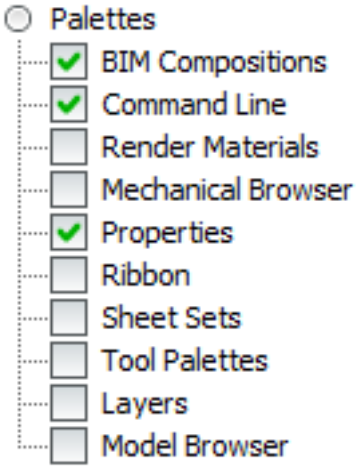


Figure 6.613

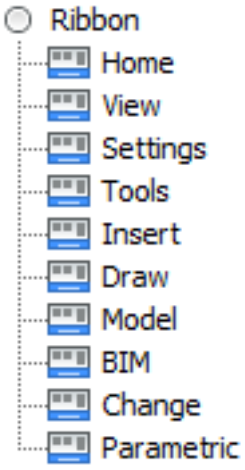


Figure 6.614

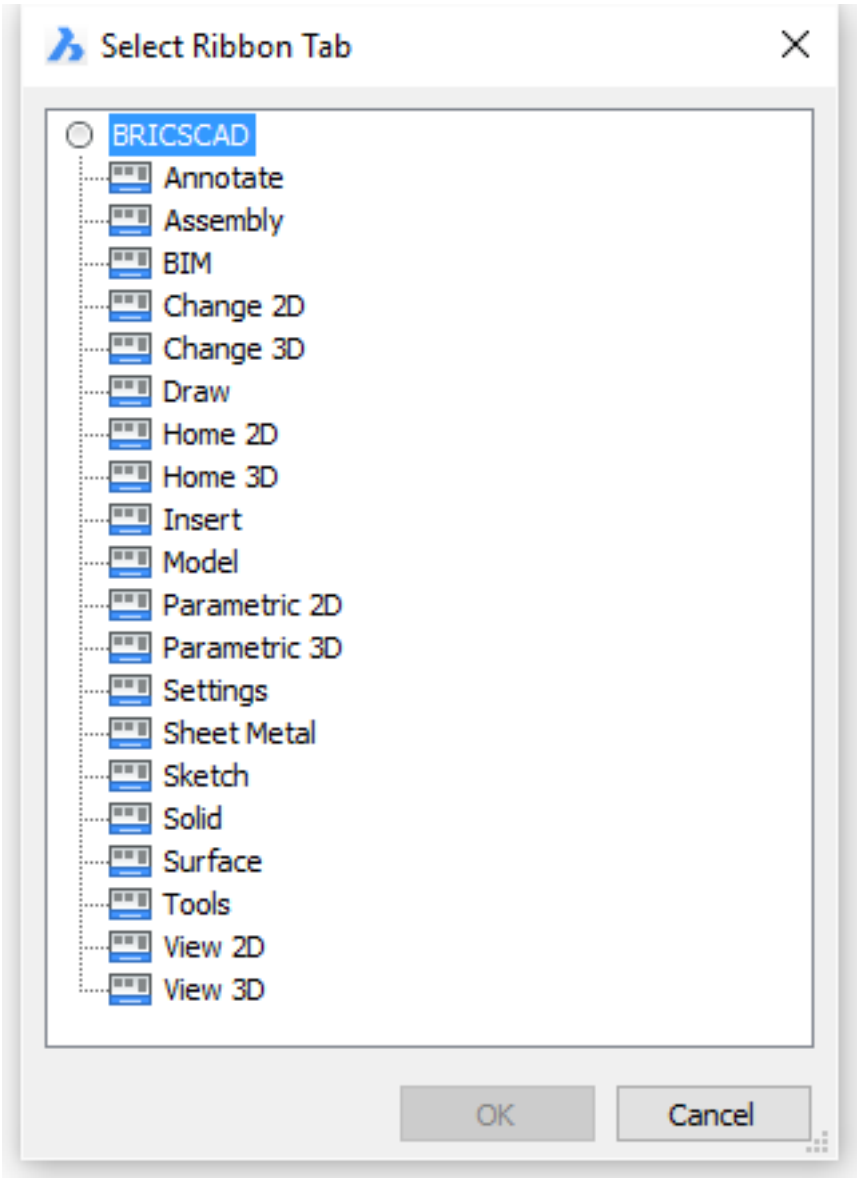


Figure 6.615

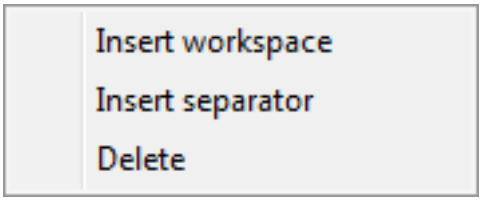


Figure 6.616

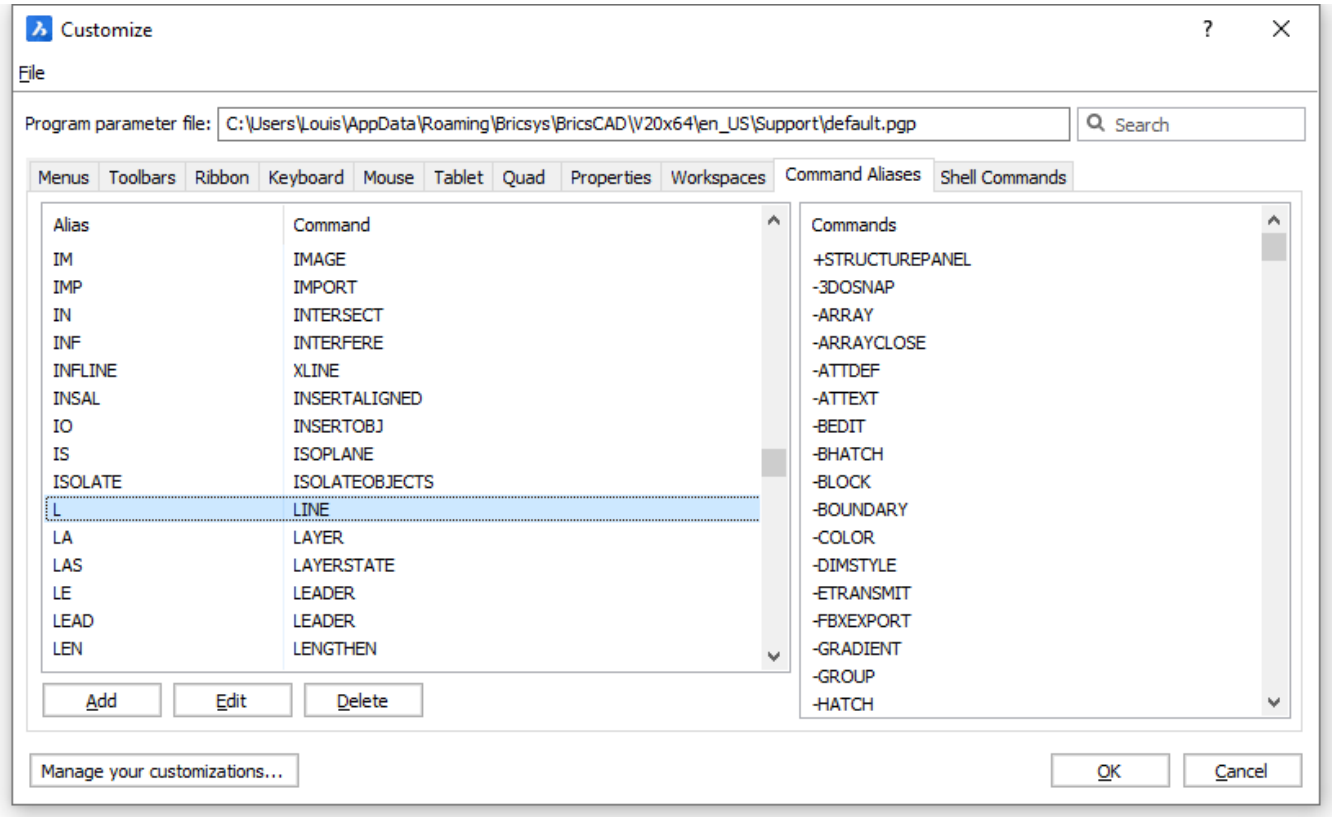


Figure 6.617

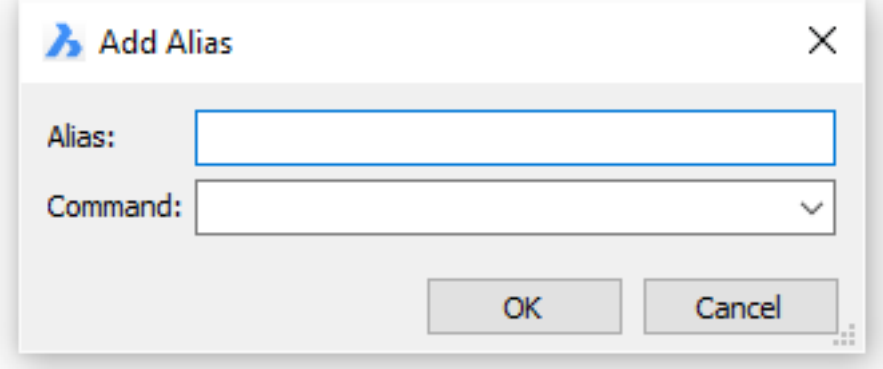


Figure 6.618

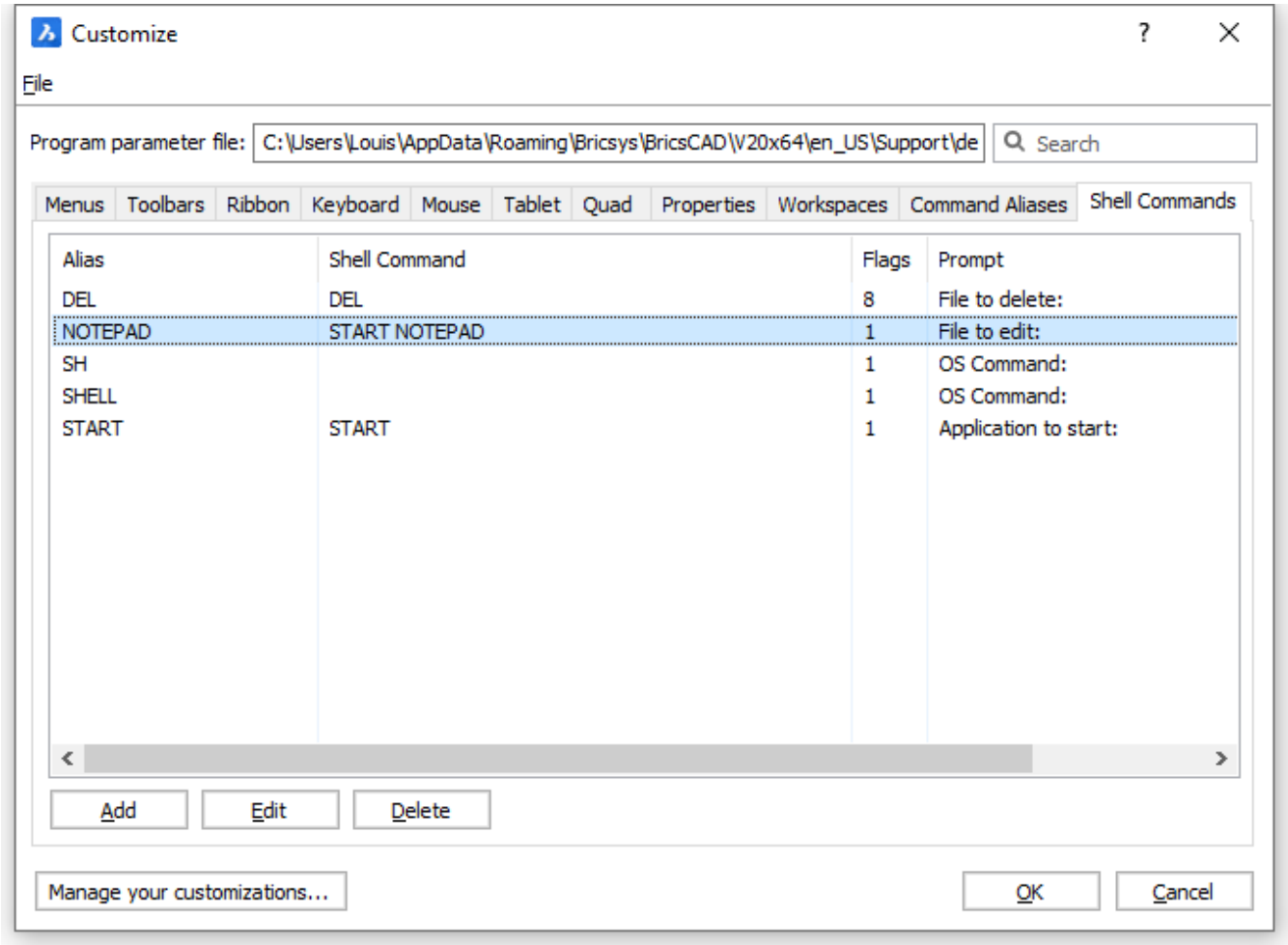


Figure 6.619

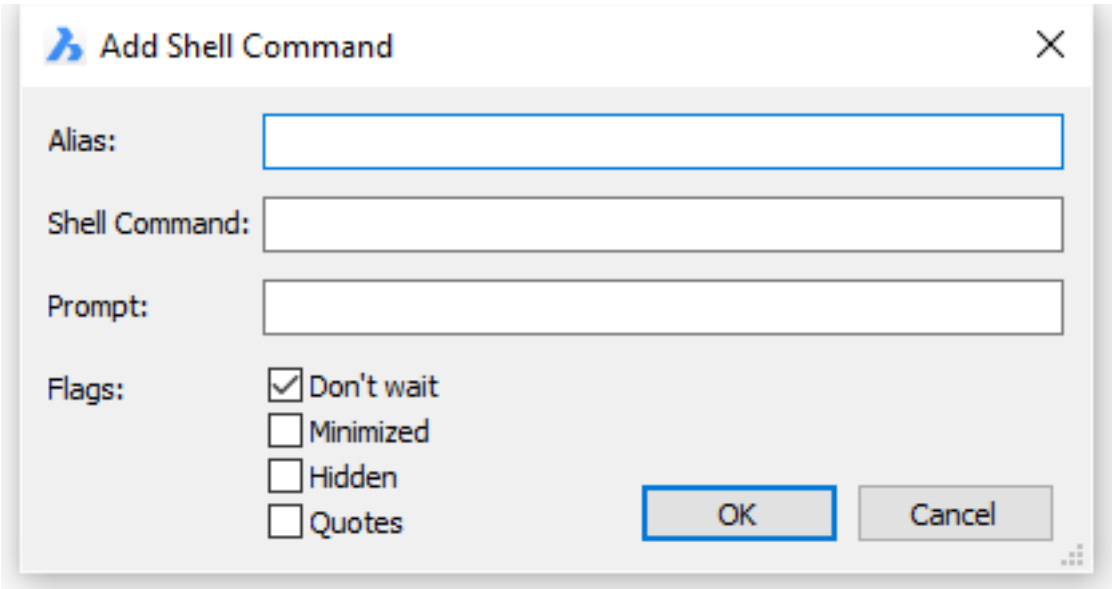


Figure 6.620



6.9.59 CutClip

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Copies entities to the Clipboard, and then erases them from the drawing.

Accessing the Command

 command bar: **cutclip** keyboard shortcut: **Ctrl+X** menu bar: **Edit | Cut** toolbar: **Standard |** ribbon:**Home | Edit | Cut** : **cutclip You are prompted:** Select entities to cut to the clipboard: **(Choose one or more entities, or press Ctrl+A to select all entities.)** Select entities to cut to the clipboard: **(Choose more entities, or eles press Enter to exit entity selection.)**

Command Options

Option	Description
Select entities to cut to the clipboard	Choose the entities to be cut; use any entity selection method. Once in the Clipboard, the entities can be pasted into the same drawing, into other drawings, and into other documents with Ctrl+V.

The **PictureExportScale** user preference sets the resolution of the image when the exported geometry is pasted in a bitmap format, e.g. in a Word document.

Procedures

- Using Cut and Paste to move entities between drawings (1.23.11)
- Move a selection as a block (1.23.11)

Related Commands

ChSpace (6.9.19) - moves entities from paper space to model space and vice versa. CopyBase (6.9.51)- copies selected entities to the Clipboard with a base point. CopyClip (6.9.52) - copies selected entities to the Clipboard. CopyHist (6.9.54) - copies all text from the Text window to the Clipboard. PasteBlock (6.24.9) - pasted entities from the Clipboard as a block at an insertion point. PasteClip (6.24.10) - pastes entities from the Clipboard into the current drawing. PasteOrig (6.24.41) - pastes entities from the Clipboard at the original coordinates. PasteSpec (6.24.11) - displays the Paste Special dialog box for controlling the format of pasted entities.

Figure 6.621



6.9.60 Cylinder

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws regular and slanted cylinders:

- Classic: cylider is drawn from a 3D polyface mesh (see Ai_Cylinder (6.5.5) command).
- Shape, Pro and Platinum: cylinder is drawn as a 3D solid.

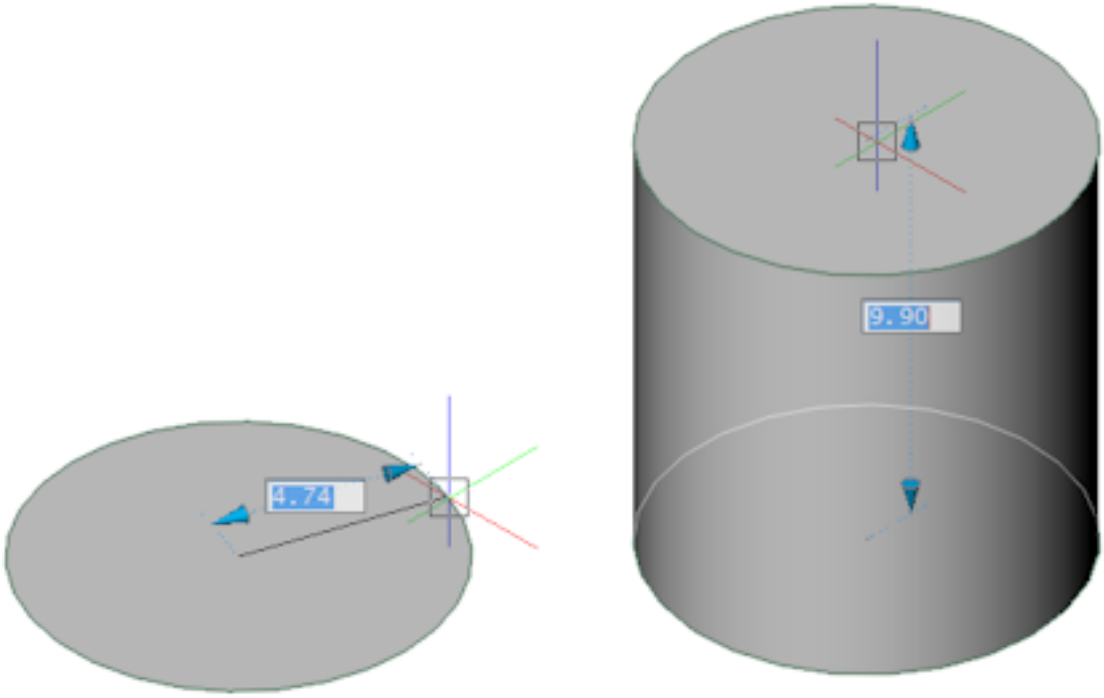
Accessing the Command

 command bar: **cylinder** alias: **cyl** menu bar: **Model | 3D Solids | Cylinder** toolbar: **3D Solids |**  ribbon: **Solid | Create | Primitive | Cylinder** : **cylinder You are prompted:** Select center point or [3Point (6.9.20)/2Point (6.9.20)/Tangent-tangent-radius (6.9.20)/Elliptica (6.13.5)l (6.13.5)]**: (Enter coordinates, pick a point, or else choose an option)** Radius of base of cylinder or Diameter <x.xx>: **(Enter a distance, pick a point, press Enter to accept the previously entered value, or else enter D)** Specify height or [2Point/Axis endpoint] <x.xx>: **(Enter a height, pick a point, press Enter to accept the previously entered value, or else choose an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image85.png}) If Dynamic Dimensions (1.15.9) are active, you can enter the dimensions in the dynamic dimensions entry fields. (\vref {fig:C:/Dev/Task-2448/images/staticbric

Figure 6.622



Figure 6.623



Command Options

Option	Description
Diameter	Specifies the radius of the cylinder; prompts you: **Diameter of base of cylinder <x.xx>** - **(enter x,y,z coordinates, a distance, pick a point, or else press Enter to accept the previously entered value.)**
2Point	Specifies the height of the cylinder as the distance between two points.Prompts you: 1. **Specify first point** - **(enter coordinates or pick a point)** 2. **Specify second point** - **(enter coordinates or pick a point)**
Axis endpoint	Specifies the endpoint of the axis, allowing to create a cylinder of which the axis is not parallel to the Z-axis of the current coordinate system.Prompts you: **Specify axis endpoint** - **(enter coordinates or pick a point)**

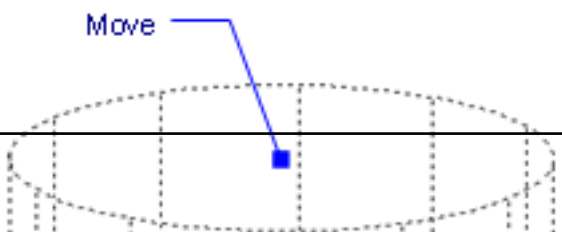
Grips Editing

Cylinders can be edited directly through grips:

1. Select the cylinder. Notice that it has two grips.
2. Drag a grip to move the cylinder. (Both grips perform the same function: move.)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image754.png})

Figure 6.624



Related Commands

Box (6.6.17) - creates 3D boxes as solid models. Cone (6.9.63) - creates 3D cones as solid models. Dish (6.10.30) - creates the lower half of 3D balls as solid models. Dome (6.10.34) - creates the upper half of 3D balls as solid models. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Pyramid (6.24.58) - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic) Region (6.26.18) - creates 2D regions from 2D entities. Revolve (6.26.30) - revolves 2D entities into 3D solid models. Section (6.27.11) - creates 2D regions from 3D models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Sphere (6.27.64) - creates 3D balls as solid models. Subtract (6.27.56) - removes one solid model from a second. Torus (6.29.22) - creates 3D donuts as solid models. Union (6.30.6) - joins two or more solid models into a single entity. Wedge (6.33.7) - creates 3D wedges as solid models.

Figure 6.625



6.9.61 ComponentsPanelOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Components panel.

Accessing the Command

command bar: **componentspanelopen** context menu: right-click any toolbar or ribbon, and then choose **Components**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/Interface/DockablePanelsList.png}) Displays the **Components** panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Comp

1. Options menu
2. Search field
3. Navigation toolbar
4. Folder and Component thumbnails

Figure 6.626

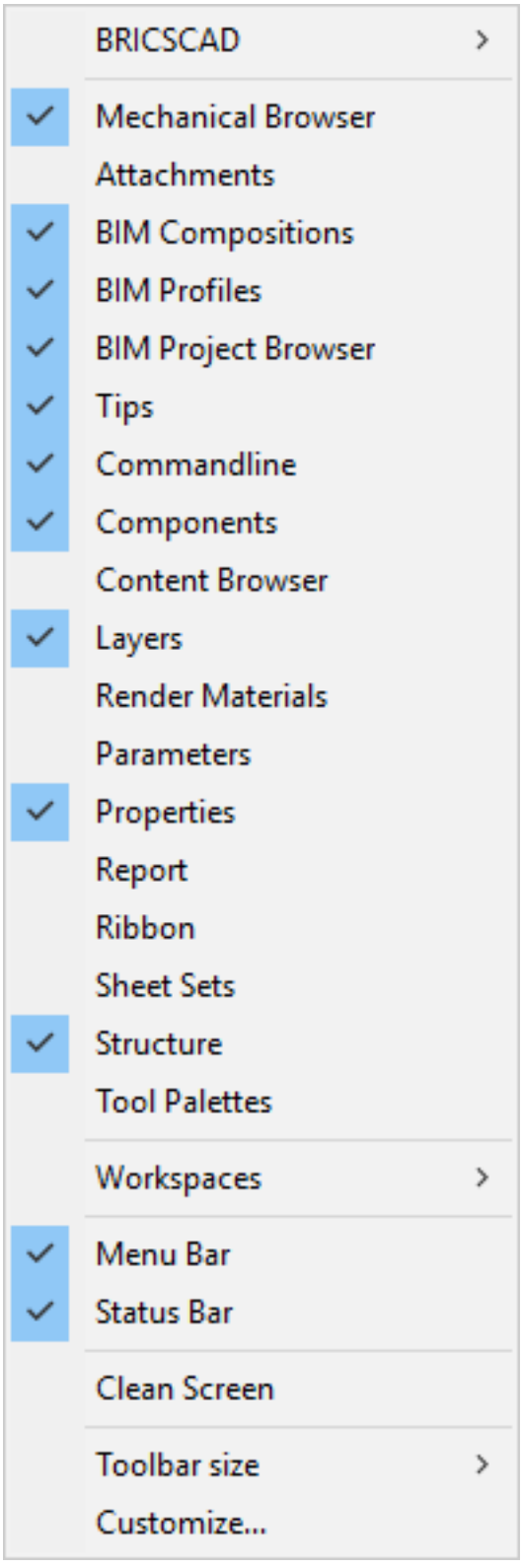
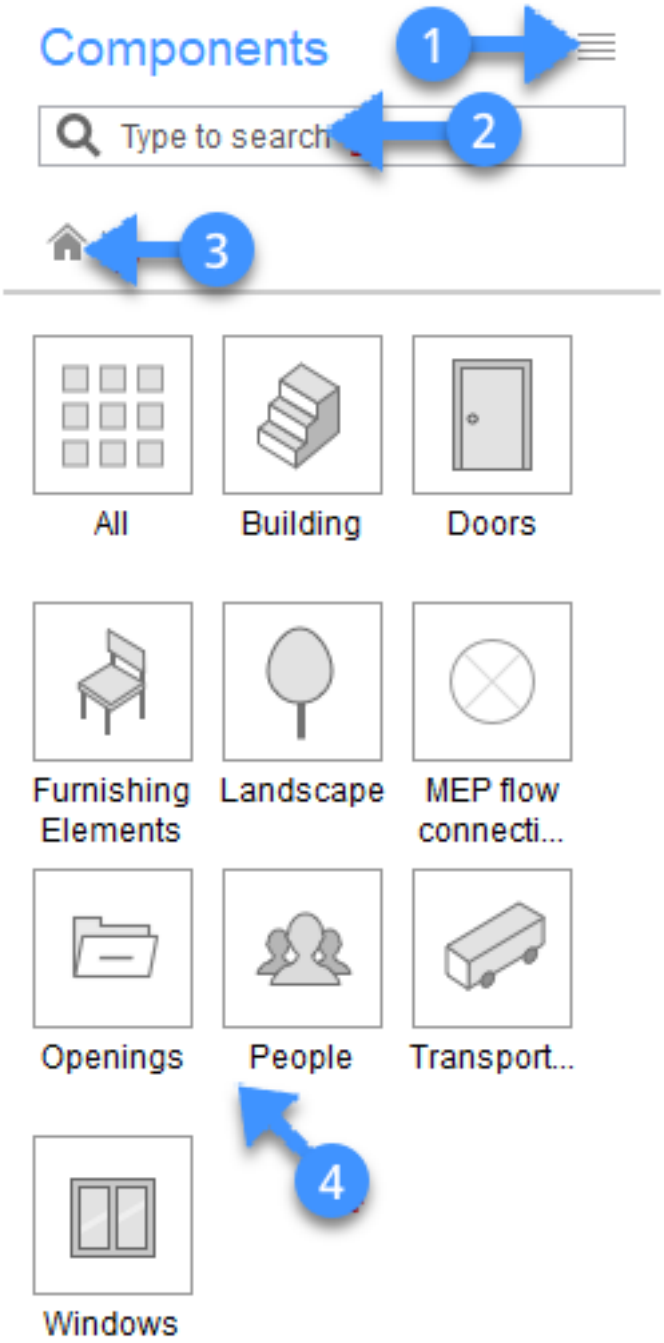

BRICSCAD
Mechanical Browser
Attachments
BIM Compositions
BIM Profiles
BIM Project Browser
Tips
Commandline
Components
Content Browser
Layers
Render Materials
Parameters
Properties
Report
Ribbon
Sheet Sets
Structure
Tool Palettes
Workspaces
Menu Bar
Status Bar
Clean Screen
Toolbar size
Customize...

Figure 6.627


Components
1
Type to search
2
3
All
Building
Doors
Furnishing Elements
Landscape
MEP flow connecti...
Openings
People
Transport...
4
Windows

Command Options

Option	Description
Options menu (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Comp.Comp.Panel.Options.Menu.png})	**Grid View**: Displays the component thumbnails in a grid pattern. Component names are truncated. Hover over a component icon to see its full name. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Co **List View**: Displays a listed view of the component thumbnails. Component names are not truncated. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Co **Generate Thumbnails**: Generates missing thumbnails and removes thumbnails of missing components.
Search components (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Comp.Comp.Panel.Search.type})	Type a text string in the search field. The component thumbnails are filtered as you type, displaying components of which the name contains the text string only.
Navigation toolbar (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Comp.Comp.Panel.Search.type})	: Browse home. Shows the content of the parent components folder(s). Furnishing Elements: Displays the name of the current components folder. : Displays the parent folder of the current components folder.
Thumbnails  All	**All** thumbnail: Click to display all components in all folders and subfoldersreferred to by theCOMPONENTSPATH (1.13.11)system variable. Select the variable in the **Program Options / Files** section on the **Settings (6.27.23)** dialog. Click the field, then click the **Browse** button ( ) to manage the components search paths: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/C/Co • Click the **Add path**button ( ) to add a folder. • Click the **Delete path**button ( ) to remove a folder. • Select a path, then click the arrow buttons to change the folder order.
 Building	**Folder** thumbnail: Click to display the components and subfolders (if any).



Window 2x2

Component thumbnail: Click then move the cursor to the model to insert the component.

Procedures

- Managing dockable panels (1.12.7)

Related Commands

-BmCreateComponent (6.8.3) -Creates a component from a selection set or drawing and add the component to the library. ComponentsPanelClose (6.9.38) - Closes the Components panel.

Figure 6.628

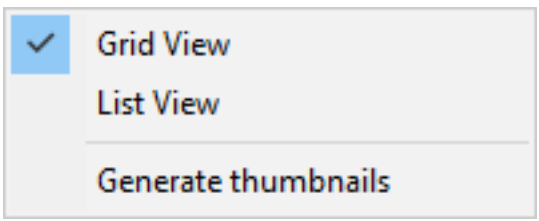


Figure 6.629

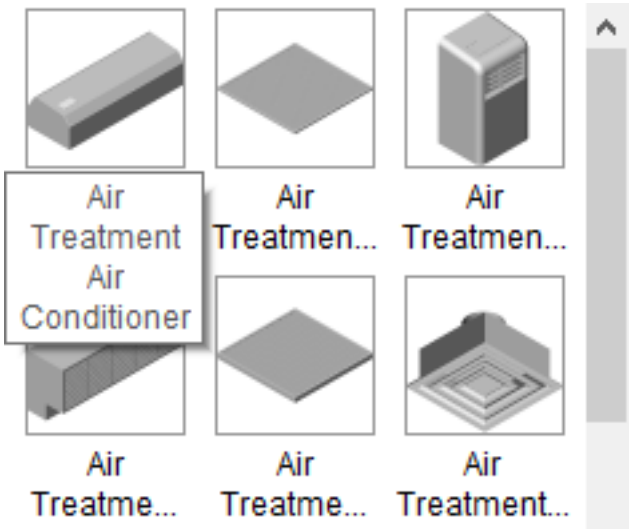


Figure 6.630

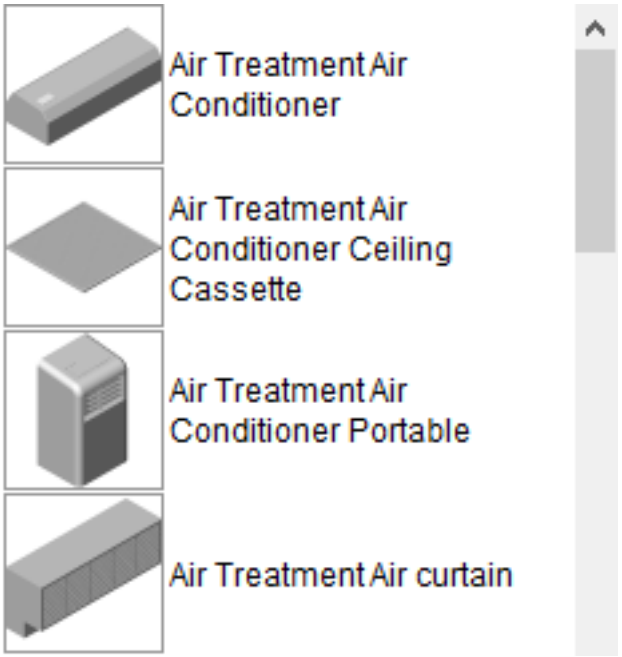


Figure 6.631



Figure 6.632

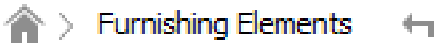


Figure 6.633

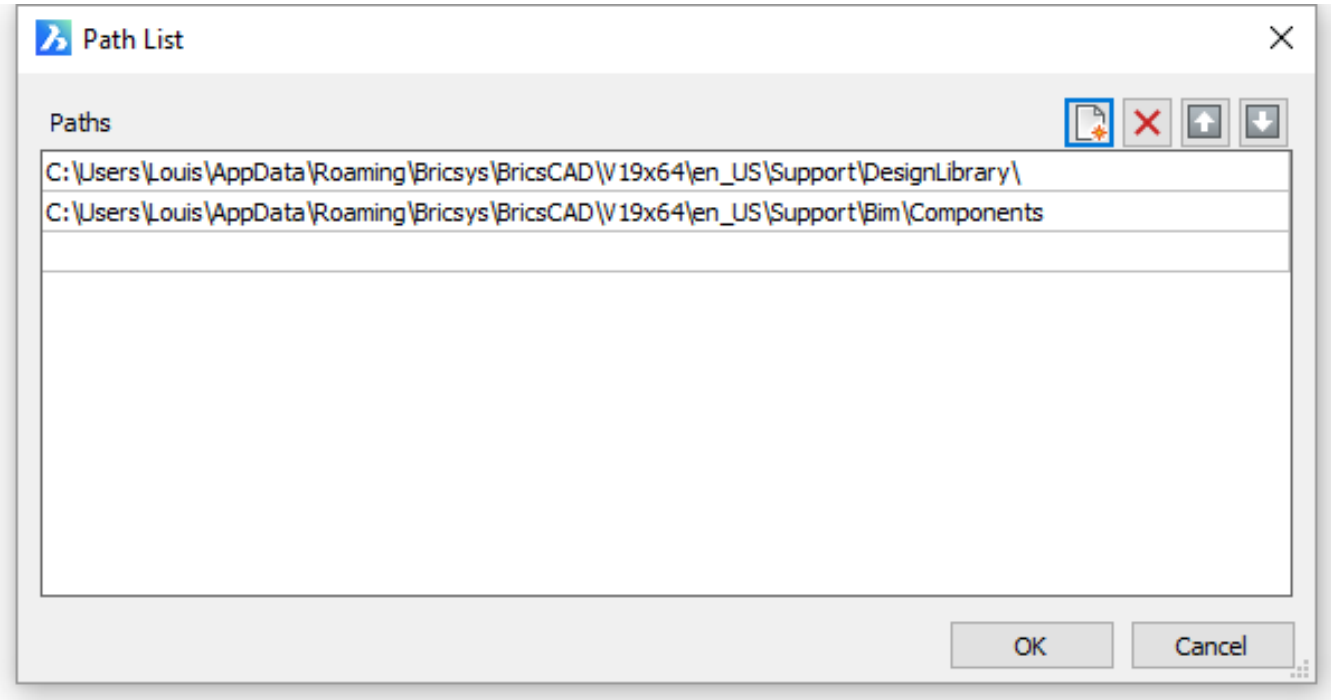

Path List
Paths
C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\Support\DesignLibrary\
C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\Support\Bim\Components
OK
Cancel

Figure 6.634



6.9.62 ConstraintBar

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Shows and hides constraint bars next to entities constrained with geometric constraints. (Constraint Bars are initially hidden when a drawing is opened.)

Accessing the Command

 command bar: **constraintbar** menu bar: **Parametric | Show/Hide Geometrical Constraint** toolbar: **2D Constraints |**  ribbon: **Prametric | 2D C**: constraintbar**onstraints | Show/Hide You are prompted:** Select entities: **(Select one or more entities, or enter all to select all entities in the drawing)** Entities in set: 1 Select entities: **(Select more entities, or else press Enter)** Select option for constraints [Show/Hide/Reset] <Show>: **(Select an option)**

Command Options

Option	Description
Show	Displays the constraint bar next to the entities you selected.
Hide	Hides the constraint bar next to the entities you selected.
Reset	Relocates constraint bars to their default positions, which is near the midpoint of the entity).

Related Commands

Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. Geomconstraint (6.15.16) - applies geometric relationships between entities and/or valid constraint points on entities. DcDisplay (6.10.9) - allows to set dimensional constraints to be displayed or hidden. Delconstraint (6.10.24) - Removes all dimensional and geometrical constraints from an entity.

Figure 6.635



6.9.63 Cone

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates cones and truncated cones:

- Classic: cone is drawn from a 3D polyface mesh (see the Ai_Cone (6.5.4) command).
- Shape, Pro and Platinum: cone is drawn as a 3D solid

Accessing the Command

 command bar: **cone** menu bar: **Model | 3D Solids | Cone** toolbar: **3D Solids |**  ribbon:**Solid | Create | Primitive | Cone : cone You are prompted:** Select center point or [3Point/2Point/Ttr/Elliptical]: **(Pick a point, or enter an option)** Radius of base of cone or Diameter <xx.xx>: **(type a value, or pick a point to define the base radius, press Enter to accept the default value, or else type D to define the diameter)** Specify height or [2Point/Axis/Top] <xx.xx>: **(Type a value, pick a point, press Enter to accept the default value, or else enter an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image74.jpg}) WhenDynamic Dimensions (1.15.9) are active, you can enter the dimensions in the dynamic dimensions entry fields. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ConeDynDim.png})

Figure 6.636

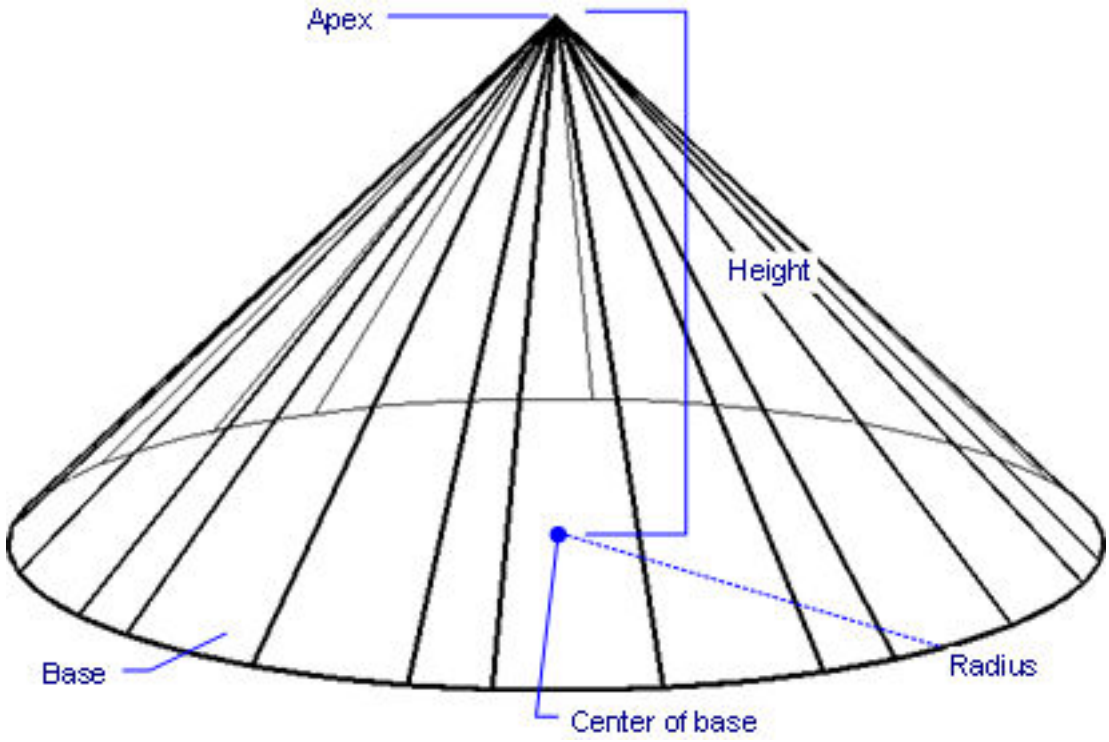
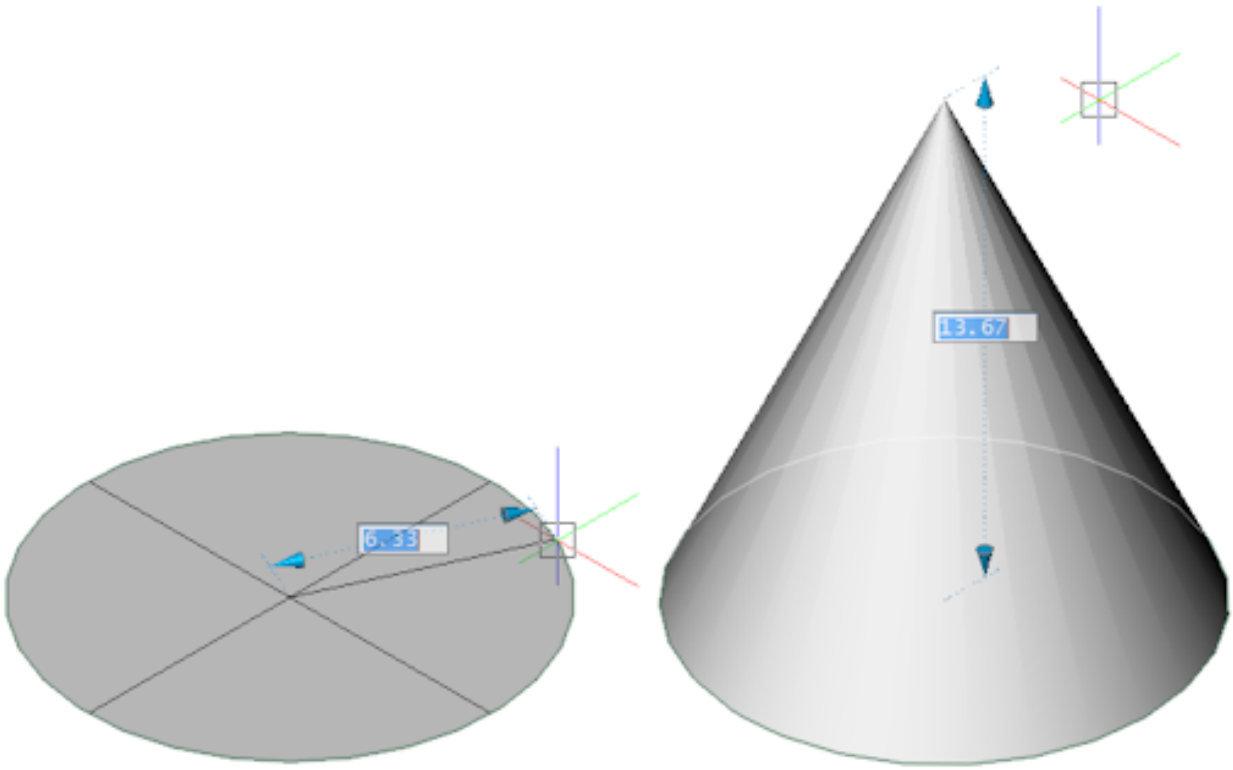


Figure 6.637



Command Options

Base Options	Description
3Point	Defines a circular base through three points that indicate the circumference.Prompts you: 1. **First point** - **(specify a point)** 2. **Second point** - **(specify a point)** 3. **Third point** - **(specify a point)** 4. **Specify height or [2Point/Axis/Top] <xx.xx>** - **(type a value or choose an option)**
2Point	Defines a circular base through two points that indicate the diameter).Prompts you: 1. **First end of diameter** - **(specify a point)** 2. **Second end of diameter** - **(specify a point)** 3. **Specify height or [2Point/Axis/Top] <xx.xx>**- **(type a value or choose an option)**
TTR(Tangent Tangent Radius)	Defines the diameter of the circular base as tangent to two entities and a radius.Prompts you: 1. **Specify point on object for first tangent** - **(select a point on a circle, arc, line or polyline segment)**; theTangent (6.29.9) entity snap turns on automatically. 2. **Specify point on object for second tangent** - **(select a point on a circle, arc, line or polyline segment)**; the Tangent entity snap turns on automatically. 3. **Radius of circle** - **(type a value or specify two points)** 4. **Specify height or [2Point/Axis/Top] <xx.xx>** - **(type a value, or else choose an option)**
Elliptical	Defines an base that is elliptical.Prompts you: 1. **Set first end of ellipse or [Center]** - **(specify the first end of the major axis of the ellipse.)** 2. **Second end of ellipse axis** - **(specify the second end of the major axis of the ellipse.)** 3. **Other end of ellipse axis** - **(specify the endpoint of the minor axis of the ellipse.)** 4. **Specify height or [2Point/Axis/Top] <xx.xx>** - **(type a value or choose an option.)**

Height options	Description

2Point	Specifies the height of the cone as the distance between any two points; prompts you: 1. **Specify first point** - **(pick a point or else enter x,y,z coordinates)** 2. **Specify second point** - **(pick a point or else enter x,y,z coordinates)**
Axis	Specifies the height of the cone by picking a point for the cone's tip, rotating the cone as necessary.Prompts you: **Specify axis endpoint** - **(pick a point or else enter x,y,z coordinates)**
Top	Create a truncated cone, where the top is flat.Prompts you: 1. **Radius of top of cone <0>** - **(specify the radius of the top face of the truncated cone)** 2. **Specify height or [2Point/Axis/Top] <xx.xx>** - **(press Enter to accept the default value, type a value, or pick a point, or enter an option)**

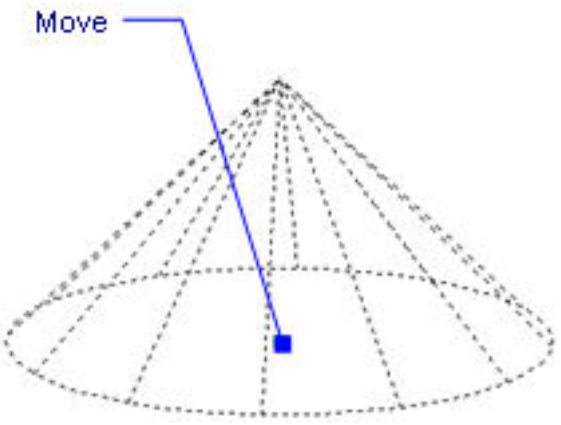
Grips Editing

Cone can be edited directly through grips:

1. Select the Cone. Notice that it has one grip.
2. Drag the grip to move the cone.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image433.jpg})

Figure 6.638



Related Commands

Box (6.6.17) - creates 3D boxes as solid models. Cylinder (6.9.60) - creates 3D cylinders as solid models. Dish (6.10.30) - creates the lower half of 3D balls as solid models. Dome (6.10.34) - creates the upper half of 3D balls as solid models. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Pyramid (6.24.58) - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic) Revolve (6.26.30) - revolves 2D entities into 3D solid models. Section (6.27.11) - creates 2D regions from 3D models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Sphere (6.27.64) - creates 3D balls as solid models. Subtract (6.27.56) - removes one solid model from a second. Torus (6.29.22) - creates 3D donuts as solid models. Union (6.30.6) - joins two or more solid models into a single entity. Wedge (6.33.7) - creates 3D wedges as solid models.

6.9.64 ChapooAccount

Obsolete. Replaced by the the CloudAccount (6.9.26) command .

6.10 D

6.10.1 DataExtraction

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Exports entity properties, block attributes, and drawing information to an external data file.

How to extract data in BricsCAD BIM Click here (2.11.4) to watch

Figure 6.639



Accessing the Command


command bar: **dataextraction** menu bar: **Tools | Data Extraction : dataextraction** Displays a 4 page dialog box.

Dialog Box Options

Option	Description

Select (Page 1 of 4)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/Da
Create data extraction definition:Creates a new .dxd (Data eXtraction Definition) file:

1. Turn on the option, then click the **Browse** button* (...).
2. In the file dialog box, select a folder, and then enter a name in the **File name** field.
3. Click the **Save** button.

Based on an existing data extraction definition:Uses an existing .dxd file as the template for the new one.

1. Turn on the option, and then click the **Browse** button* (...).
2. In the filedialog box, select a .dxd file.
3. Click the **Open** button.

* By default .dxd files are saved in the **...\Bim\Schedules** subfolder of the **Support (6.27.58)** folder. Newly created custom *.dxd files can be used to create schedules in the Project Browser (2.6.9). .dxd files can be edited in a text editor such as MS Notepad. **Select entities to extract data from:**

-  - dismisses the dialog box temporarily so that you can select entities in the drawing; enter Ctrl+A to select all entities in the drawing.Press Enter or right click to stop selecting entities.
- **Add Folder** - adds a folder of drawing from which to extract data Double click to open the **Folder Options** dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/
  - **Folder**: Click te **Browse** (...) button, then select a folder in **Select Folder** dialog box.
  - (option) Check the **Include subfolders** option
  - (option) Check the **Use wildcard** option, then type a text string in the filter field. E.g. House*.dwg includes all files starting with 'House'.
- **Add Drawing** - adds a drawingfrom which to extract data
- **Add Sheet Set** -adds a sheet set which to extract data

Extract entities from blocks: Includes entities in selected blocks. **Extract entities from Xrefs**: Includes entities in externally referenced drawings. **Add Xrefs to block counts**: Counts selected xrefs as blocks.

Data Extraction - Entities (Page 2 of 4)

Click **Next** for the next page: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/Da

Controls which entity types are included () or not () in the data extraction procedure. Select multiple items:

- To select more than one item in a row: select an item, and then hold down the **Shift**key while click additional items
- To select multiple items: hold down the **Ctrl** key while selecting multiple entities.
- To unselect items, click them again while holding down the **Ctrl** key.

To sort the list, click the title in the column heading. Click again to sort in reverse order.

- **Entity** - lists the entity types and blocks in the selection, according to the options set in the Data Extractiondialog.
- **Name** - entity names. Accept the default names or type a customized name. The entity type name displays in the default **Name** field in the data extraction export.
- **Type** - reports the type of the selected entity: Non-Block, Block or Block with attributes.

Select All: Selects all items for data extraction. **Select Blocks**: Selects only blocks in the list, because they usually contain attributes. **Back**: Returns back to the **Data Extraction - Select** page.

Data Extraction - Properties (Page 3 of 4)

Click **Next** for the next page: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/Da
Select the properties to be included in the data extraction file: click the checkbox to check/uncheck a property, which indicates whether it is selected () or not (). Use the same techniques to select more than one item as described in **Data Extraction - Entities** page. To sort the list, click the title in the column heading. Click again to sort in reverse order.

- **Property** - lists of all properties of the selected entities.
- **Name** - lists all names of the selected entities.Accept the default names or type a customized name. The property names display in the column headingsin the data extraction export.
- **Category** - reports the category of the selected entities.
- **Additional Format** - Formats the item in the export file. Click to display the dialog box. The layout of the dialog box depends on the selected item, justlike the Format section of the Field dialog (1.21.23).
- **Select Attributes** - select only items that contain attributes
- **Show Checked Only** - show only selected items, hiding those that are not selected.
- **Show All** - shows all items, whether checked or not.
- **Move Selected Up** - move the selected item(s) up the list.
- **Move Selected Down** -move the selected item(s) down the list.
- **Combine Equal Rows** - combine rows that contain the same entities.
- **Include Name Column** - toggles the display of the Name column in the exported data.

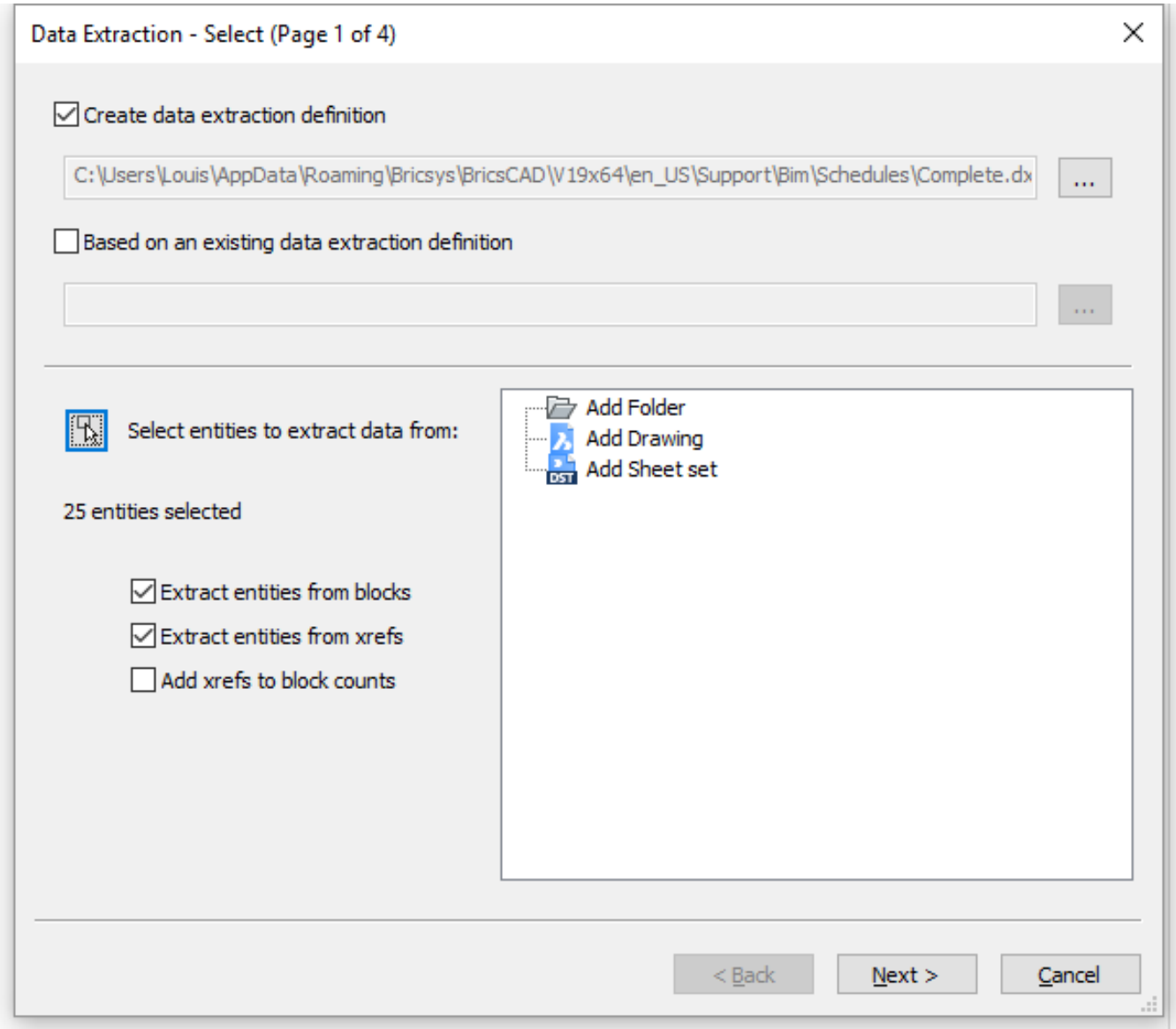
Back: Switches back to theData Extraction - Entitiespage.

Data Extraction - Output (Page 4 of 4)	Click **Next** for the next page: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/Da **Output to CSV**: Exports the data in a delimiter separated format (.csv): Defines the format and name of the exported data file: • **Delimiter** - specifies the separator between fields of data: – **Semicolon** - separates fields with **;** – **Comma*** - separates fields with **,** – **Space** - separates fields with spaces – **Tab** - separates fields with tabs *Multiply composition names contain a comma, therefore do not select comma as the delimiter character in case multiply compositions are processed. • Click the **Browse** button (...), give the data file a name, and then click **Save**. If you select an existing .csv file, you will be prompted to overwrite it. **Create Data**: Exports the data in a Table (6.29.1): • **Create table**: Creates a static table. • **Create table with datalink**: Creates a dynamic table. Executing the DataLinkUpdate (6.10.3) command synchronizes the data in table with the model. Save the model first. You will be prompted to save the *.dxd definition file with the table. **Table Style**: • Click the list button and select a table style. • Click the **Table Style** icon (); edit an existing table style or create a new one (see the TableStyle (6.29.6) command). • **Table formatting**: Type a title in the **Title** field.
Finish	Depending on the selected output: • Extracts the data and saves them to the .csv file. The file can be opened in a spreadsheet or database program. • Prompts you to specify an insertion point for the table(s).

Procedure

Editing Data Extraction Definition (.dxd) files (2.12.2)

Figure 6.640



Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttDisp (6.4.35) - toggles the display of attribute text. AttEdit (6.4.36) - edits attribute values and properties. -AttExt (6.4.29) - extracts attributes from drawings to data files through the command bar. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. Datalink (6.10.2) - links data from an Excel spreadsheet directly into drawing tables for bi-directional updating. DatalinkUpdate (6.10.3) - synchronizes the linked data in tables in the current drawing with data in the linked source file. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. Properties (6.24.49) - edits attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place. Table (6.29.1) -creates a table entity in the drawing through a dialog box, either from scratch or from data in an external file or a data link. TableStyle (6.29.6)- creates and edits table styles through the Drawing Explorer.

Figure 6.641

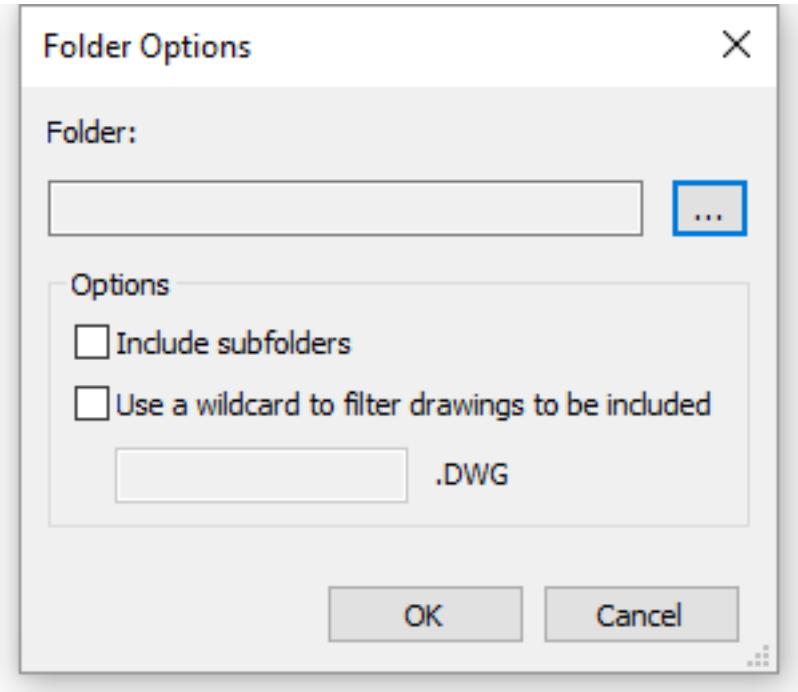


Figure 6.642

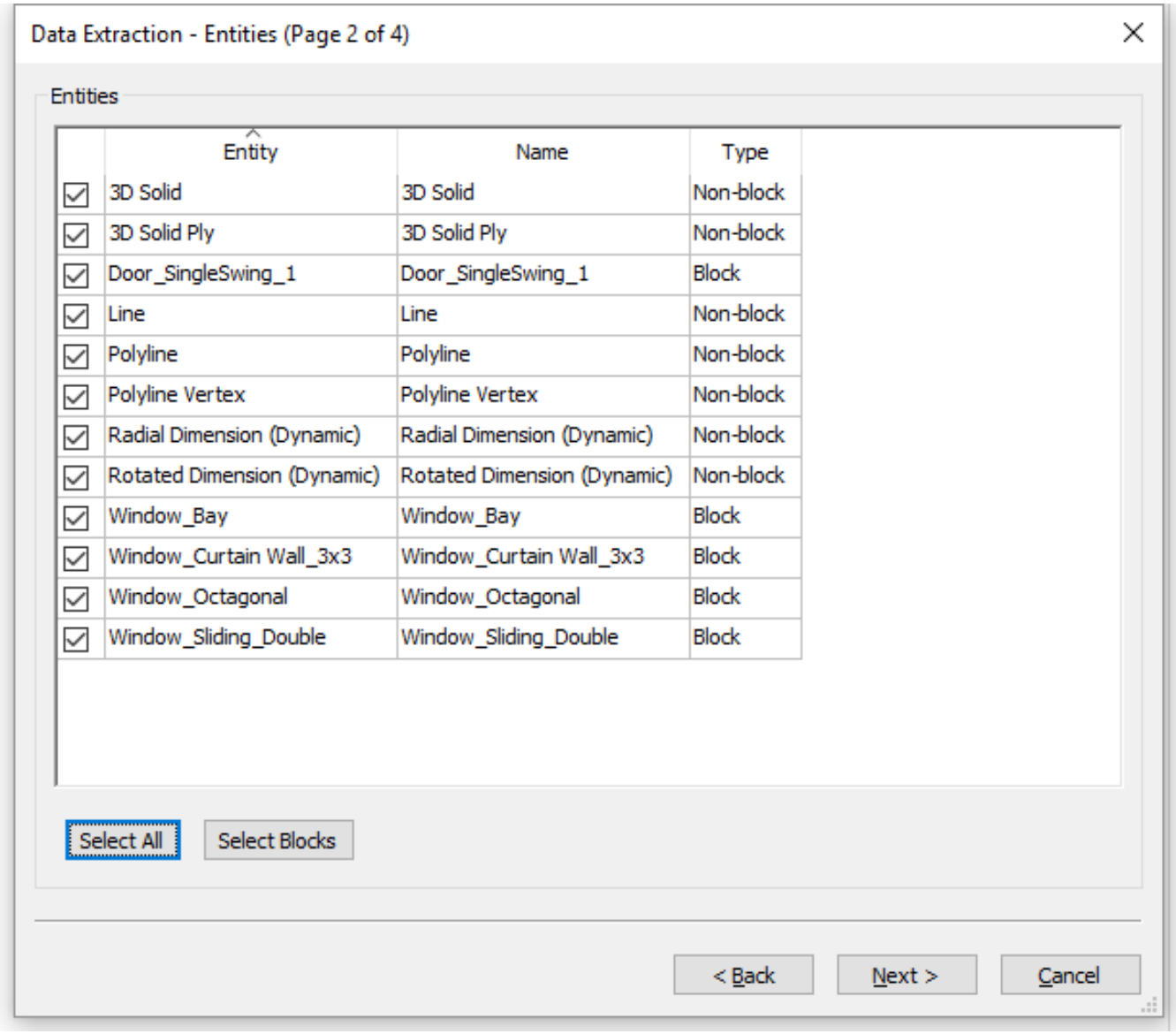


Figure 6.643

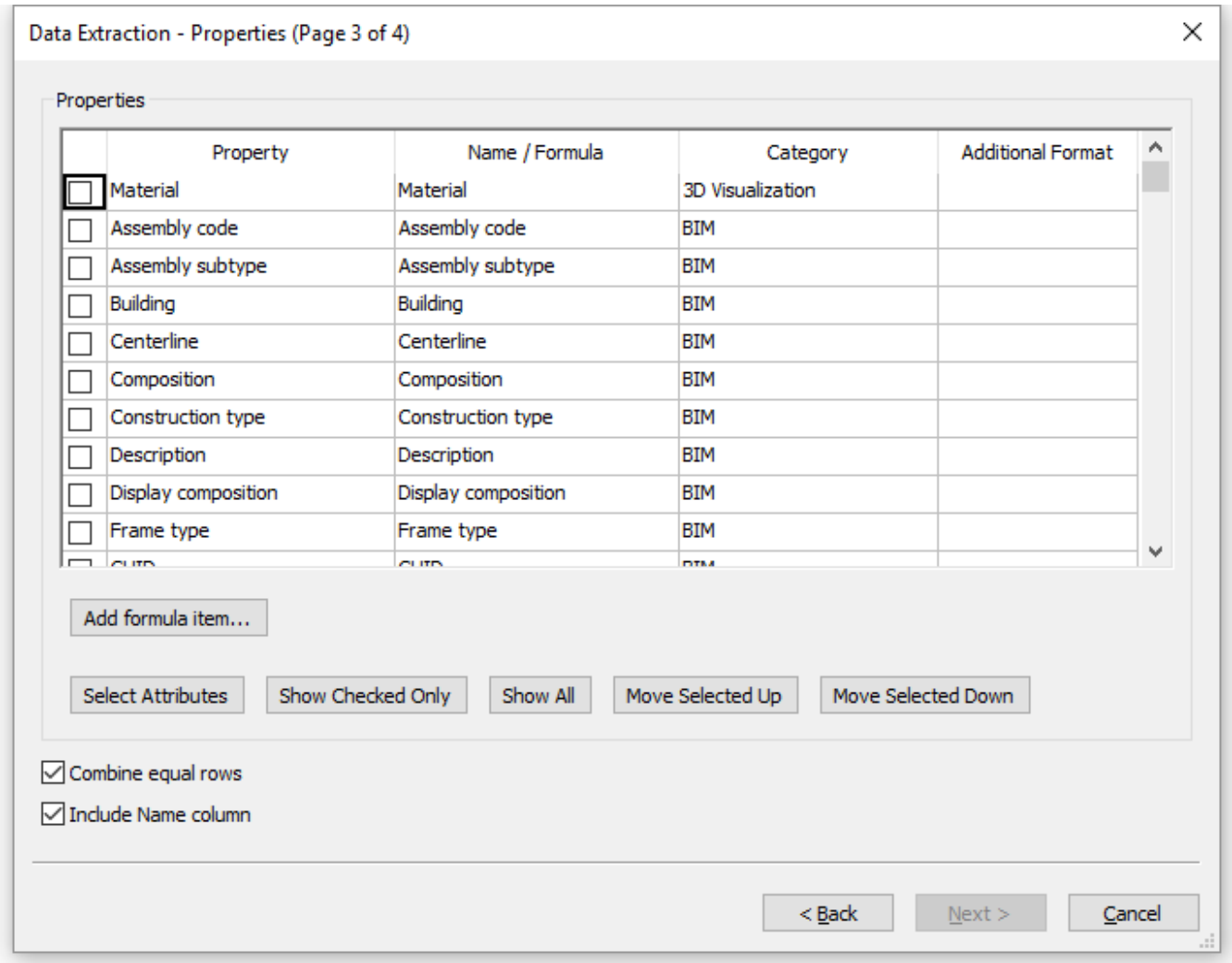

Data Extraction - Properties (Page 3 of 4)
Properties
Property
Name / Formula
Category
Additional Format
Material
Material
3D Visualization
Assembly code
Assembly code
BIM
Assembly subtype
Assembly subtype
BIM
Building
Building
BIM
Centerline
Centerline
BIM
Composition
Composition
BIM
Construction type
Construction type
BIM
Description
Description
BIM
Display composition
Display composition
BIM
Frame type
Frame type
BIM
Add formula item...
Select Attributes
Show Checked Only
Show All
Move Selected Up
Move Selected Down
Combine equal rows
Include Name column
< Back
Next >
Cancel

Figure 6.644

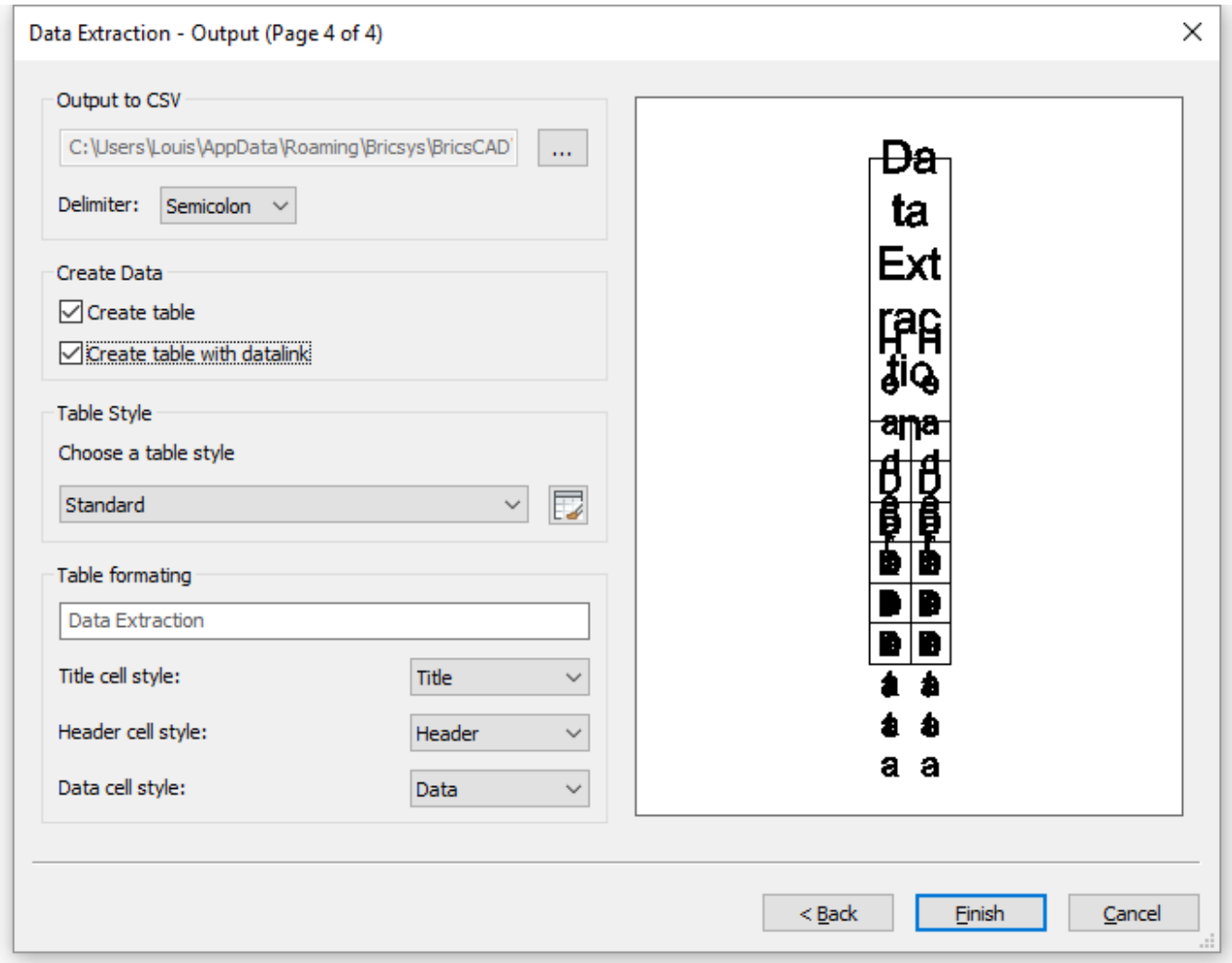

Data Extraction - Output (Page 4 of 4)
Output to CSV
C:\Users\Louis\AppData\Roaming\Bricsys\BricsCAD
...
Delimiter:
Semicolon
Create Data
Create table
Create table with datalink
Table Style
Choose a table style
Standard
Table formating
Data Extraction
Title cell style:
Title
Header cell style:
Header
Data cell style:
Data
< Back
Finish
Cancel

Figure 6.645



6.10.2 DataLink

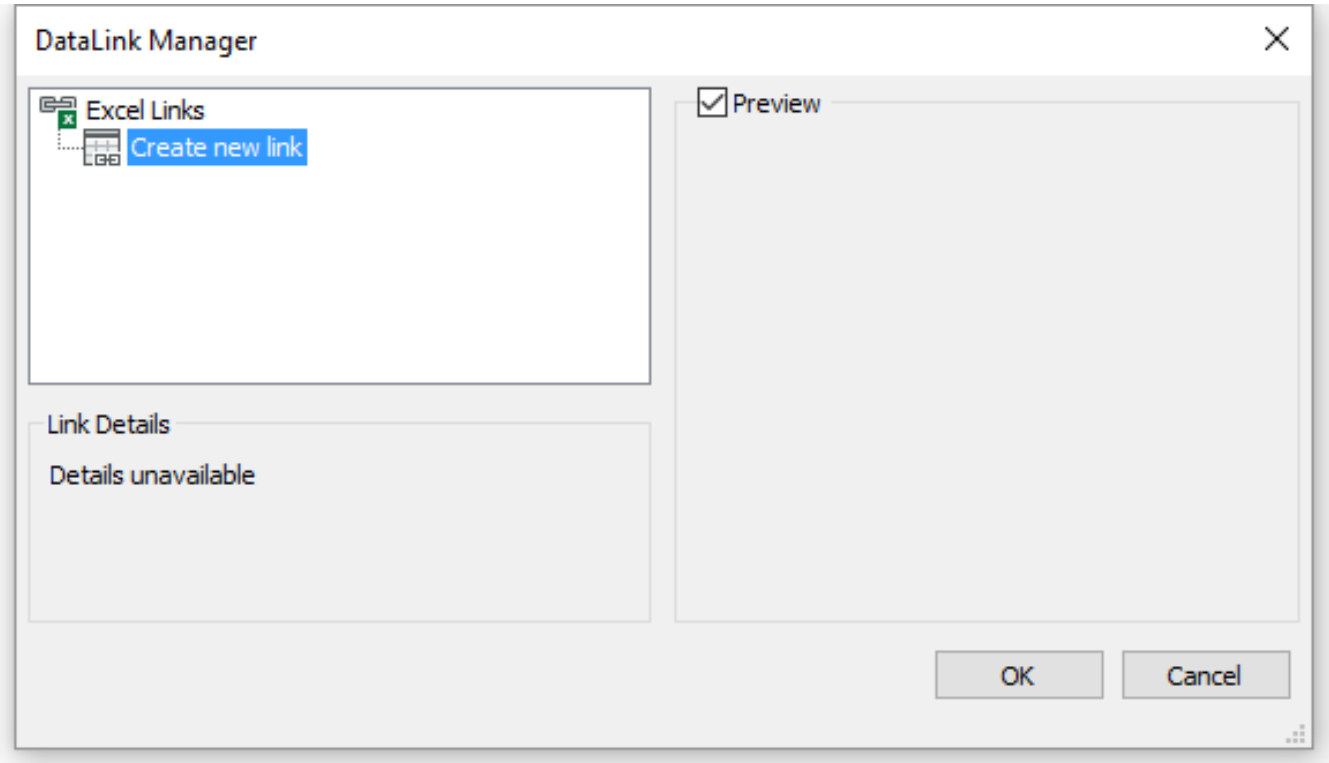
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Links data from an Excel spreadsheet directly into drawing tables for bi-directional updating.

Accessing the Command



Keyboard: **datalink** Menu: Toolbar: Quad Tab: Ribbon Tab | Panel: **:datalink** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/DatalinkManager1.png})

Figure 6.646



Command Options

Option	Description



Creates a new link. Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/Da
Do the following:

1. Click the **Browse** button (), then select a data file. *.xls, *.xlsx and *.csv files are accepted. If selected, a **Preview** of the data file displays.
2. Type a name in the **DataLink Name** field.
3. Choose a **Path type** option. The options are:
   - **Full Path**: BricsCAD searches for the data file in the specified folder only.
   - **Relative Path**: The data file is searched for in a subfolder of the drawing file folder.
   - **No Path**: The data file is searched for in the drawing file folder only.
4. Define the link options. Select a sheet or accept the default (= active sheet).
   - **Link to the entire sheet**: all cells that contain data are linked.
   - **Linked to named range**: links to a named range of cells. Click the button to select a named range.
   - **Link to range**: specify a range of cells. Click the **Preview** button to display the selected range.
5. Choose a cell contents option.
   - **Keep data formats and formulas**:Imports data with formulas and supported data formats.
   - **Keep data formats, solve formulas in Excel**:Imports data formats. Data is calculated from formulas in Excel.
   - **Convert to text, solve formulas in Excel**:Imports data as text with data calculated from formulas in Excel.
   - **Allow writing to Excel**:If checked, the DATALINKUPDATE command uploads changes to linked data in your drawing to the original external spreadsheet.
6. Specify whether Excel cell formatting is used.
   - **Keep table updated to Excel formatting**: Updates any changed formatting when the DATALINKUPDATE command is executed.
   - **Start with Excel formatting. Do not update**:Imports the formatting specified in the original data file. The DATALINKUPDATE command does not affect any changes made to the cell formatting.

Edit Link	Select a data link, then right click and choose **Edit** in the context menu. A dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/Da The following properties are editable: • DataLink Name • Cell Contents • Cell Formatting
Delete Link	Select a data link, then right click and choose **Delete**in the context menu. The link is deleted without warning.

Related Commands

DataLinkUpdate (6.10.3) - Synchronizes the linked data in tables in the current drawing with data in the linked source file. Table (6.29.1) -Creates a table entity in the drawing through a dialog box.

Figure 6.647

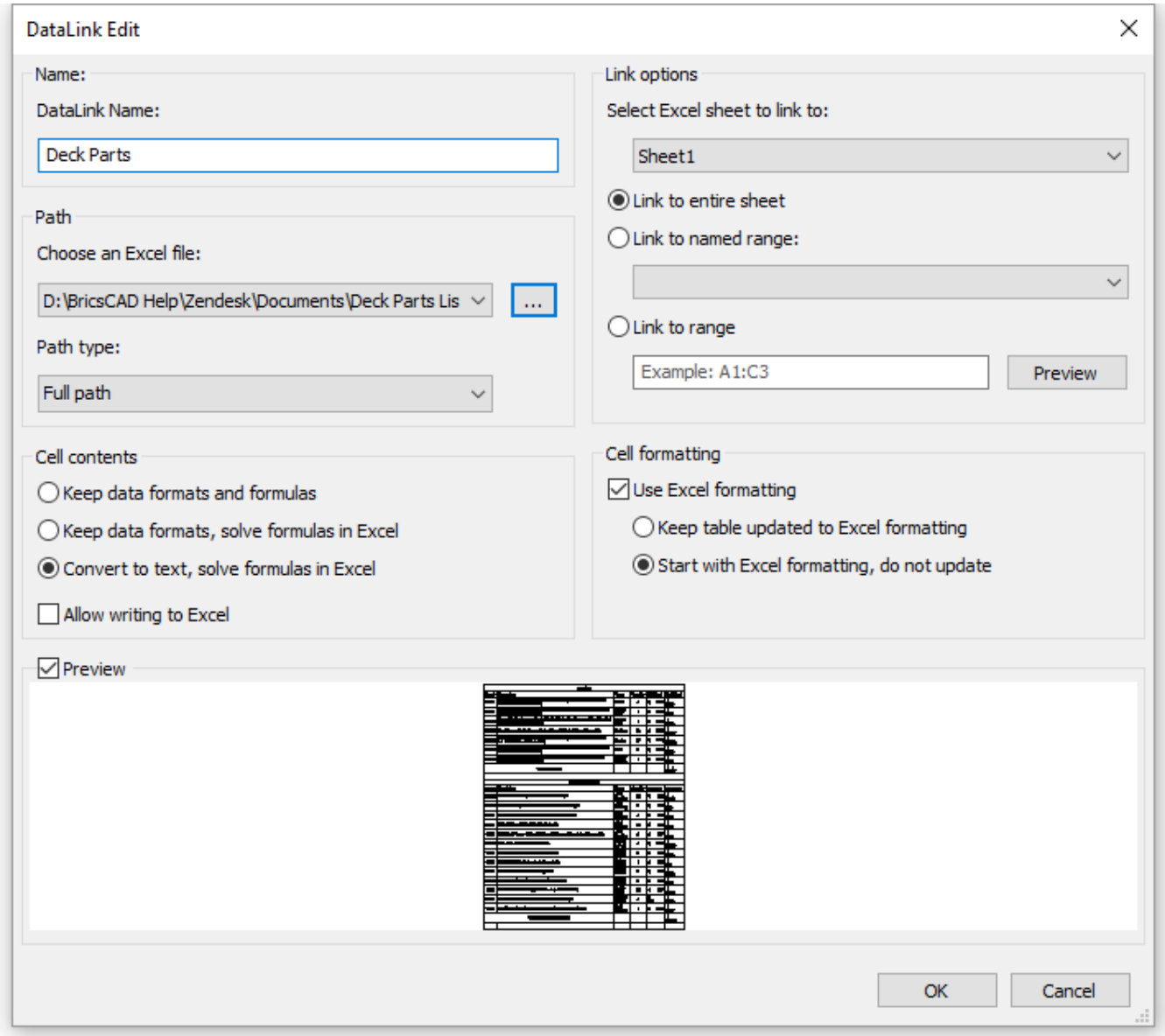

DataLink Edit
Name:
DataLink Name:
Deck Parts
Path
Choose an Excel file:
D:\BricsCAD Help\Zendesk\Documents\Deck Parts Lis
...
Path type:
Full path
Link options
Select Excel sheet to link to:
Sheet1
Link to entire sheet
Link to named range:
Link to range
Example: A1:C3
Preview
Cell contents
Keep data formats and formulas
Keep data formats, solve formulas in Excel
Convert to text, solve formulas in Excel
Allow writing to Excel
Cell formatting
Use Excel formatting
Keep table updated to Excel formatting
Start with Excel formatting, do not update
Preview
OK
Cancel

Figure 6.648

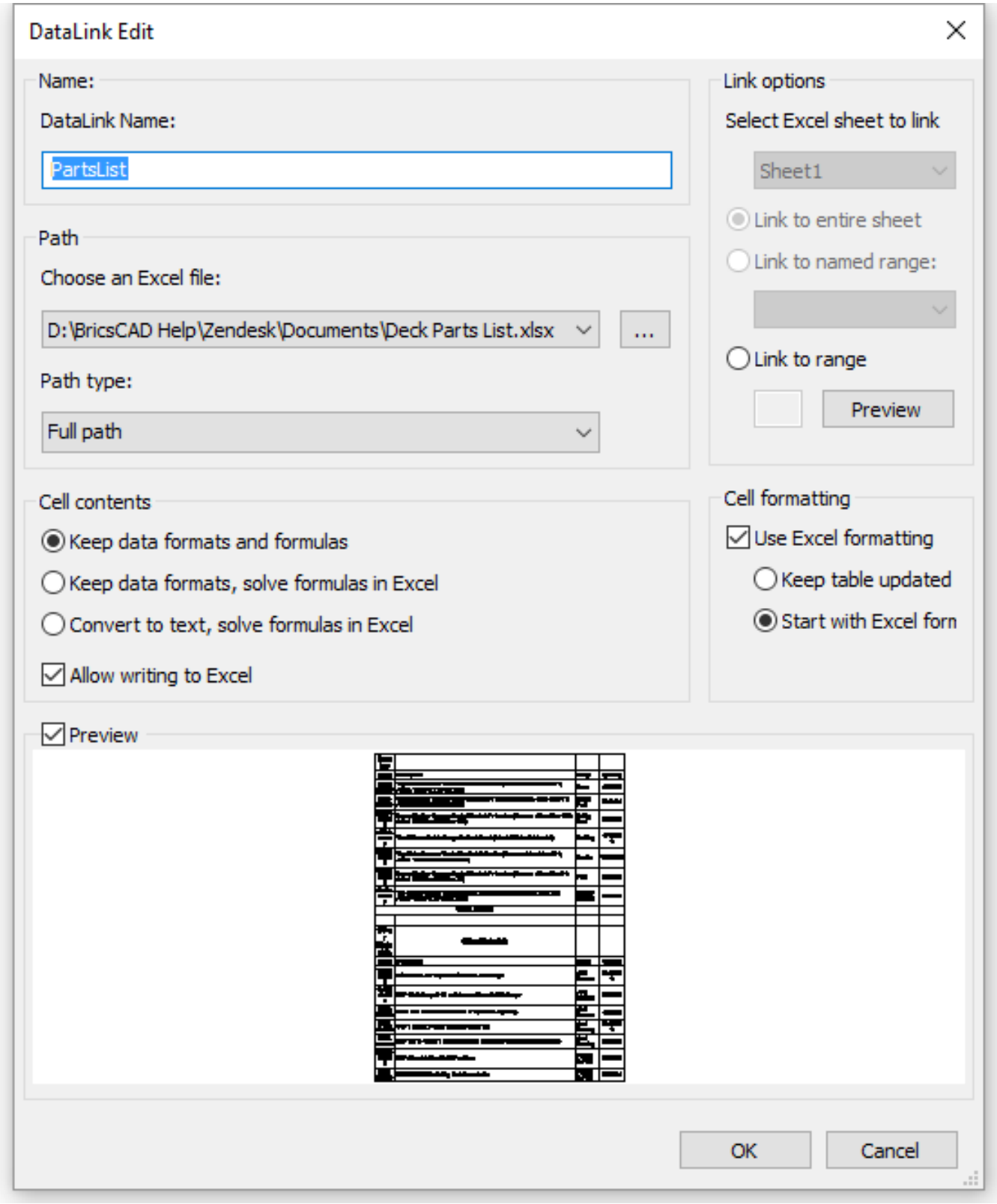

DataLink Edit
Name:
DataLink Name:
PartsList
Path
Choose an Excel file:
D:\BricsCAD Help\Zendesk\Documents\Deck Parts List.xlsx
...
Path type:
Full path
Link options
Select Excel sheet to link
Sheet1
Link to entire sheet
Link to named range:
Link to range
Preview
Cell contents
Keep data formats and formulas
Keep data formats, solve formulas in Excel
Convert to text, solve formulas in Excel
Allow writing to Excel
Cell formatting
Use Excel formatting
Keep table updated
Start with Excel forn
Preview
OK
Cancel

Figure 6.649



6.10.3 DataLinkUpdate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Synchronizes the linked data in tables in the current drawing with data in the linked source file.

Accessing the Command

Keyboard: **datalinkupdate :datalinkupdate You are prompted:** Select an option[Update data link / Write data link] <Update data link>: **(Choose an option.)** Select objects or [Update all]: **(Select a table entity.)** Entities in set: 1 Select objects or [Update all]: **(Select more table entities or pres Enter.)**

Command Options

Option	Description
Update data link	Updates the linked data in a table in the drawing with data that has been changed in the external source file.
Write data link	Updates the linked data in an external file with data that has been changed in a table in the drawing.
Select objects	Prompts you to select table entities
Update all	Synchronizes all linked data in all tables in the drawing.

Related Commands

DataLink (6.10.2) - Links data from an Excel spreadsheet directly into drawing tables for bi-directional updating.

Figure 6.650



6.10.4 DbList

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Lists information about all entities in the drawing (short for ”database listing”).

Accessing the Command

command bar: **dblist** : **dblist** Displays data about every entity in the command bar, such as the following: ---------- Line ---------- Handle: 7A Current space: Model Layer: 0 Color: BYLAYER Linetype: ByLayer From point: X= 8 9/16” Y= 4 5/16” Z= 0” To point: X= 9 15/16” Y= 3 13/16” Z= 0” Length: 1 7/16” Angle in UCS XY plane: 339 Delta values: X= 1 5/16” Y= -1/2” Z= 0”

Command Options

Press **F2** to see the data listed in the Prompt History window. For drawings with many entities, the list can take a long time to finish. Press **Esc** to stop the command.

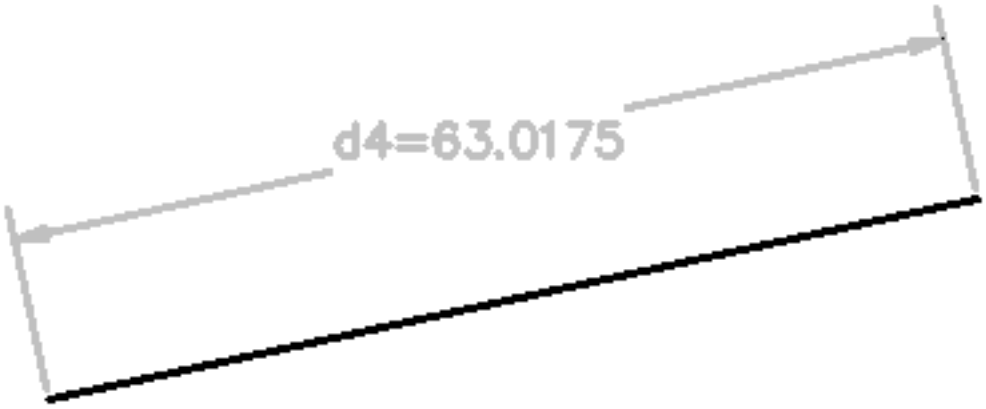
Related Commands

List (6.20.35) - lists the properties of individual entities in the command bar.. Properties (6.24.49) - lists properties of entities in a palette. Status (6.27.46) - reports the status of the drawing.

Figure 6.651



Figure 6.652



6.10.5 DcAligned

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains the distance between two points, or the distance between a point and an entity, or the length of a line or polyline segment. (This command is equivalent to the Aligned (6.10.29) option of the DIMCONSTRAINT (6.10.29) command.) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/eg.png})

Accessing the Command

 command bar: **dcaligned** menu bar: **Parametric | 2D Constraints | Aligned** toolbar: **2D Constraints |**  **ribbon:** : **dcaligned Prompts you in the command bar:** Specify first constraint point or [Entity/Point/2Lines] <Entity>: **(Select a valid constraint point (6.15.16) or choose an option)** Specify second constraint point: **(Select a valid constraint point (6.15.16).)** Specify dimension line location: **(Select a point to position the dimension line.)** Dimension text <xx.xx>: **(Type a value or press Enter to accept the current value.)** The distance between the two points is constrained, and can be edited directly or with the Parameters panel.

Command Options

Option	Description
Entity	Constraints the length of a line, a polyline segment or the chord of an arc.Prompts you: 1. **Select an entity** - **(Select a line, polyline segment, or an arc)** 2. **Specify dimension location** - **(pick a point)** 3. **Dimension text <xx.xx>** - **(type a value, or else press Enter to accept the current value)** The length of the entity is now constrained, although it can still be moved, rotated, and so on.

Point	Constraints the perpendicular distance between a point and a line or polyline segment.Prompts you: 1. **Specify constraint point or [Line] <Line>** - **(pick a valid constraint point (6.15.16), orelsefirst select the line, polyline segment, or arc, and then select the point)** 2. **Select constraint point**- **(pick a valid constraint point)** 3. **Specify dimension location** - **(pick a point)** 4. **Dimension text <xx.xx>** - **(type a value, or press else Enter to accept the current value)** The distance between the point and the entity is now constrained, and can be edited directly or with the Parameters panel.
2Lines	Constraints the distance between two straight lines and/or straight polyline segments. If the entities are not parallel already, the second entity is rotated around its midpoint to be parallel to the first entity.Prompts you: 1. **Select first line** - **(select a line or polyline segment.)** 2. **Select second line to make parallel** - **(select a line or polyline segment)** 3. **Specify dimension location** - **(pick a point)** 4. **Dimension text <xx.xx>** - **(type a value or press Enter to accept the current value)** The shortcut for this option is 2L. The distance between the two enties is now constrained, and can be edited directly or with the Parameters panel.

Procedures

- To edit dimensional constraints (1.15.6)
- Using expressions to define dimensional constraints (1.15.6)

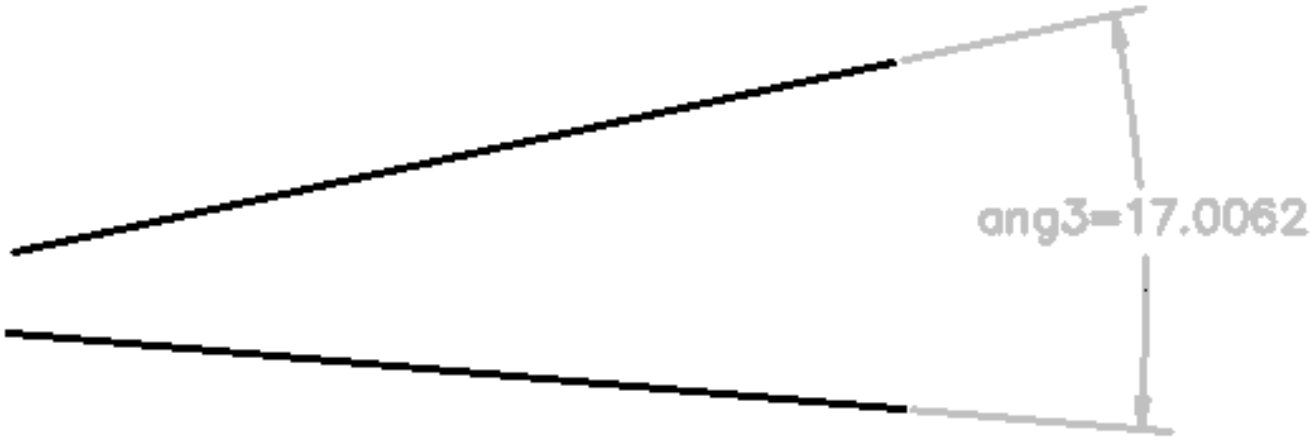
Related Commands

Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. Geomconstraint (6.15.16) - applies geometric relationships between entities and/or valid constraint points on entities. DcAngular (6.10.6) - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. DcConvert (6.10.7) - converts an associative dimension to a dimensional constraint. DcDiameter (6.10.8) - constrains the diameter of a circle or an arc. DcHorizontal (6.10.10) - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. DcLinear (6.10.39) - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. DcRadius (6.10.11) - constrains the radius of a circle or an arc. DcVertical (6.10.12) - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. DcDisplay (6.10.9) - allows to set dimensional constraints to be displayed or hidden. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity. CleanUnusedVariables (6.9.23) - purges variables not used in constraint expressions and not linked to dimensions. -Parameters (6.24.5) - manages 2D dimensional constraint parameters and user variables in the current drawing.

Figure 6.653



Figure 6.654



6.10.6 DcAngular

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains the angle between two lines or straight polyline segments; also constrains the angle of an arc or polyline arc.(This command is equivalent to the Angular (6.10.29) option of the DIMCONSTRAINT (6.10.29) command.) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dc

Accessing the Command

 command bar: **dcangular** menu bar: **Parametric | 2D Constraints | Angular** ribbon:**Parametric | 2D Constraints | Angular** toolbar: **2D Constraints |** : **dcangular Prompts you in the command bar:** Select first line or arc or [3Point] <3Point>: **(Select a line or straight polyline segment, or an arc or polyline arc segment, or else press Enter to select the 3Point option)** The angle of the entity is now constrained, although the entity can still be moved, rotated, and so on.

Command Options

Option	Description
Select first line	When you select a line or straight polyline segment, prompts you: 1. **Select second line- (select another line or straight polyline segment)** 2. **Specify dimension line location- (pick a point to position the dimension line)** 3. **Dimension text <xx.xx>- (type a value, or elkse press Enter to accept the current value)**

Select arc	When you select an arc or a polyline arc segment, prompts you 1. **Specify dimension line location- (select a point to position the dimension line)** 2. **Dimension text <xx.xx>- (type the dimension text, or else press Enter to accept the current dimension text)**
3Point	Creates an angular constraint between three valid constraint (6.15.16) points.Prompts you: 1. **Specify angle vertex** - **(pick a point)** 2. **Specify first angle constraint point** - **(pick a point)** 3. **Specify second angle constraint point** - **(pick a point)** The abbreviation for this option is 3P.

Procedures

- To edit dimensional constraints (1.15.6)
- Using expressions to define dimensional constraints (1.15.6)

Related Commands

DimConstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GeomConstraint (6.15.16) - applies geometric relationships between entities and/or valid constraint points on entities. DcAligned (6.10.5) - constrains the distance between two points. DcConvert (6.10.7) - converts an associative dimension to a dimensional constraint. DcDiameter (6.10.8) - constrains the diameter of a circle or an arc. DcHorizontal (6.10.10) - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. DcLinear (6.10.39) - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. DcRadius (6.10.11) - constrains the radius of a circle or an arc. DcVertical (6.10.12) - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. DcDisplay (6.10.9) - allows to set dimensional constraints to be displayed or hidden. DelConstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity. CleanUnusedVariables (6.9.23) - purges variables not used in constraint expressions and not linked to dimensions. -Parameters (6.24.5) - manages 2D dimensional constraint parameters and user variables in the current drawing.

Figure 6.655



6.10.7 DcConvert

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Converts an associative dimension to a dimensional constraint.

Accessing the Command

 command bar: **dcconvert** menu bar: **Parametric | 2D Constraints | Convert** toolbar: **2D Constraints |** ribbon:**Parametric | 2D Constraints | Convert** : **dcconvert Prompts you in the command bar:** Select associative dimension to convert: **(Select the associative dimension you want to convert to a dimensional constraint)**

Command Options

Option	Description
Select associative dimension to convert	Select an associative dimension; the program converts it a a dimensional constraint.

Procedures

- To edit dimensional constraints (1.15.6)
- Using expressions to define dimensional constraints (1.15.6)

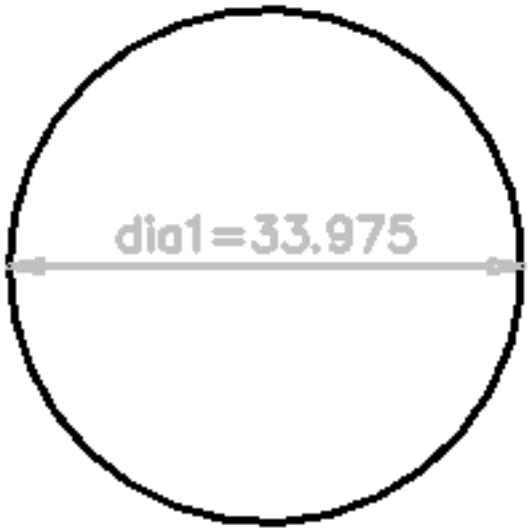
Related Commands

DimConstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GeomConstraint (6.15.16) - applies geometric relationships between entities and/or valid constraint points on entities. DcAligned (6.10.5) - constrains the distance between two points. DcAngular (6.10.6) - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. DcDiameter (6.10.8) - constrains the diameter of a circle or an arc. DcHorizontal (6.10.10) - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. DcLinear (6.10.39) - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. DcRadius (6.10.11) - constrains the radius of a circle or an arc. DcVertical (6.10.12) - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. DcDisplay (6.10.9) - allows to set dimensional constraints to be displayed or hidden. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity. CleanUnusedVariables (6.9.23) - purges variables not used in constraint expressions and not linked to dimensions. -Parameters (6.24.5) - manages 2D dimensional constraint parameters and user variables in the current drawing.

Figure 6.656



Figure 6.657



6.10.8 DcDiameter

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains the diameter of a circle, an arc or an polyline arcsegment. (This command is equivalent to the Diameter (6.10.29) option of the DIMCONSTRAINT (6.10.29) command.) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dcdiamater-eg.png})

Accessing the Command

 command bar: **dcdiameter** menu bar: **Parametric | 2D Constraints | Diameter** ribbon: **Parametric | 2D Constraints | Radius | Diameter** toolbar: **2D Constraints |** : **dcdiameter Prompts you in the command bar:** Select arc, circle or curved polyline segment: **(Select a circle, arc or polylinearcsegment)** Specify dimension location: **(Select a point to position the dimension line)** Dimension text <xx.xx>: **(Type a value, or else press Enter to accept the current value)** The diameter of the entity is now constrained, although the entity can still be moved, rotated, and so on.

Command Options

Option	Description
Select arc, circle or curved polyline segment	Select one entity: • Circle • Arc • Polyline arc segment
Specify dimension location	Pick a point to position the dimension line
Dimension text	Enter an option: • Type a value • Press Enter to accept the current value

Procedures

- How to edit dimensional constraints (1.15.6)
- Using expressions to define dimensional constraints (1.15.6)

Figure 6.658



6.10.9 DcDisplay

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the display of dimensional constraints between visible and hidden. (Dimensional constraints are always hidden when the drawing is first opened.)

Accessing the Command



command bar: **dcdisplay** menu bar: **Parametric | Show/Hide Dimensional Constraint** toolbar: **Dimensional Constraint** | ribbon:**Parametric | 2D Constraints | Show/Hide** : **dcdisplay Prompts you in the command bar:** Select entities: **(Select one or more entities.)** Entities in set: 1 Select entities: **(Select more entities, or else press Enter to stop selecting)** Enter an option [Show/Hide]<Show>: **(Enter an option)**

Command Options

Option	Description
Show	Dimensional constraints on the selected entities become visible.
Hide	Dimensional constraints on the selected entities no longer display.

Procedures

- To set the display of dimensional constraints of a selection of entities (1.15.6)
- To set the visibility of the dimensional constraints of all entities (1.15.6)

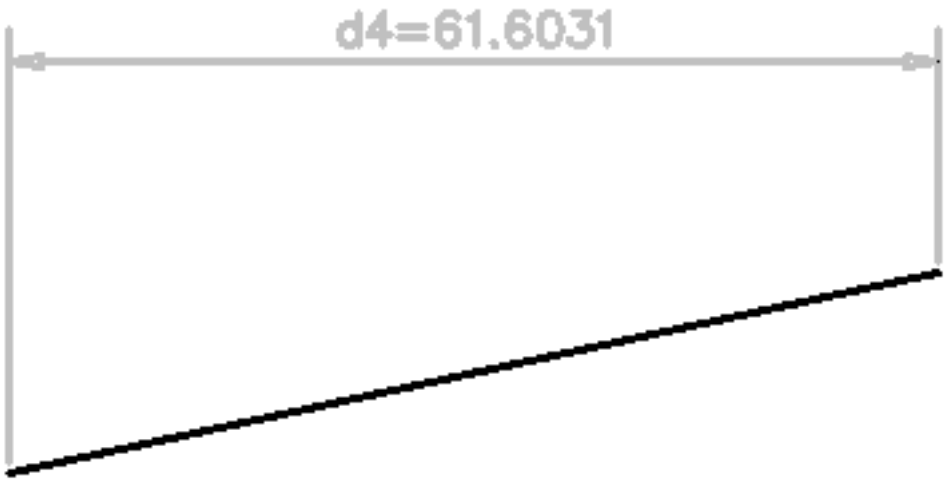
Related Commands

ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.659



Figure 6.660



6.10.10 DcHorizontal

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains the distance between two points, or the length of a single entity, to the horizontal (X direction) in the current coordinate system. (This command is equivalent to the Horizontal (6.10.29) option of the DIMCONSTRAINT (6.10.29) command.) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/eg.png})

Accessing the Command


command bar: **dchorizontal** menu bar: **Parametric | 2D Constraints | Horizontal** toolbar: **2D Constraints**

| ribbon:**Parametric | 2D Constraints | Linear | Horizontal** : **dchorizontal Prompts you in the command bar:** Specify first constraint point or [Entity] <Entity>: **(Select a valid constraint point (6.15.16), or else press Enter to select an entity.)** Specify second constraint point: **(Select another valid constraint point (6.15.16))** Specify dimension location: **(Select a point to position the dimension line)** Dimension text <xx.xx>: **(Type a value, or else press Enter to accept the current value)** The horizontal distance (parallel to the X-axis of the current coordinate system) between the two points is constrained.

Command Options

Option	Description

Entity	Constrains the horizontal distance between the endpoints of an entity.Prompts you: 1. **Select an entity** - **(select a line, arc or polyline segment)** 2. **Specify dimension line location** - **(pick a point)** 3. **Dimension text <xx.xx>** - **(type a value or press Enter to accept the current value)** The entity is constrained horizontally (parallel to the X-axis of the current coordinate system), but can continue change its size in other directions.

Procedures

- To edit dimensional constraints (1.15.6)
- Using expressions to define dimensional constraints (1.15.6)

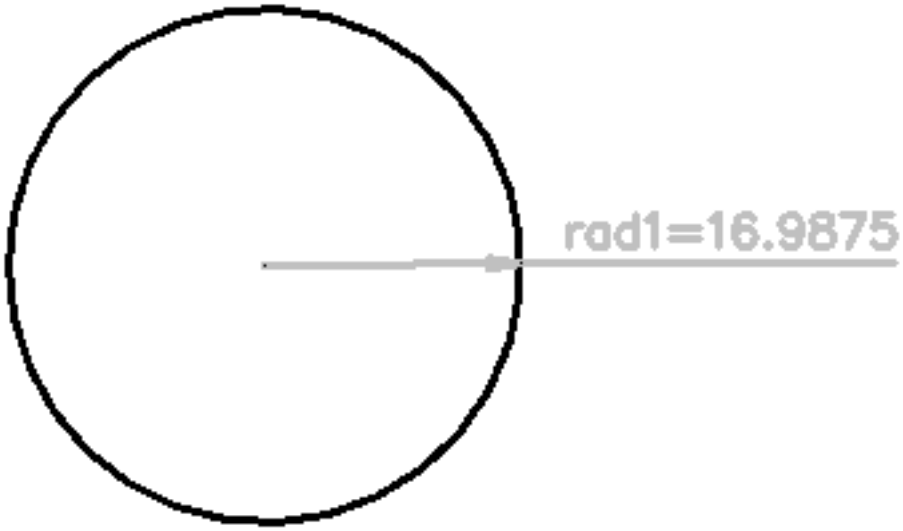
Related Commands

Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. Geomconstraint (6.15.16) - applies geometric relationships between entities and/or valid constraint points on entities. DcAligned (6.10.5) - constrains the distance between two points. DcAngular (6.10.6) - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. DcConvert (6.10.7) - converts an associative dimension to a dimensional constraint. DcDiameter (6.10.8) - constrains the diameter of a circle or an arc. DcLinear (6.10.39) - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. DcRadius (6.10.11) - constrains the radius of a circle or an arc. DcVertical (6.10.12) - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. DcDisplay (6.10.9) - allows to set dimensional constraints to be displayed or hidden. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity. CleanUnusedVariables (6.9.23) - purges variables not used in constraint expressions and not linked to dimensions. -Parameters (6.24.5) - manages 2D dimensional constraint parameters and user variables in the current drawing.

Figure 6.661



Figure 6.662



6.10.11 DcRadius

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains the the radius of a circle, an arc, or polyline arc segment. (This command is equivalent to the Radius (6.10.29) option of the DIMCONSTRAINT (6.10.29) command.) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dcradius-eg.png})

Accessing the Command

 command bar: **dcradius** menu bar: **Parametric | 2D Constraints | Radius** toolbar: **2D Constraints |**  ribbon:**Parametric | 2D Constraints | Radius** : **dcradius Prompts you in the command bar:** Select arc, circle or curved polyline segment: **(Select a circle, arc or polyline arc segment)** Specify dimension location: **(Select a point to position the dimension line)** Dimension text <xx.xx>: **(Type a value, or else press Enter to accept the current value)** The radius of the entity is now constrained, although the entity can still be moved, rotated, and so on.

Command Options

Option	Description
Select arc, circle or curved polyline segment:	Select one: • Circle • Arc • Polyline arc segment
Specify dimension location	Move the cursor, and then pick a point to position the dimension line
Dimension text <xx.xx>	Choose one: • Type a value • Press Enter to accept the current value

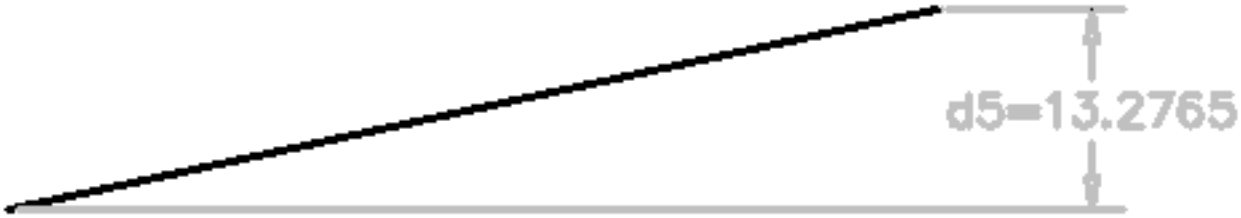
Procedures

- Using dimensional constraints (1.15.6)
- Using expressions to define dimensional constraints (1.15.6)

Figure 6.663



Figure 6.664



6.10.12 DcVertical

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains the distance between two points, or the length of a single entity, to the vertical (Y direction) in the current coordinate system. (This command is equivalent to the Vertical (6.10.29) option of the DIMCONSTRAINT (6.10.29) command.) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comm eg.png})

Accessing the Command

 command bar: **dcvertical** menu bar: **Parametric | 2D Constraints | Vertical** toolbar: **2D Constraints |**  ribbon: **Sketch |2D Constraints |**  : **dcvertical you are prompted:** Specify first constraint point or [Entity] <Entity>: **(Select a valid constraint point (6.15.16), or else press Enter to select an Entity)** Specify second constraint point: **(Select a valid constraint point (6.15.16))** Specify dimension location: **(Select a point to position the dimension line)** Dimension text <xx.xx>: **(Type a value, or else press Enter to accept the current value)** The vertical distance (parallel to the X-axis of the current coordinate system) between the two points is constrained.

Command Options

Option	Description
Entity	Constrains the vertical distance between the endpoints of an entity.Prompts you: 1. **Select an entity** - **(select a line, arc or polyline segment)** 2. **Specify dimension line location** - **(pick a point)** 3. **Dimension text <xx.xx>** - **(type a value, or else press Enter to accept the current value)** The entity is constrained vertically (parallel to the T-axis of the current coordinate system), but can continue change its size in other directions.

Procedures

- To edit dimensional constraints (1.15.6)
- Using expressions to define dimensional constraints (1.15.6)

Related Commands

Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. Geomconstraint (6.15.16) - applies geometric relationships between entities and/or valid constraint points on entities. DcAligned (6.10.5) - constrains the distance between two points. DcAngular (6.10.6) - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. DcConvert (6.10.7) - converts an

Figure 6.665



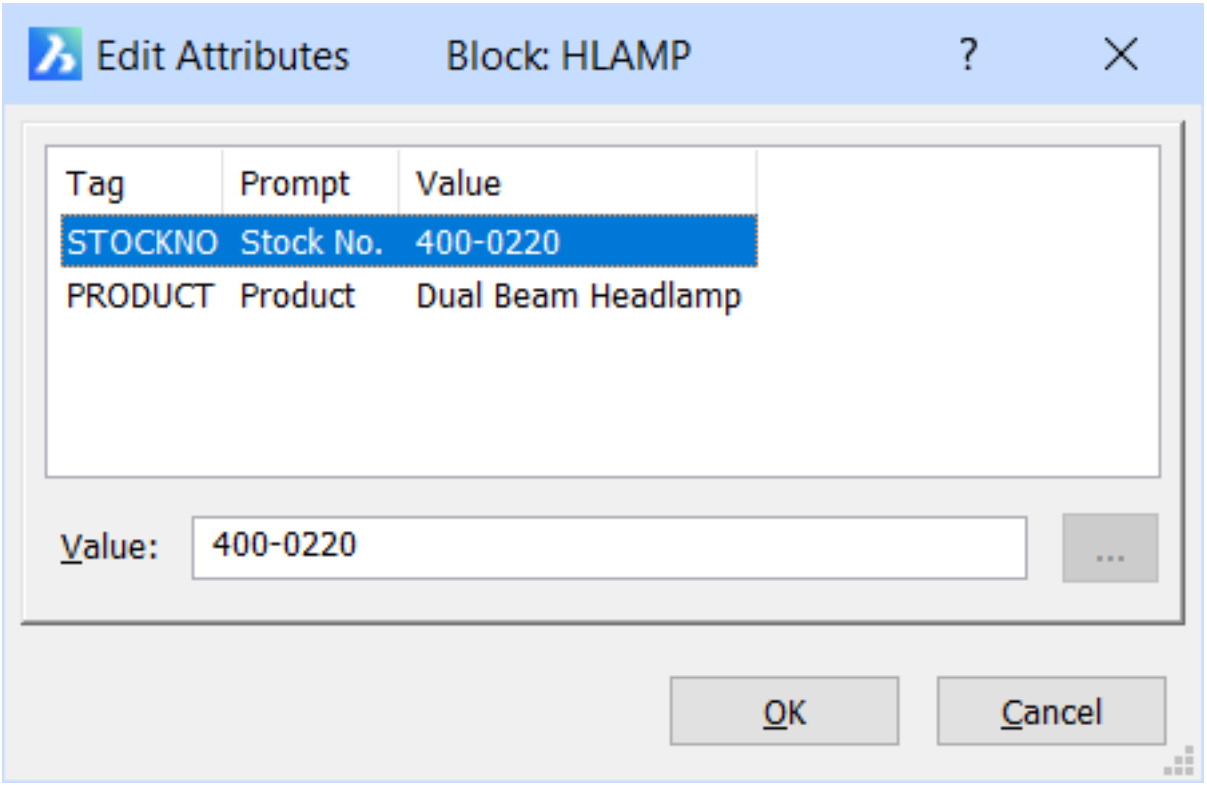
6.10.13 DdAttE

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits the values of attributes through a dialog box (short for "dynamic dialog attribute editor"). This command is superseded by the **BATTMAN (6.6.2)** command.

Accessing the Command

command bar: **ddatte** : **ddatte Prompts you in the command bar:** Select a block: **(Choose one block)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/ddatte-dlg.png}) Change the value of an attribute, and then click **OK**.

Figure 6.666



Command Options

Option	Description
Value	Changes the value of the attribute text. (The ... button is inoperative.)

Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttDisp (6.4.35) - toggles the display of attribute text. AttEdit (6.4.36) - edits attribute values and properties. -AttExt (6.4.29) - extracts attributes from drawings to data files through the command bar. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attribute Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities.s of a block definition. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. InsertAligned (6.17.15) - inserts blocks repeatedly, and inserts mirrored blocks. Properties (6.24.49) - edits attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place.

Figure 6.667



6.10.14 DdEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Edits single-line text, multi-line text (mtext), dimension text, and leader text (short for "dynamic dialog editor").

Accessing the Command

command bar: **ddedit** alias: **ed** mouse: double-click single-line text, multi-line text (mtext), attribute definition or text, dimension or leader. : **ddedit Prompts you in the command bar:** Select entity to modify: **(Choose a text entity)** Displays a dialog box; edit the text, and then click **OK**. The command repeats: Select entity to modify: **(Choose a text entity, or else press Enter to end the command)** The interface you see for the DdEdit command depends on the value of the **TextEd** system variable:

TextEd Value	Description
0	DdEdit displays the in-place editor, which has no user interface (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/ddtexted-0.png})
1	DdEdit displays the dialog box editor for text (shown below) or for mtext: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/ddtexted-1.png})
2 (the initial value)	DdEdit displays the in-place editor, and then repeats the 'Select entity to modify' prompt automatically

Command Options

No options.

Procedures

- To edit a text entity (1.21.19)
- To edit a selection of Mtext entities (1.21.19)
- To select a different multiline text editor (1.21.19)
- To edit tag, prompt or default text of an attribute (1.22.12)
- To edit an attribute definition (1.22.12)

Related Commands

Text (6.29.12) - places single-lines of text in drawings. -Text (6.29.12) - places text and evaluates LISP expressions. MText (6.21.35) and -MText (6.21.33) - places paragraph text. DdAtte (6.10.13) - edits attribute values. Style (6.27.53) - creates and modifies text styles through the Drawing Explorer. -Style (6.27.54) - creates and modifies text styles through the command bar. BmBalloon (6.8.1) - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout. Change (6.9.10) - changes the position of text. TxtExp (6.29.27) - explodes text into polylines. Spell (6.27.63) - checks the spelling of text in drawings. Properties (6.24.49) - modifies the properties of text. Find (6.14.8) - finds and replaces text.

Figure 6.668



Figure 6.669

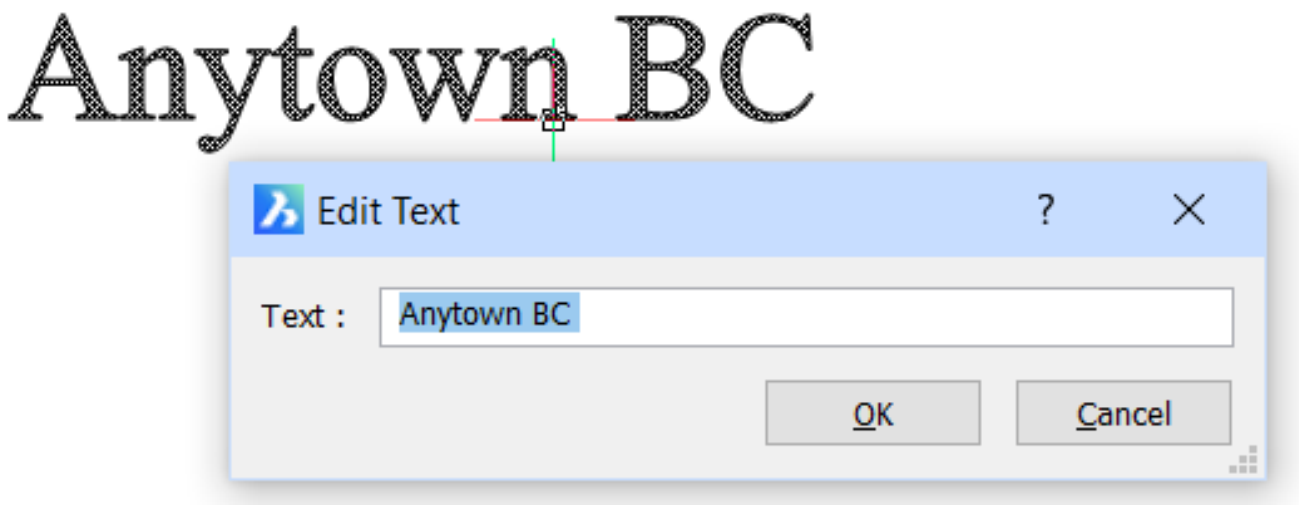


Figure 6.670



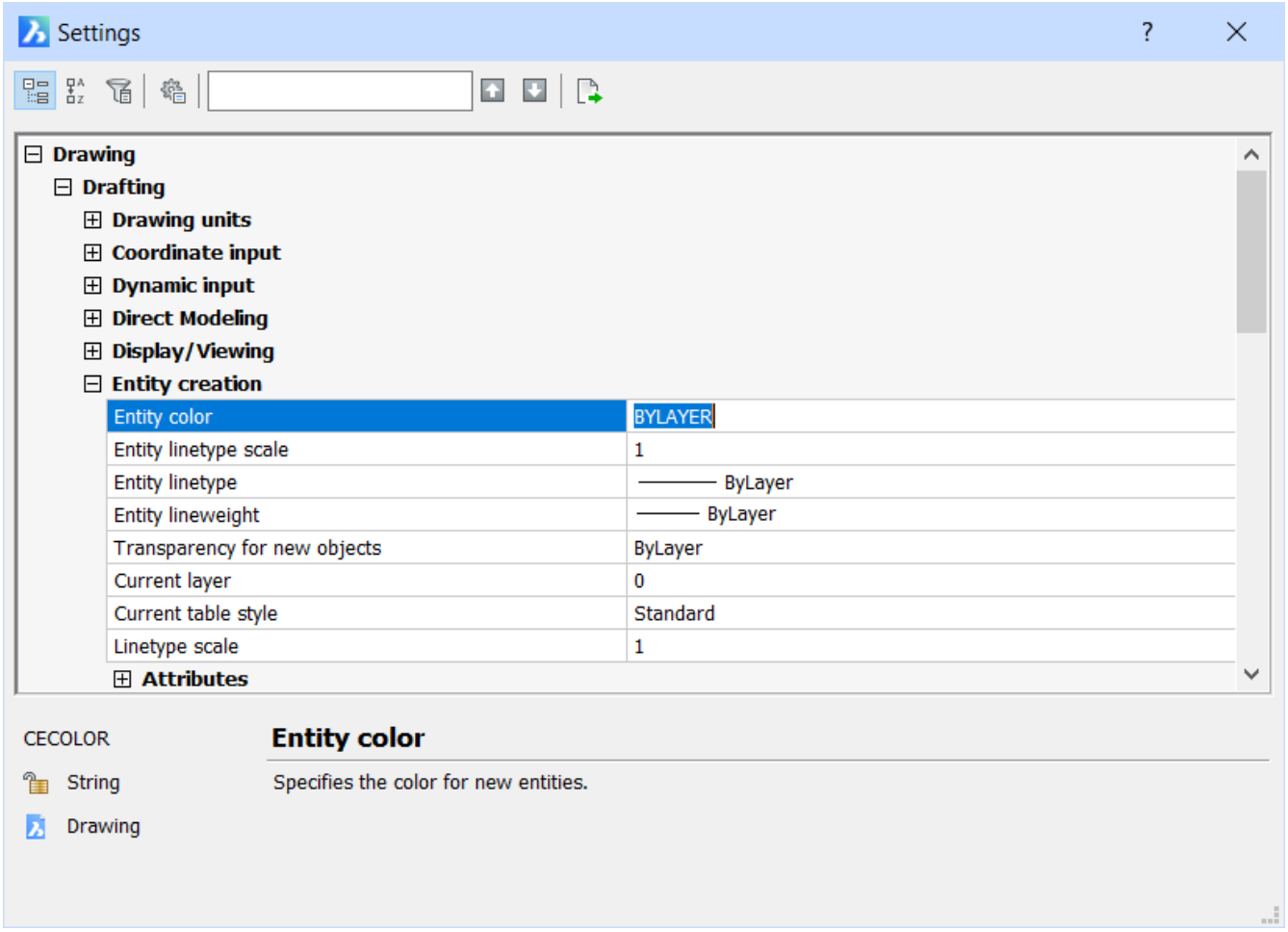
6.10.15 DdEModes

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets default values for creating entities (short for "dynamic dialog entity modes"); displays the Entity Creation section of the Settings dialog box. This command is superseded by the **Settings (6.27.23)** command.

Accessing the Command

command bar: **ddemodes** alias: **settings (6.27.23)** : **ddemodes** Displays Settings dialog box at the Entity Creation section: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/ddemodes-dlg.png}) Change one or more settings, and then click **X**.

Figure 6.671


Settings
Drawing
Drafting
Drawing units
Coordinate input
Dynamic input
Direct Modeling
Display/Viewing
Entity creation
Entity color
BYLAYER
Entity linetype scale
1
Entity linetype
ByLayer
Entity lineweight
ByLayer
Transparency for new objects
ByLayer
Current layer
0
Current table style
Standard
Linetype scale
1
Attributes
CECOLOR
String
Drawing
Entity color
Specifies the color for new entities.

Command Options

Option	Description
Entity Color	Specifies the default color of new entities; initial value is ByLayer, which means that entity colors are governed by layer properties.
Entity Linetype Scale	Specifies the default scale factor for linetypes; initial value is 1.0000.
Entity Linetype	Specifies the default linetype for new entities; initial value is ByLayer, which means that entity linetypes are governed by layer properties.
Entity Lineweight	Specifies the default linetype for new entities; initial value is ByLayer, which means that entity lineweights are governed by layer properties.
Current Layer	Specifies the default layer for new entities; initial value is layer 0.
Current Table Style	Specifies the initial value of the table style name for new tables.
Linetype Scale	Specifies the global linetype scale factor; this factor affects entity linetype scale factors.

Related Commands

Color (6.9.33) and -Color (6.9.35) - set the working color. Linetype (6.20.33) and -Linetype (6.20.32) - loads linetype definitions and set the current linetype. LWeight (6.20.41) - sets the current linetype. Layer (6.20.2) and -Layer (6.20.3) - create layers and set their properties. Properties (6.24.49) - reports and changes properties through a palette.

Figure 6.672



6.10.16 DdFilter

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a selection set of the entities (short for "dynamic dialog filter"). This command is superseded by the **Select (6.27.17)** command.

Accessing the Command

command bar: **ddfilter** alias: **select (6.27.17)** : **ddfilter Prompts you in the command bar:** Select entities to include in set: **(Choose one or more entities using any selection mode)** S elect entities to include in set: **(Choose more entities, or else press Enter to exit the command)** The selection set is invisible, and so you access it by entering "**P**" (previous) the next time a command prompts you "**Select entities**."

Command Options

Option	Description
Select entities to include in set	Selects one or more entities using any selection mode.

Related Commands

Properties (6.24.49) - filters entities through the Quick Select button. Select (6.27.17) - pre-selects entities based on their location.

Figure 6.673



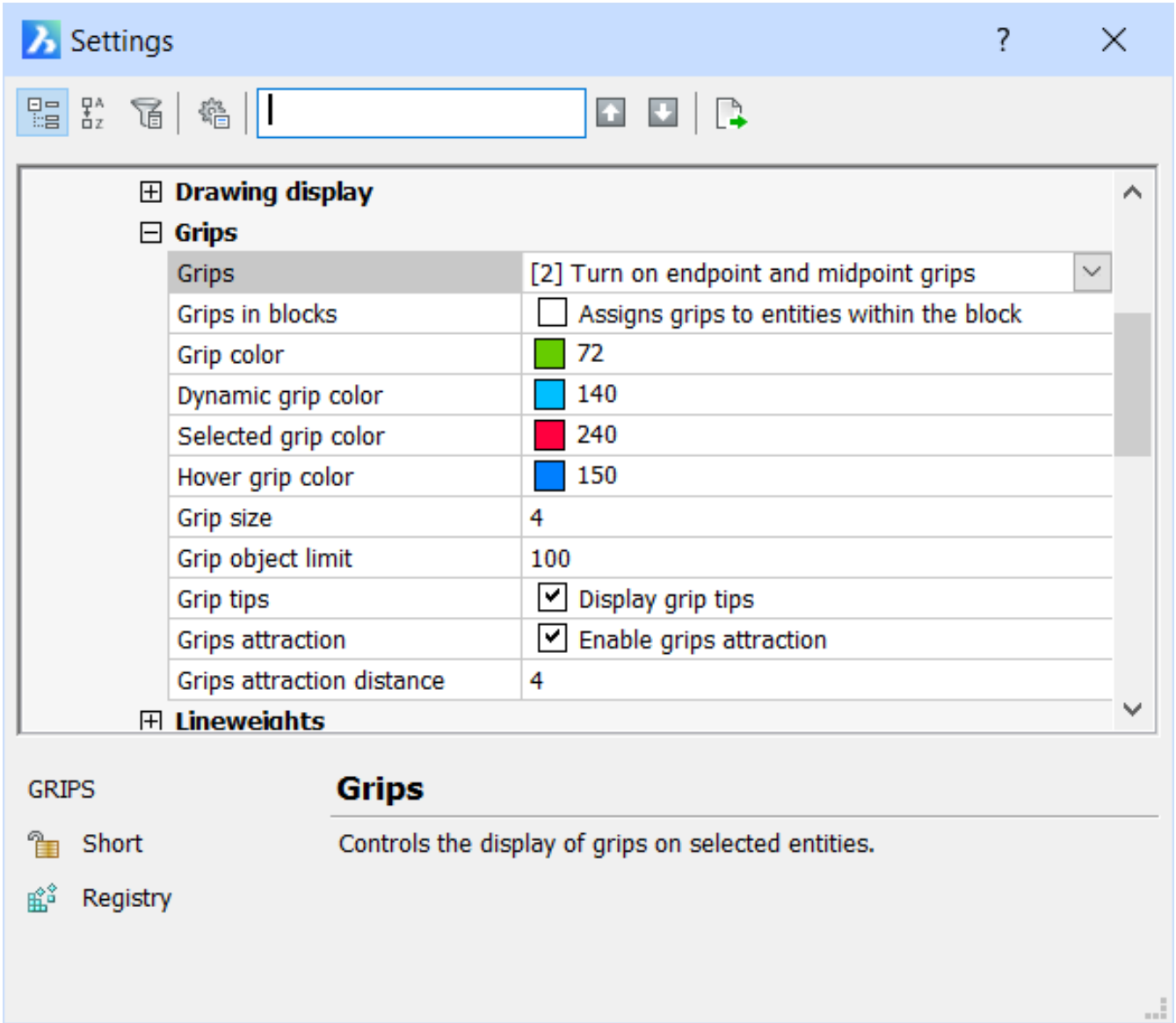
6.10.17 DdGrips

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the Grips section of the Settings dialog box (short for "dynamic dialog grip settings"). This command is superseded by the **Settings (6.27.17)** command.

Accessing the Command

command bar: **ddgrips** transparent: **'ddgrips** aliases: **gr, settings (6.27.23)** : **ddgrips** Displays the Settings dialog box at the Grips section: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/ddgrips-dlg.png}) Change one or more settings, and then click **X**.

Figure 6.674


Settings
Drawing display
Grips
Grips
[2] Turn on endpoint and midpoint grips
Grips in blocks
Assigns grips to entities within the block
Grip color
72
Dynamic grip color
140
Selected grip color
240
Hover grip color
150
Grip size
4
Grip object limit
100
Grip tips
Display grip tips
Grips attraction
Enable grips attraction
Grips attraction distance
4
Lineweights
GRIPS
Short
Registry
Grips
Controls the display of grips on selected entities.

Command Options

Option	Description
Grips	Sets the display of grips when entities are selected. Allows to control the display of midpoint grips on polyline segments. • **0** - no grips displayed • **1** - grips displayed; on polyline segments, only the endpoint grips are displayed • **2** - grips displayed; on polyline segments, both endpoint and midpoint grips are displayed (initial value)
Grips in blocks	Toggles the display of grips inside blocks: • **On** - displays all grips on all entities making up the block (initial value) • **Off** - display a singe grip at the block's insertion point
Grip color	Specifies the color of unselected grips; initial value = index color 72 (bright green)
Dynamic grip color	Specifies the color of custom grips for dynamic blocks; initial value= index color 140 (light blue)
Selected grip color	Specifies the color of selected grips; initial value= index color 240 (red)
Hover grip color	Specifies the color of unselected grips when the cursor pauses over it; initial value= index color 150 (dark blue)
Grip size	Specifies the size of grips, in pixels;initial value = 4
Grip object limit	Specifies the maximum number of entities displayed with grips; initial value = 100.When the selection set is of more than 100 entities, no grips are displayed. When set to 0, there is no limit to the number of entities that display grips. Maximum limit is 32,767 grips.
Grip Tips	Determines whether grip tips display when the cursor hovers over grips on custom entities or dynamic blocks that support grip tips (not supported yet)
Grips attraction	Toggles the use of **magnetism** of entity grips, whether the cursor jumps top the grip;initial value = on
Grips attraction distance	Sets the grips attraction distance in pixels;initial value = 4

Related Commands

Select (6.27.17) - selects entities. SelGrips (6.27.22) - elects entities and displays grips.

Figure 6.675



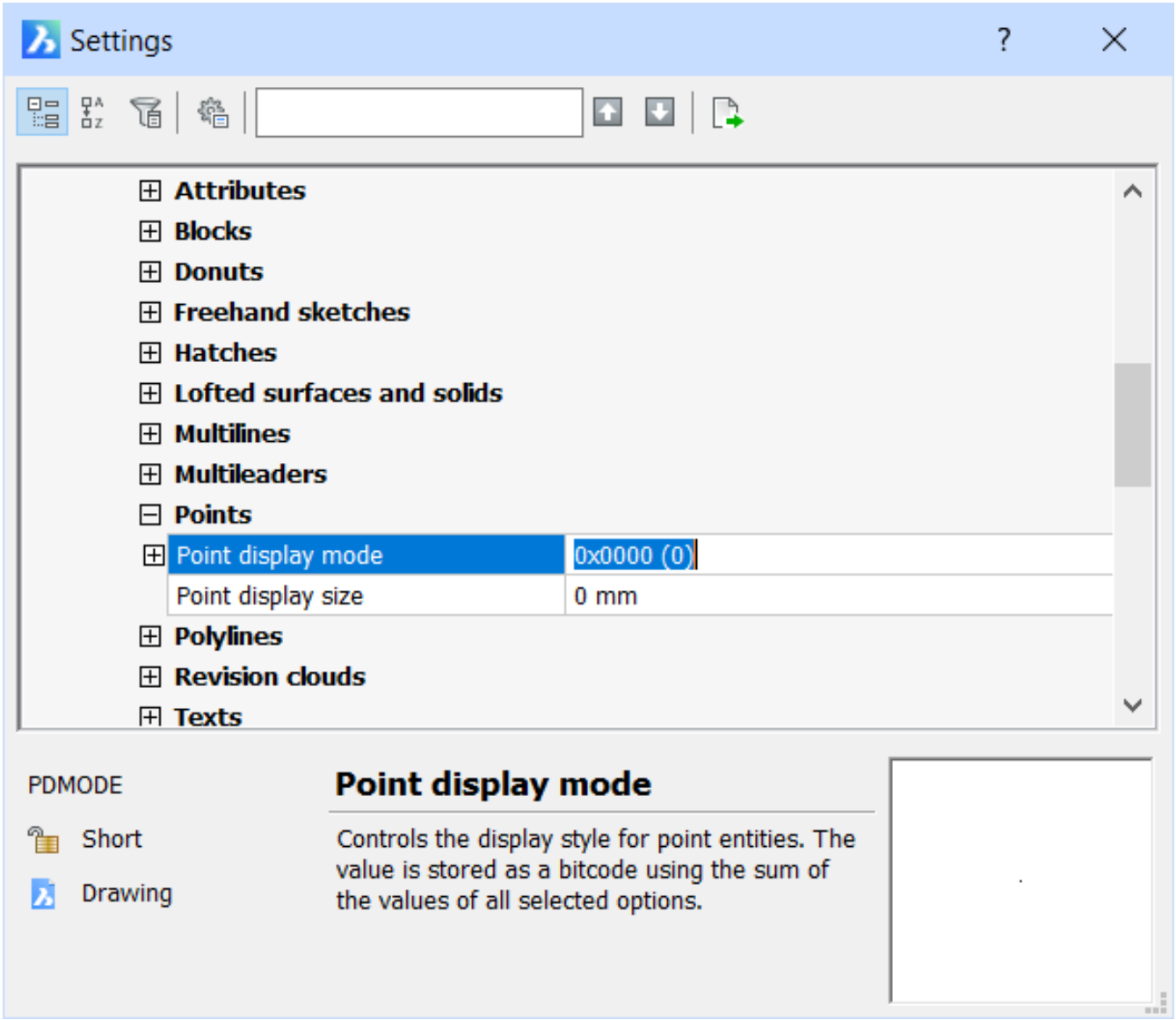
6.10.18 DdPType

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Specifies the look and size of point entities, through the Points section of the Settings dialog box (short for "dynamic dialog point type"). This command is superseded by the **Settings (6.27.23)** command.

Accessing the Command

command bar: **ddptype** transparent: **'ddptype** alias: **settings (6.27.23) : DdPType** Displays the Settings dialog box at the Points section: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/ddptype-dlg.png}) Change one or more settings, and then click **X**.

Figure 6.676


Settings
Attributes
Blocks
Donuts
Freehand sketches
Hatches
Lofted surfaces and solids
Multilines
Multileaders
Points
Point display mode
0x0000 (0)
Point display size
0 mm
Polylines
Revision clouds
Texts
PDMODE
Short
Drawing
Point display mode
Controls the display style for point entities. The value is stored as a bitcode using the sum of the values of all selected options.

Command Options

Option	Description
Point Display Mode	Specifies the look of points: • **Mask**- points are displayed as . (0), + (2), x (3), or ’ (4) • **Circle** - points are displayed as **mask** + circle:  • **Box** - points are displayed as **mask** + square: 
Point Display Size	Specifies the size of points: • **0** - points are sized to be 5% of the viewport’s height • Positive number - size of the point in pixels • Negative number - size of the point as a percentage of the viewport’s height

Related Commands

Point (6.24.31) - places points in drawings. Divide (6.10.33) - places evenly-spaced points along entities. Measure (6.21.13) - places equi-distant points along entities.

Figure 6.677



6.10.19 DdSetVar

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the Settings dialog box (short for "dynamic dialog set variables"). This command is superseded by the **Settings (6.27.23)** command.

Accessing the Command

command bar: **ddsetvar** alias: **settings (6.27.23) : DdSetVar** See the Settings command.

Related Command

SetVar (6.27.26) - change the values of system variables through the command bar. Settings (6.27.23) - change the values of system variables through a dialog box. SettingsSearch (6.27.24) - opens the Settings dialog box at the specified category, system variable or user preference.

Figure 6.678



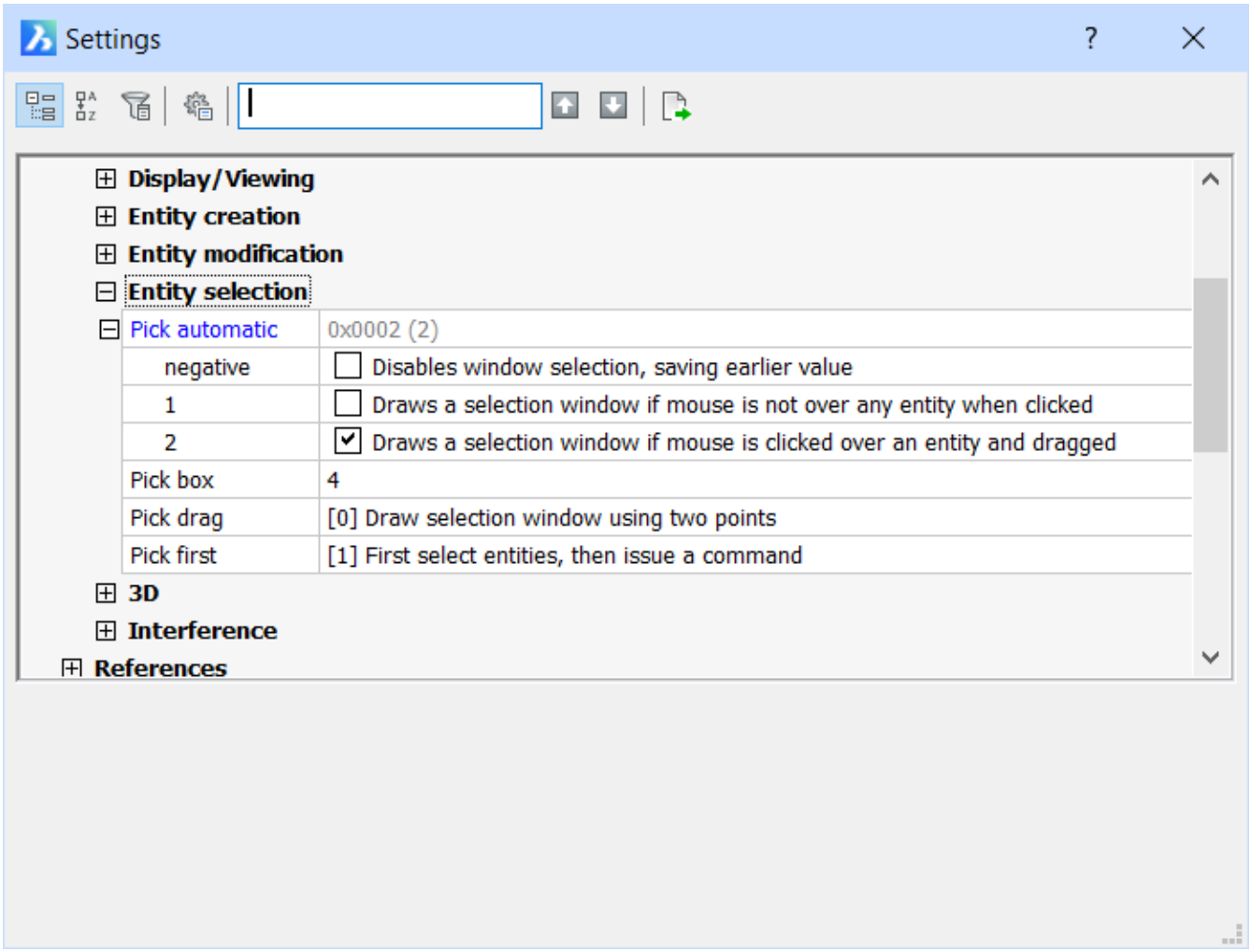
6.10.20 DdSelect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Specifies the properties for selecting entities, through the Settings dialog box (short for "dynamic dialog selection"). This command is superseded by the **Settings (6.27.23)** command.

Accessing the Command

command bar: **ddselect** transparent: **'ddselect** alias: **se, settings (6.27.23) : DdDelect** Displays the Entity Selection section of the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dlg.png}) Change one or more settings, and then click **X**.

Figure 6.679


Settings
Display/Viewing
Entity creation
Entity modification
Entity selection
Pick automatic
0x0002 (2)
negative
Disables window selection, saving earlier value
1
Draws a selection window if mouse is not over any entity when clicked
2
Draws a selection window if mouse is clicked over an entity and dragged
Pick box
4
Pick drag
[0] Draw selection window using two points
Pick first
[1] First select entities, then issue a command
3D
Interference
References

Command Options

Option	Description
Pick Automatic	Toggles whether BricsCAD automatically creates window or crossing selections at the 'Select entities' prompt: • **negative number** - disables window selection when no command is active. Requires a single pick point to select an entity, or the entry of a selection mode option when a command prompts to select entities. • **1** - enables window selection if the mouse is not over an entity when you click • **2** - enables window selection if the mouse is clicked when over an entity and dragged. This option is recommended when working in a rendered 3D model
Pick Box	Specifies the size of the pick box; range is 0 to 50 pixels; initial value = 3 pixels.
Pick Drag	Determines how selection windows are created: • **0** - draws selection window using two points; default • **1** - draws selection window using dragging
Pick First	Determines how entities are selected: • **0** - first issues a command, and then selects entities. • **1** - first select entities, then issues the command; default.

Related Commands

Select (6.27.17) - selects entities. SelGrips (6.27.22) - elects entities and displays grips.

Figure 6.680



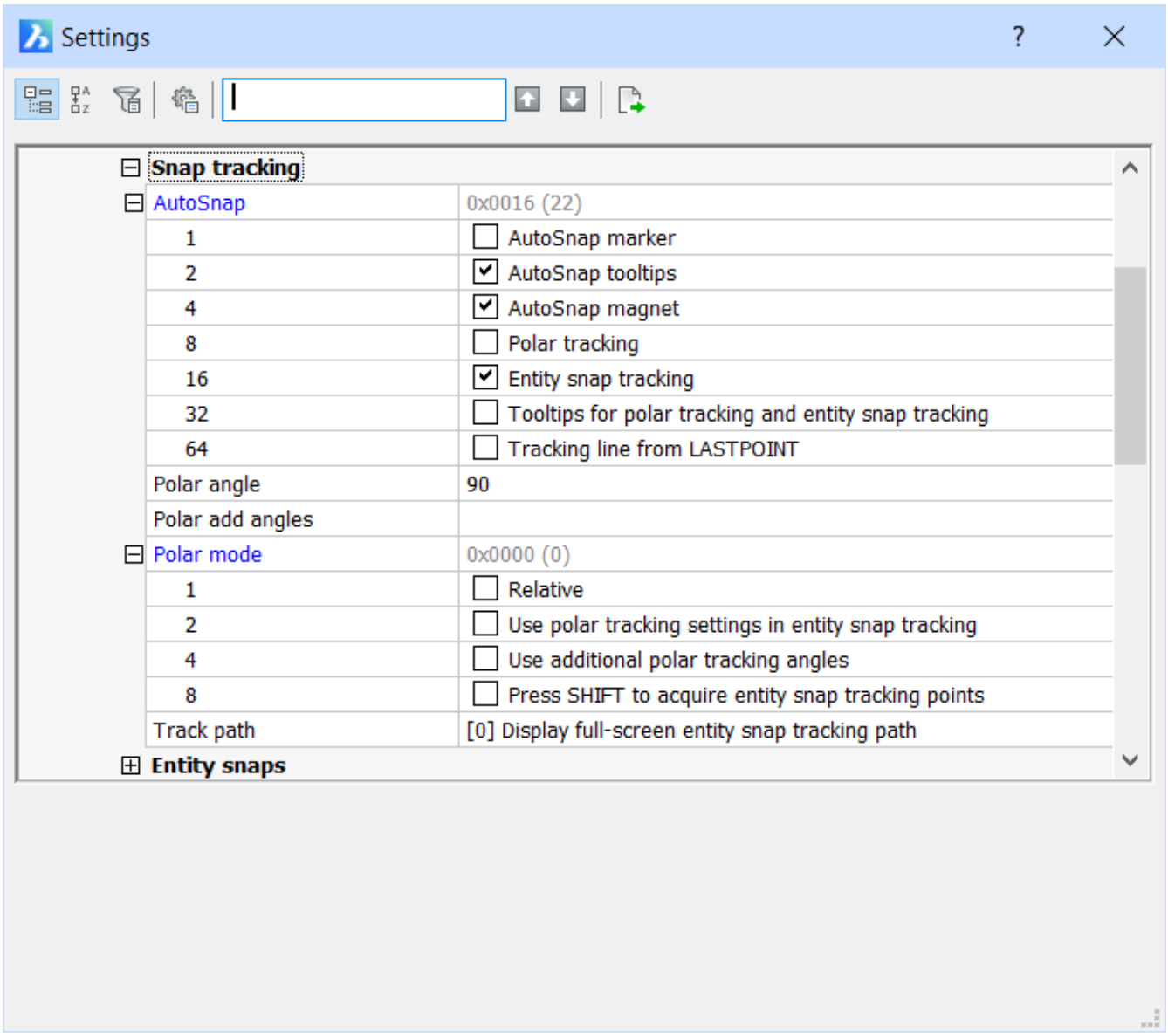
6.10.21 DdSTrack

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the properties for snap tracking by displaying the Snap Tracking section of the Settings dialog box (short for ”dynamic dialog snap tracking”).

Accessing the Command

command bar: **ddstrack** transparent: **'ddstrack** alias: **settings (6.27.23)**status bar: **STRACK : DdSTrack** Displays the Snap Tracking section of the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/C dlg.png}) Change settings, and then click X.

Figure 6.681


Settings
Snap tracking
AutoSnap
0x0016 (22)
1
AutoSnap marker
2
AutoSnap tooltips
4
AutoSnap magnet
8
Polar tracking
16
Entity snap tracking
32
Tooltips for polar tracking and entity snap tracking
64
Tracking line from LASTPOINT
Polar angle
90
Polar add angles
Polar mode
0x0000 (0)
1
Relative
2
Use polar tracking settings in entity snap tracking
4
Use additional polar tracking angles
8
Press SHIFT to acquire entity snap tracking points
Track path
[0] Display full-screen entity snap tracking path
Entity snaps

Command Options

Option	Description
AutoSnap	Specifies the look and action of AutoSnap: • **1 AutoSnap Marker** - toggles the display of the AutoSnap marker, which indicates the type of entity snap (6.23.12); default = off • **2 AutoSnap Tooltips** - toggles the display of tooltips adjacent to the entity snap location; default = on • **4 AutoSnap Magnet** - toggles whether AutoSnap jumps to the nearest entity snap geometry; default = on • **8 Polar Tracking** - toggles polar tracking, where BricsCAD shows polar angles; default = off • **16 Entity Snap Tracking** - toggles entity snap (6.23.12) tracking; default = on • **32 Tooltips for Polar Tracking and Entity Snap Tracking** - toggles the display of tooltips; default = off • **64 Tracking line from LASTPOINT** - toggle the display of a tracking line from the most recently specified point; default = off
Polar Angle	Specifies the default polar snap increment; default = 90.
Polar Add Angles	Specifies additional user-defined angles for polar snap; default = none.
Polar Mode	Specifies how polar mode is handled. The value is stored as the sum of the values of the selected options: • **1 Relative**- measures polar angles from selected entities (relative); default • **2 Use polar tracking settings in entity snap settings** - use polar tracking settings in entity snap tracking • **4 Use additional polar tracking angles** - use the tracking angles listed in Polar Add Angles • **8 Press SHIFT to acquire entity snap tracking points**- press the Shift-key to acquire entity snap tracking points
Track Path	Determines how the tracking path is displayed. The options are: • **0** - displays full-screen entity snap tracking path. • **1**- displays entity snap path only between the alignment path and the From point to the cursor location; default. • **2** - does not display polar tracking path. • **3**- does not display polar or entity snap tracking paths.

Related Commands

-OSnap (6.23.13) - toggles entity snap modes at the command line. OSnap (6.23.12) - sets entity snaps through the Settings dialog box.

Figure 6.682



6.10.22 DdVPoint

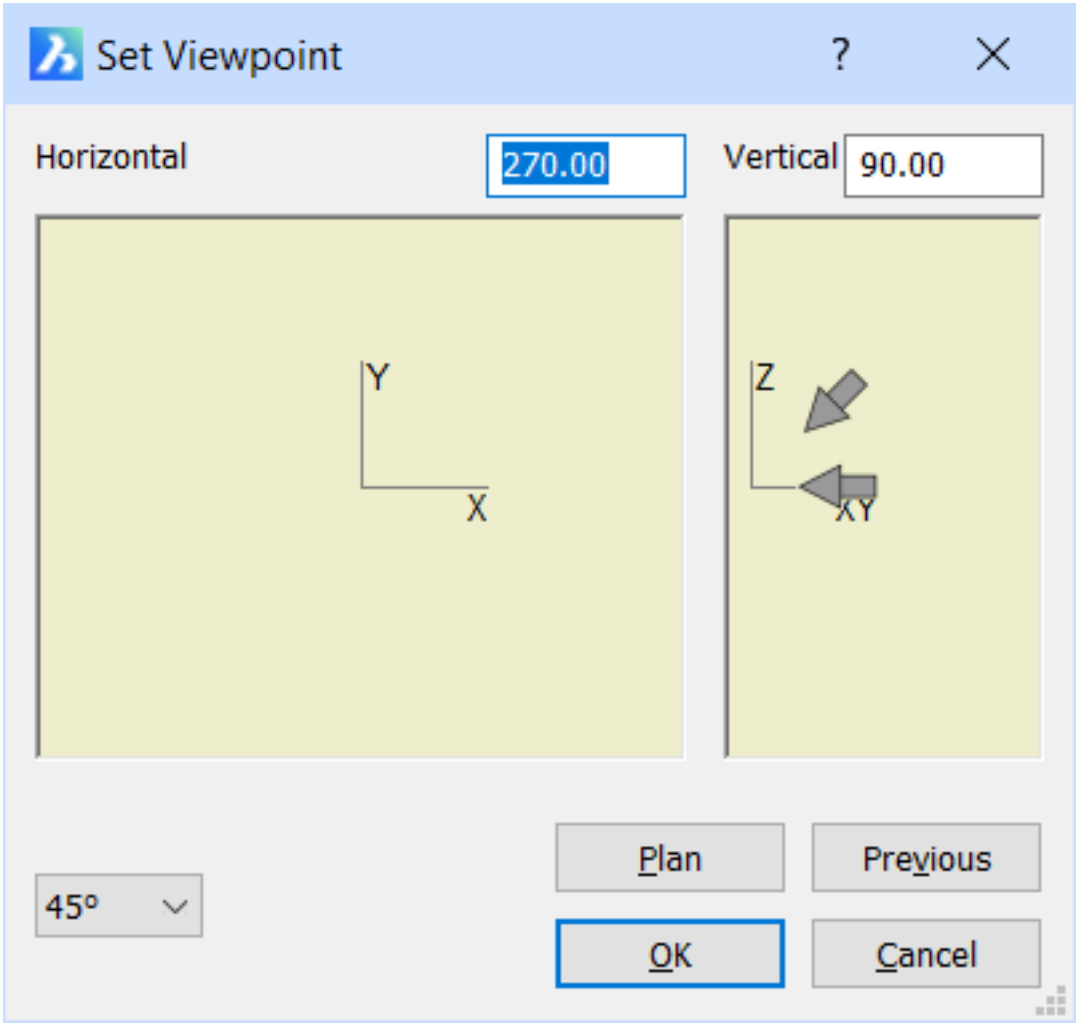
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the current 3D viewpoint or plan view, through a dialog box (short for "dynamic dialog view point"). This command is superseded by the **LookFrom (1.16.8)** control.

Accessing the Command



command bar: **ddvpoint** aliases: **setvpoint, viewctl, vp** menu bar: **View | Set Viewpoint** toolbar: **View |**  ribbon:**View | View | Viewpoint | Set Viewpoint : DdVPoint** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/ddvpoint-dlg.png}) Choose a horizontal and/or vertical viewing angle, and then click **OK**.

Figure 6.683


Set Viewpoint
?
Horizontal
270.00
Vertical
90.00
Y
X
Z
XY
45°
Plan
Previous
OK
Cancel

Command Options

Option	Description
Horizontal	Specifies the horizontal angle from the X-axis.Click a preset angle or type a value in the **Horizontal** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/DDVPo (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/DDVPo Click at 0°, 45°, 90°, 135°, 180°, 225°, 270° or 315° in the inner circle.The shape of the arrow indicates whether the view is horizontal () or at an angle () (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/DDVPo
Vertical	Specifies the vertical angle from the XY plane.Click a preset angle, or else type a value in the **Vertical** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/D (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/D Click in the inner half circle. (1) View direction is horizontal. (2)View direction is 45° down (not available if **PresetAngle** = 45°). (1)(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/D (2)(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/D View direction is 45° down, except for 0°, 90°, 180° and 270° horizontal angles, which generate orthographic views: front, back, left and right. (1) First select the horizontal arrow, then (2) click the 45° down arrow. (1)(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/D (2)(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/D
Angle Preset	Sets markers at 5, 15, or 45 degrees; default = 45.
Plan	Reverts the 3D viewpoint to 2D plan view.
Previous	Reverts to the previous viewpoint.
OK	Sets the viewpoint and exits the dialog box.

Look From control

The Look From control (1.16.8) allows to select a number of preset views such as orthographic views and isometric views.

Related Commands

VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. DView (6.10.45) - changes the 3D viewpoint interactively, and turns on perspective mode. RtRot (6.26.38) - rotates 3D drawings in real-time.

Figure 6.684

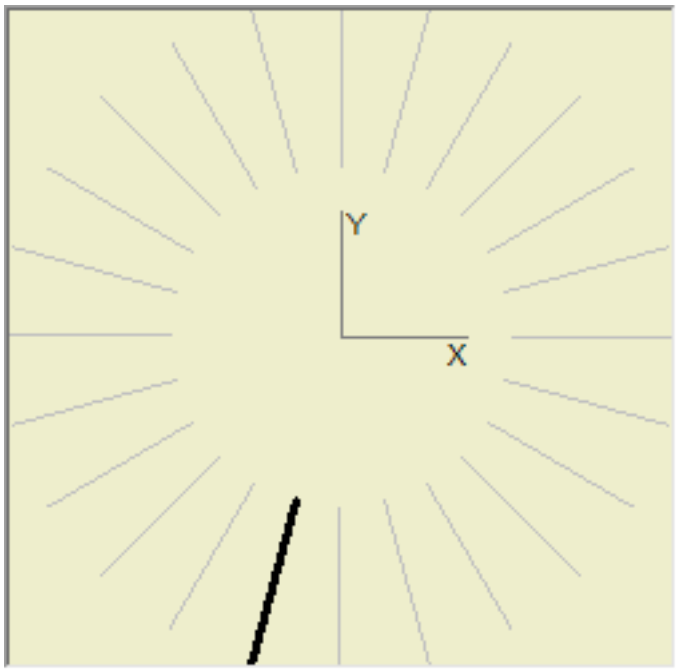


Figure 6.685



Figure 6.686

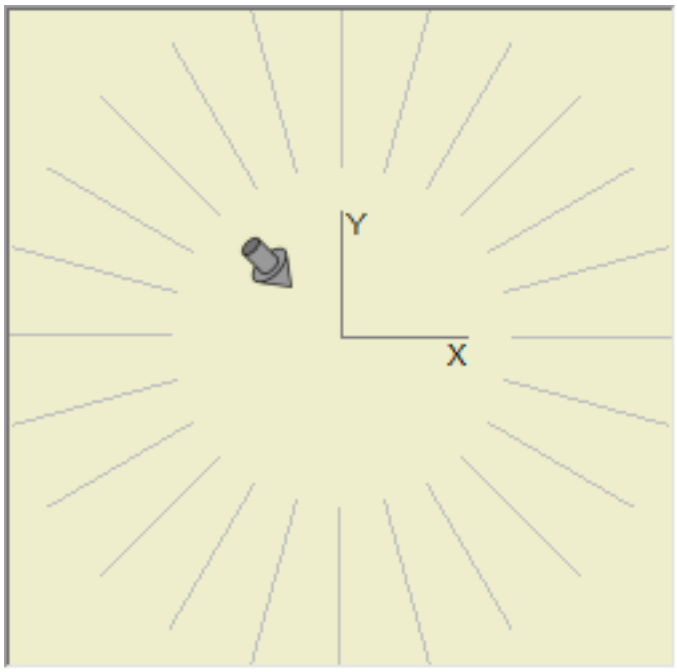


Figure 6.687

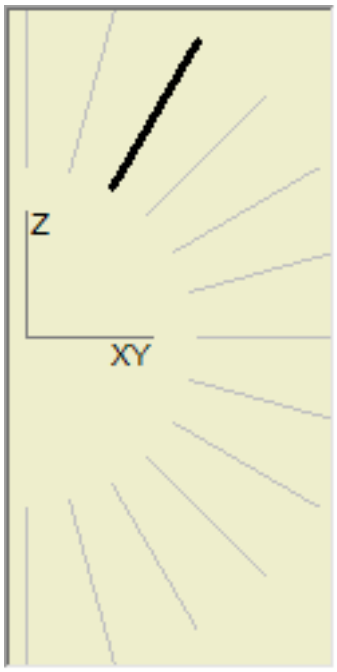


Figure 6.688



Figure 6.689

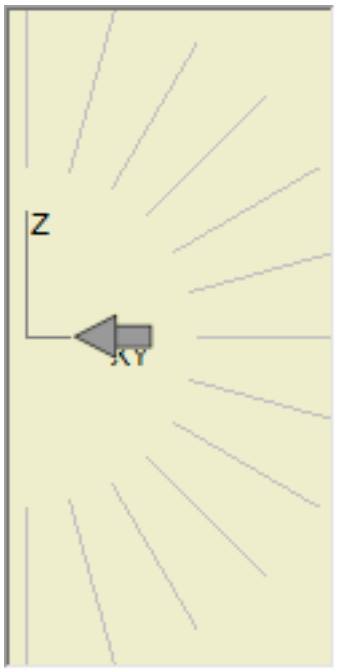


Figure 6.690

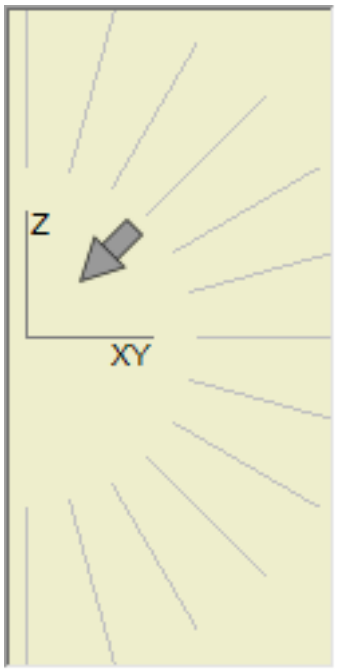


Figure 6.691

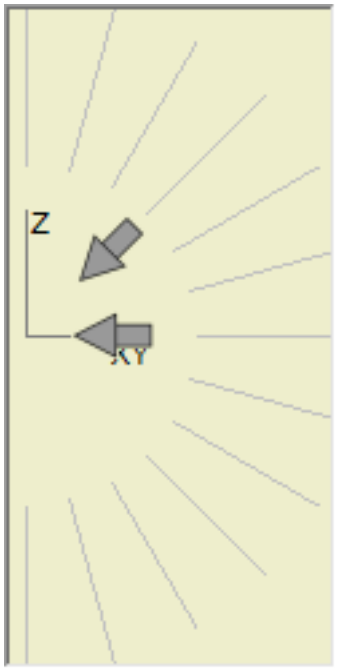


Figure 6.692

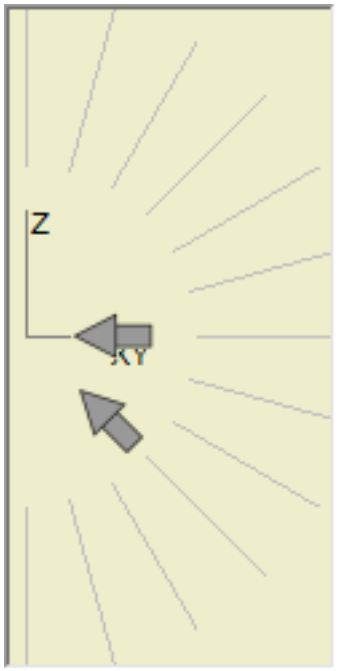


Figure 6.693



6.10.23 Delay

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Delays execution of the next command; meant for use with scripts; see **Script (6.27.9)** command.

Accessing the Command

command bar: **delay** transparent: **'delay** : **Delay Prompts you in the command bar:** Milliseconds to delay: **(Enter a number between 0 and 2,147,483,627 -- approximately 24 days)**

Command Options

Option	Description
Milliseconds to Delay	Specifies the amount of time BricsCAD waits before continuing to the next command in the script; specified in milliseconds (1/1000th of a second).

Related Commands

RecScript (6.26.5) - records scripts to SCR files. StopScript (6.27.48) - stops recording the script, and saves it to an SCR file. Script (6.27.9) - loads and runs a SCR script file. RScript (6.26.36) - reruns the script. Resume (6.26.28) - resumes a stopped script.

Figure 6.694



6.10.24 DelConstraint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Removes dimensional and geometrical constraints from selected entities.

Accessing the Command

 command bar: **delconstraint** menu bar: **Assembly | Deletee 2D Constraints** toolbar: **2D Constraints |**  ribbon:**Parametric | 2D Constraints | Delete 2D Constraints** : **DelConstraint Prompts you in the command bar:** Select entities to delete all constraints: **(Select one or more entities, or enter all to selectall entities)** Entities in set: 1 Select entities to delete all constraints: **(Select more entities, or else press Enter to complete the command)** The dimensional and geometric constraints of the selected entities are deleted.

Command Options

Option	Description
Select entities to delete all constraints	Selects the entities from which to delete constraints: • **Pick** - one or more entities • Type **all** - selects all entities in the drawing • Press **Enter**- ends the command Both dimensional and geometric constraints are removed from the entities.

Note:To remove individual constraints one at a time:

- **Dimensional constraints** - select the dimensional constraint, and then press **Del**
- **Geometric constraints** - click the tiny **x** on the constraint bar:



Related Commands

-Parameters (6.24.5) - manages 2D dimensional constraint parameters and user variables in the current drawing. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. Geomconstraint (6.15.16) - applies geometric relationships between entities and/or valid constraint points on entities. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. DcDisplay (6.10.9) - allows to set dimensional constraints to be displayed or hidden.dimensions.

Figure 6.695



6.10.25 DelEData

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Deletes extended entity data for specific applications from the selected entities (short for ”delete entity data”).

Accessing the Command

 command bar: **deledata** transparent: **'deledata** menu bar: **Tools | Entity Data | Delete Entity Data** toolbar: **Entity Data |**  **: DelEData Prompts you in the command bar:** Delete Entity Data: Name of application or [?]: **(Enter name of application, or else type ? for a list of names)** Select entities to delete Entity Data: **(Choose one or more entities from which to delete entity data)** Entities in set: 1 Select entities to delete Entity Data: **(Select more entities, or else press Enter to end command)**

Command Options

Option	Description
?	Lists names of applications loaded in this session of BricsCAD
Name of Application	Specifies the name of the application associated with the extended entity data
Select Entities to Delete Entity Data	Specifies the entities from which to erase extended entity data

Related Commands

CopyEData (6.9.53) - copies extended entity data to other entities. EditEData (6.13.3) - creates and edits extended entity data. MoveEData (6.21.32) - moves extended entity data to other entities. ReAssocApp (6.26.2) - associates applications with entities.

Figure 6.696



6.10.26 DesignTable

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Imports .csv files that contain sets of parameter values or else creates them from scratch. Parameters are added to the **MechanicalBrowserOpen (3.2.2)**panel and to parametric component properties.

Accessing the Command

 command bar: **designtable** transparent: **'designtable** menu bar:

- **Parametric | Design Table from file**
- **Parametric | Empty Design Table**

toolbar:**Design Table** ribbon panel: **Sheet Metal |Constraints | Design Table : DesignTable Prompts you in the command bar:** Create design table [From file/Empty] <From file>: **(Choose an option)**

Command Options

Option	Description
From file 	Creates a design table by importing a .csv file (comma-separated values file).Please make sure the list separator character on your system is the same as the one used in the CSV file. 1. Displays the Select Design Table File dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/file-dlg.png}) 2. Select the .csv file 3. Click the **Open** button to open the file in drawing 4. Enter the **MechanicalBrowserOpen** command to view the constraints (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/mechanicalbrowser-panel.png})

Empty 	Creates an empty design table from scratch; prompts you: 1. **Enter design table name <DesignTable_1> - (type a name, or else press Enter to accept the default name)** 2. **Enter parameter names ('*' to link all constant parameters) <*>** - specifies names of parameters using one the following methods: • Press Enter to add all parameters that have a constant value (i.e., that do not depend on another parameter) • Enter the names of the parameters to include in the design table; press Enter after each name. Press Enter twice to stop adding names and create the design table.

Procedures

- Creating a Design Table from a .csv file (1.19.2)
- Direct Lookup in a Design Table (1.19.2)
- Reverse Lookup in a Design Table (1.19.2)
- Adding a New Configuration to a Design Table (1.19.2)
- Creating an Empty Design Table (1.19.2)
- Using a Parametric Component with Design Tables (1.19.2)

Related Commands

-DesignTableEdit (6.10.27) - allows to edit design tables. -BmParameters (6.8.19) - allows to list and edit the parameters of inserted components and associative array -Parameters (6.24.5) - manages dimensional constraint parameters in the current drawing. MechanicalBrowserOpen (6.8.29) - opens the Mechanical Browser panel.

Figure 6.697

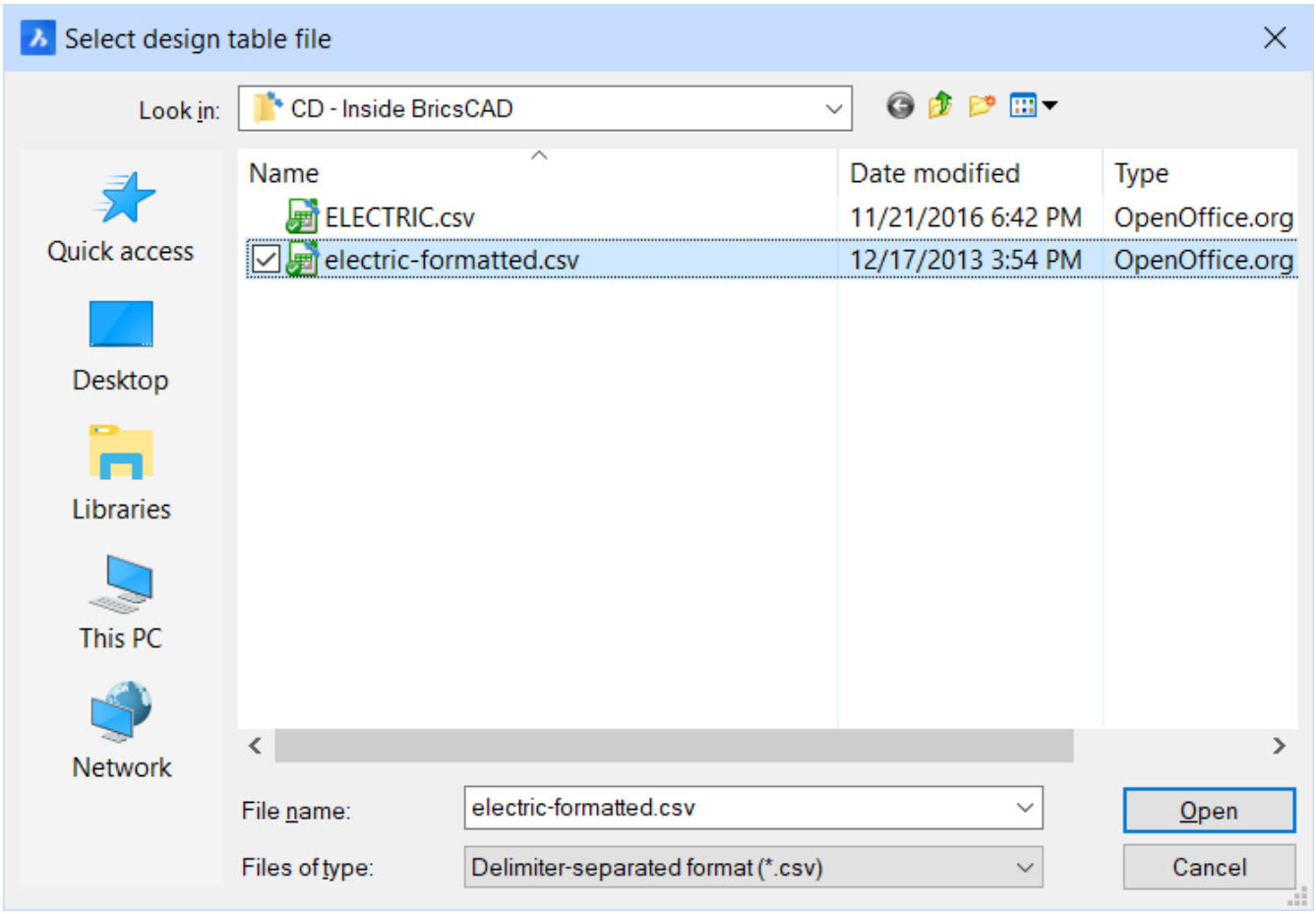

Select design table file
Look in:
CD - Inside BricsCAD
Quick access
Desktop
Libraries
This PC
Network
Name
Date modified
Type
ELECTRIC.csv
11/21/2016 6:42 PM
OpenOffice.org
electric-formatted.csv
12/17/2013 3:54 PM
OpenOffice.org
File name:
electric-formatted.csv
Files of type:
Delimiter-separated format (*.csv)
Open
Cancel

Figure 6.698

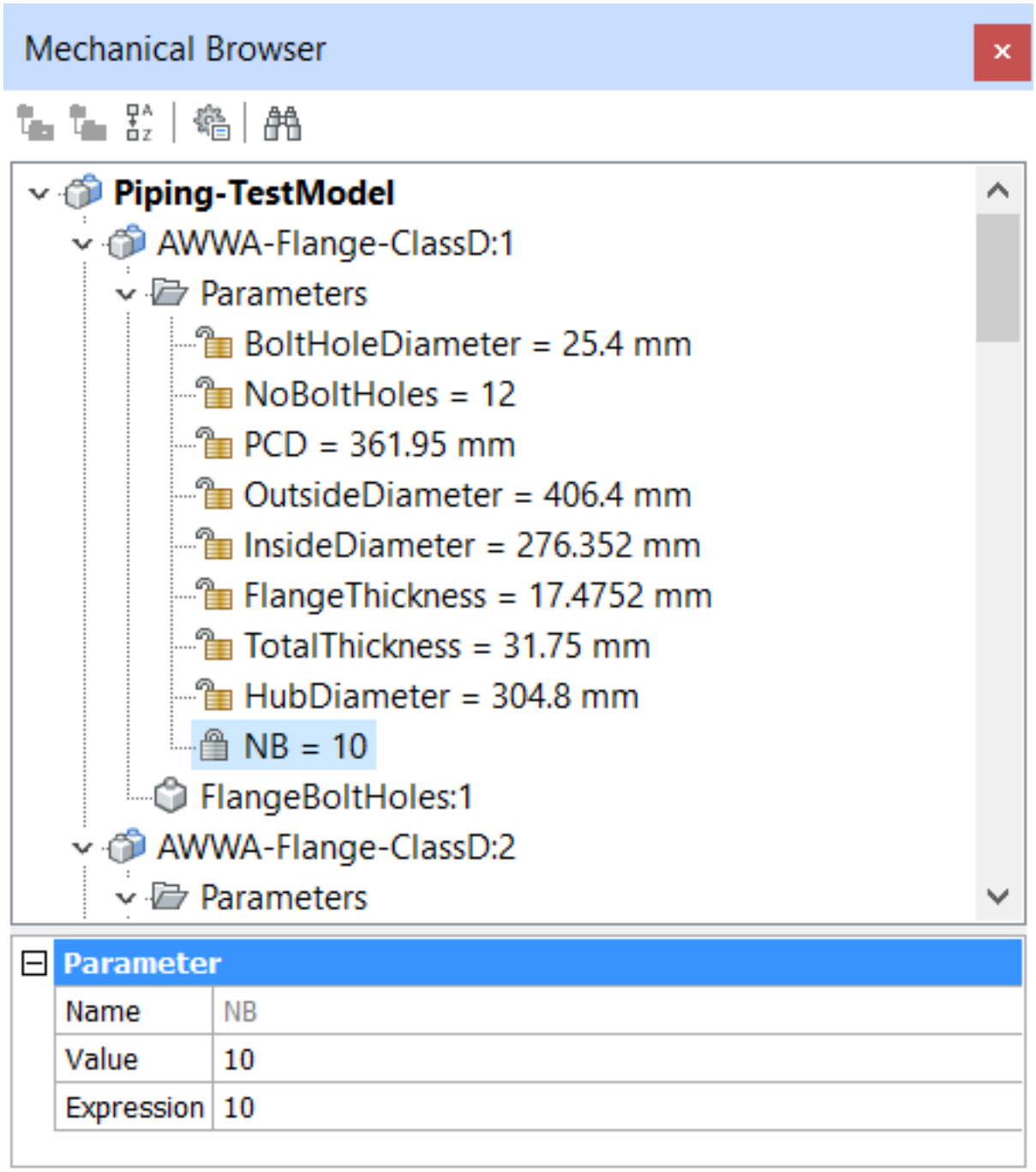

Mechanical Browser
Piping-TestModel
AWWA-Flange-ClassD:1
Parameters
BoltHoleDiameter = 25.4 mm
NoBoltHoles = 12
PCD = 361.95 mm
OutsideDiameter = 406.4 mm
InsideDiameter = 276.352 mm
FlangeThickness = 17.4752 mm
TotalThickness = 31.75 mm
HubDiameter = 304.8 mm
NB = 10
FlangeBoltHoles:1
AWWA-Flange-ClassD:2
Parameters
Parameter
Name NB
Value 10
Expression 10

Figure 6.699



6.10.27 -DesignTableEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Edits design tables.

Accessing the Command

command bar: **-designtableedit** : **-DesignRableEdit Prompts you in the command bar:** Design table [eXport/Replace/Delete/C configuration /? to list] <? to list>: **(Choose an option)**

Command Options

Option	Description
Export	Exports the content of a design table entity in .csv format file; prompts you: **Enter design table name <DesignTable_1>** - **(enter the name of the design table you want to export, or else press Enter to accept the default name.)** When the Select design table file dialog box displays, do thefollowing: 1. Type a name in the **Name** field 2. Select a folder 3. Click the **Save** button
Replace	Replaces the content of a design table with data from a .csv format file; prompts you: **Enter design table name <DesignTable_1>** - **(type the name of the design table you want to replace, or else press Enter to accept the default name.)** When the **Select design table** file dialog box displays, do thefollowing: 1. Select a .csv design table file 2. Click the **Open** button
Delete	Deletes a design table; prompts you: **Enter design table name <DesignTable_1>** - **(type the name of the design table you want to delete, or else press Enter to accept the default name.)**

Configurations	Creates, removes, or views configurations; prompts you: **Configuration [Save current/Remove/? to list] <? to list>** - **(choose an option:)** • **Save current**- creates a new configuration as a copy of an existing one. Prompts you: 1. **Enter design table name <v1_2>** - **(type the name of an existing table.)** 2. **Enter new configuration name <v1_2_1>** - **(type a name for the new table or press Enter to accept the default name.)** • **Remove** - removes a configuration.Prompts you: **Enter design table name <v1_2>** - **(type the name of the table to remove.)** • **? to list** - **(list the names of design tables attached to the current drawing): Design tables designtablename Total design tables: 1**
Parameters	Manages parameters; prompts you: **Parameter linked to design table [Unlink/Link/Clear expressions/? to list columns] <? to list columns>** - **(enter an option:)** • **Unlink** - unlinks parameters from the design table • **Link** - links parameters to the design table • **Clear expressions** - clears expressions (values and formulae) from parameters • **? to list columns** - lists parameters
Apply Configuration	Creates a new row using the current values of variables; prompts you: **Enter design table name <>** - **(enter the name of a design table)**
? to list	Lists design tables attached to the current drawing: **Design tables designtablename Total design tables: 1**.

Related Commands

-BmParameters (6.8.19) - allows to list and edit the parameters of inserted components and associative array. -Parameters (6.24.5) - manages dimensional constraint parameters in the current drawing. DesignTable (6.10.26) - creates or imports .csv files containing sets of parameter values. Drop-down lists with the parameters values from the tables are added to the Mechanical Browser (3.2.2) and parametric component properties. MechanicalBrowserOpen (6.8.29) - opens the Mechanical Browser panel.

Figure 6.700



6.10.28 DgnImport

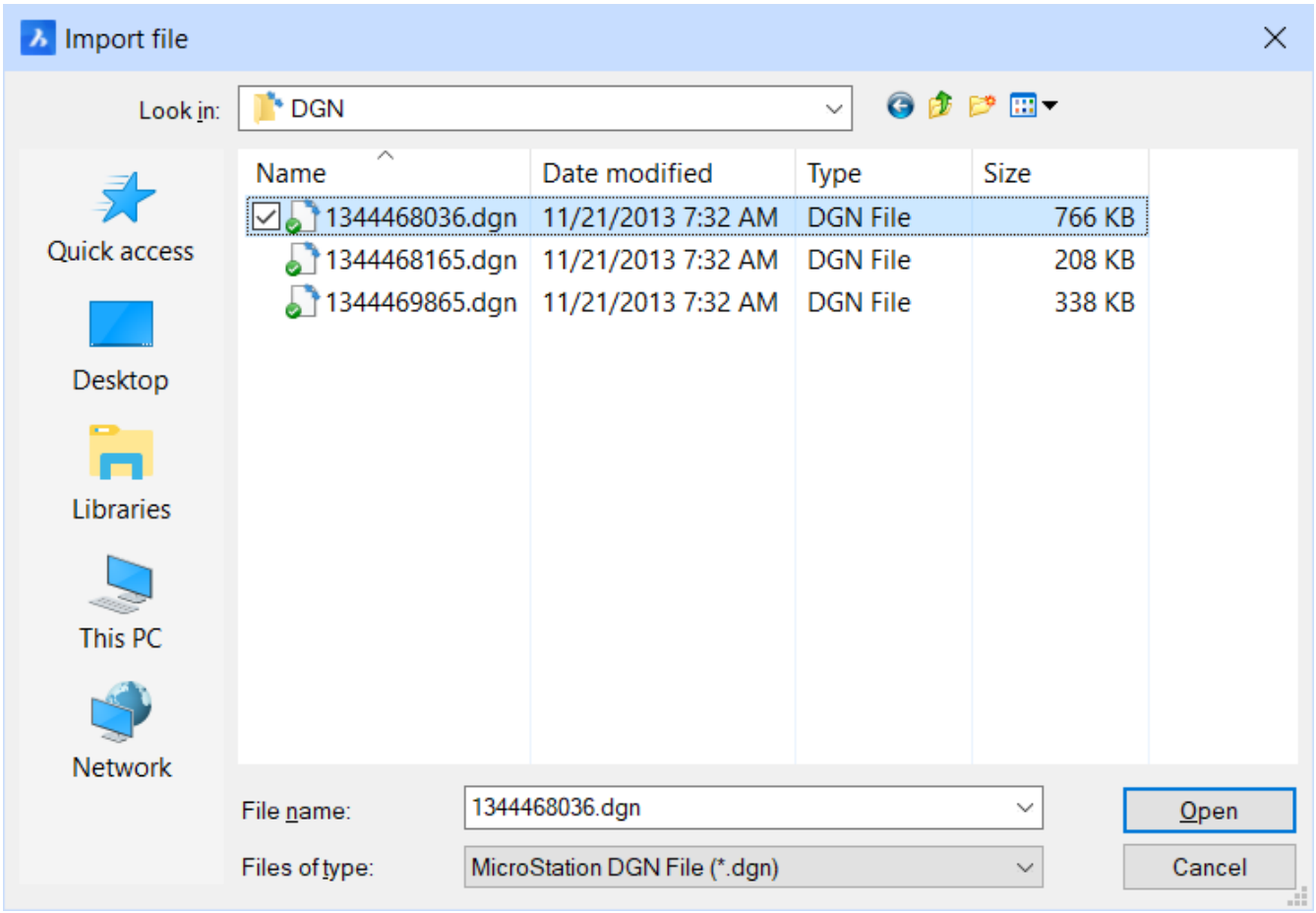
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Imports DGN design files (created by Microstation from Bentley Systems) as entities.

Accessing the Command

Keyboard: **dgnimport** Menu: **File | Import**, and then select DGN format **: DgnImport** Displays the Import dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dgnimport-dlg.png})

1. Choose a DGN file.
2. Click **Open**. BricsCAD converts the Microstation elements into BricsCAD entities using the settings described below.

Figure 6.701



Command Options

Option	Description
File name	Chose a DGN file from the dialog box
Open	Opens the selected DGN file in the current drawing as entities, such as lines and blocks
Cancel	Cancels the operation

Microstation Jargon

Microstation has its own vocabulary:

Microstation Term	BricsCAD Equivalent Term
Element	Entity or object
Shape, complex shape	Polyline
Text node	Empty text field to be filled in later
Design model	Model space
Active model	Current view of the model
Sheet model	Paper space
Cell	Block
Level	Layer
Resource files	Support files, such as fonts and line types
Line style	Linetype

If an element (entity) is filled, then BricsCAD makes a hatch of it. Microstation assigns color numbers to colors that are different from BricsCAD's color numbers. and so BricsCADattempts to matchcolors from the DGN color table with a color in the DWG color table; if no match is found, the DGN color is saved as an RGB value.

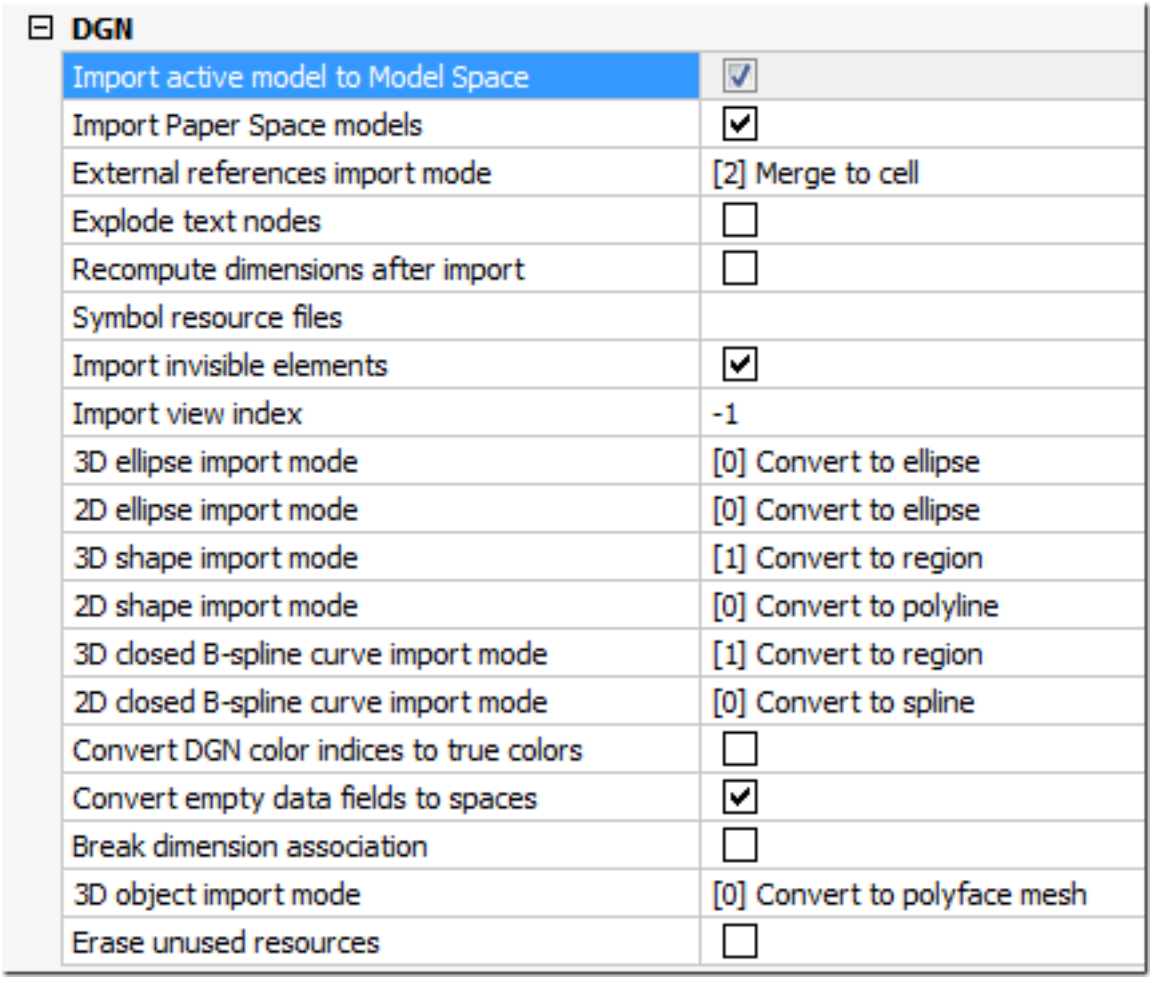
DGN Settings

Settings determine how this command converts design files to DWG format. In the Settings dialog box, search for "DgnImp": (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dgnimport-

DgnImp3dEllipseImportMode	Determines how to convert 3D ellipses: • 0 - convert to ellipse (default) • 1 - convert to region
DgnImp3dObjectImportMode	Determines how to convert 3D elements: • 0 - convert to polyface mesh (default) • 1 - convert to a 3D solid or body Microstation uses the word ”element” for entities or objects.
DgnImp3dShapeImportMode	Determines how to convert 3D shapes and 3D complex shapes: • 0 - convert to polyline • 1 - convert to region (default) • 2 - convert to polyface mesh A Microstation ”shape” and ”complex shape” is like a polyline in BricsCAD.
DgnImpBreakDimensionAssociation	Determines if dimensions loose their associativity upon import: • 0 - associativity is maintained (default) • 1 - dimension associations are broken
DgnImpConvertDgnColorIndicesToTrueColors	Determines how Microstation colors are mapped to BricsCAD colors: • 0 - converts DGN color indices to DWG color indices • 1 - converts DGN color indices to RGB true colors (default) Microstation assigns color numbers to colors that are different from BricsCAD: 1. BricsCAD gets the color from the DGN color table 2. Then it attempts to match it with a color in the DWG color table. 3. If no match is found, the DGN color is saved as an RGB (true color index) value.
DgnImpConvertEmptyDataFieldsToSpaces	Determines how empty field values are handled: • 0 - replaced by underscore symbols (_) • 1 - replaced by space symbols ( ) (default)
DgnImpEraseUnusedResources	Determines how to import unreferenced elements, such as text styles and linetypes: • 0 - import unreferenced elements (default) • 1 - erase unreferenced imported items

DgnImpExplodeTextNodes	Determines how to handle text nodes (empty text fields): • 0 - convert them to multiline text (default) • 1 - convert them to simple entities, such as text and lines Microstation uses text nodes as empty fields that are used to reserve space for text that will be added later, and so the length of the text is not yet known.
DgnImpImportActiveModelToModelSpace	Determines how active spaces are handled: • 0 - import active models to model space first, and then attach the design model from model table • 1 - import the active model to model space (default) Microstation uses the phrase “design model” for model space, and “active model” for the current view of a model.
DgnImpImportInvisibleElements	Determines how invisible elements (entities) are handled: • 0 - skip invisible DGN elements • 1 - import invisible them as invisible entities (default)
DgnImpImportPaperSpaceModels	Determines how to import sheet models (paper space): • 0 - import one DGN model to model space only • 1 - import all DGN sheet models to paper space layouts (default) Microstation uses the phrase “sheet model” for paper space.
DgnImpImportViewIndex	Determines DGN view settings: • 0 through 7 - specifies level mask • -1 - view is not defined Microstation uses the word “level” for layers; a ”mask” hides content in areas or levels/layers.
DgnImpRecomputeDimensionsAfterImport	Determines how to handle dimensions: • 0 - create DGN-style dimension geometry blocks (default) • 1 - re-compute all dimensions to create DWG dimension geometry blocks
DgnImpSymbolResourceFiles	Specifies the paths to folders holding DGN and RSC files Microstation uses RSC resource files to store fonts, line styles, and so on.
DgnImpXRefImportMode	Determines how to import reference attachments: • 0 - don’t import DGN reference attachments • 1 - convert attached DGN files to DWG-style xref files • 2 - create a block definition of the attached DGN file, then create a block reference (default) • 3 - attach the DGN references as underlays Microstation uses the word “cell” for blocks.

Figure 6.702


DGN
Import active model to Model Space
Import Paper Space models
External references import mode
[2] Merge to cell
Explode text nodes
Recompute dimensions after import
Symbol resource files
Import invisible elements
Import view index
-1
3D ellipse import mode
[0] Convert to ellipse
2D ellipse import mode
[0] Convert to ellipse
3D shape import mode
[1] Convert to region
2D shape import mode
[0] Convert to polyline
3D closed B-spline curve import mode
[1] Convert to region
2D closed B-spline curve import mode
[0] Convert to spline
Convert DGN color indices to true colors
Convert empty data fields to spaces
Break dimension association
3D object import mode
[0] Convert to polyface mesh
Erase unused resources

Related Commands

Import (6.17.10) - imports files in a variety of file formats.

Figure 6.703



6.10.29 DimConstraint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

Accessing the Command

command bar: **dimconstraint** menu bar: **Parametric | 2D Constraints |** toolbar: **2D Constraints |** ribbon: **Parametric | 2D Constraints |** : **DimConstraint Prompts you in the command bar:** Select associative dimension to convert or [LInear / Horizontal / Vertical / Aligned / ANgular / Radial / Diameter] <Aligned>: **(Select an associative dimension, or else choose an option to place a dimensional constraint)**

Command Options

Option	Description
Select associative dimension to convert	Select an associative dimension. The dimension is converted to the dimensional constraint of the same type. This option is equivalent to the DcConvert (6.10.7) command ( ).
Linear	Constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. This option is equivalent to the DcLinear (6.10.39) command ( ).
Vertical	Constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. This option is equivalent to the DcVertical (6.10.12) command ( ).
Horizontal	Constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. This option is equivalent to the DcHorizontal (6.10.10) command ( ).
Aligned	Constrains the distance between two points. This option is equivalent to the DcAligned (6.10.5) command ( ).
Angular	Constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. This option is equivalent to the DcAngular (6.10.6) command ( ).
Radius	Constrains the radius of a circle or an arc. This option is equivalent to the DcRadius (6.10.11) command ( ).
Diameter	Constrains the diameter of a circle or an arc. This option is equivalent to the DcDiameter (6.10.8) command ( ).

Procedures

- To edit dimensional constraints (1.15.6)
- Using expressions to define dimensional constraints (1.15.6)

Related Commands

Geomconstraint (6.15.16) - applies geometric relationships between entities and/or valid constraint points on entities. DcAligned (6.10.5) - constrains the distance between two points. DcAngular (6.10.6) - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. DcDiameter (6.10.8) - constrains the diameter of a circle or an arc. DcHorizontal (6.10.10) - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. DcLinear (6.10.39) - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. DcRadius (6.10.11) - constrains the radius of a circle or an arc. DcVertical (6.10.12) - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. CleanUnusedVariables (6.9.23) - purges variables not used in constraint expressions and not linked to dimensions. -Parameters (6.24.5) - manages 2D dimensional constraint parameters and user variables in the current drawing.

Figure 6.704



6.10.30 Dish

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) This command is obsolete, and exists for backward compatibility only.Please instead use the**Ai_Dish (6.5.7)**command.

Figure 6.705



6.10.31 Dist

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Reports the distance and angle between two points (short for distance).

Accessing the Command





Command bar: **dist** transparent: **'dist** alias: **di** menu bar: **Tools | Inquiry | Distance** toolbar: **Inquiry |** ribbon: **Tools | Inquiry | Distance** : **Dist Prompts you in the command bar:** Starting point for distance: **(Pick a point, or enter coordinates)** End point: **(Pick another point, or also enter coordinates)** Sample response: Distance =

Figure 6.706

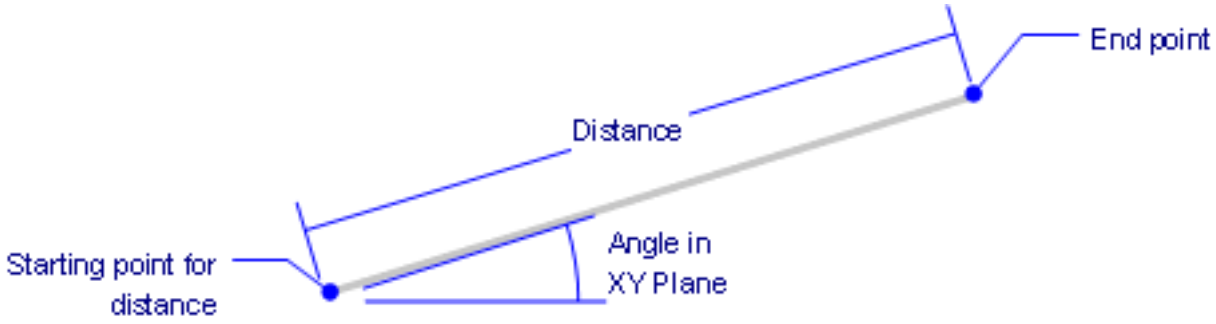
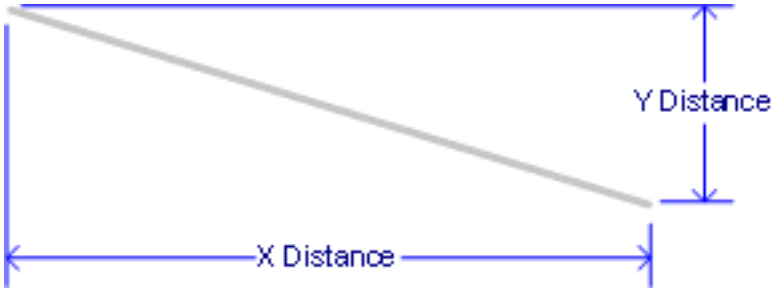


Figure 6.707



Command Options

Option	Description
Starting Point for Distance	Specifies the first point from which the measurement takes place
End Point	Specifies the other point

Related Commands

Area (6.4.13) - reports the area, circumference, and perimeter of entities. Id (6.17.1) - reports the x,y,z coordinates of picked points. List (6.20.35) - lists the area and other parameters of entities in the command bar. MassProp (6.21.4) - reports the properties of solid models and regions. Properties (6.24.49) - lists the area and other parameters of entities in a palette.

Figure 6.708



6.10.32 Distantlight

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Places distant lights (displayed as glyphs) in drawings for use with rendering. Distant lights represent sources that are far off, such as the sun, 

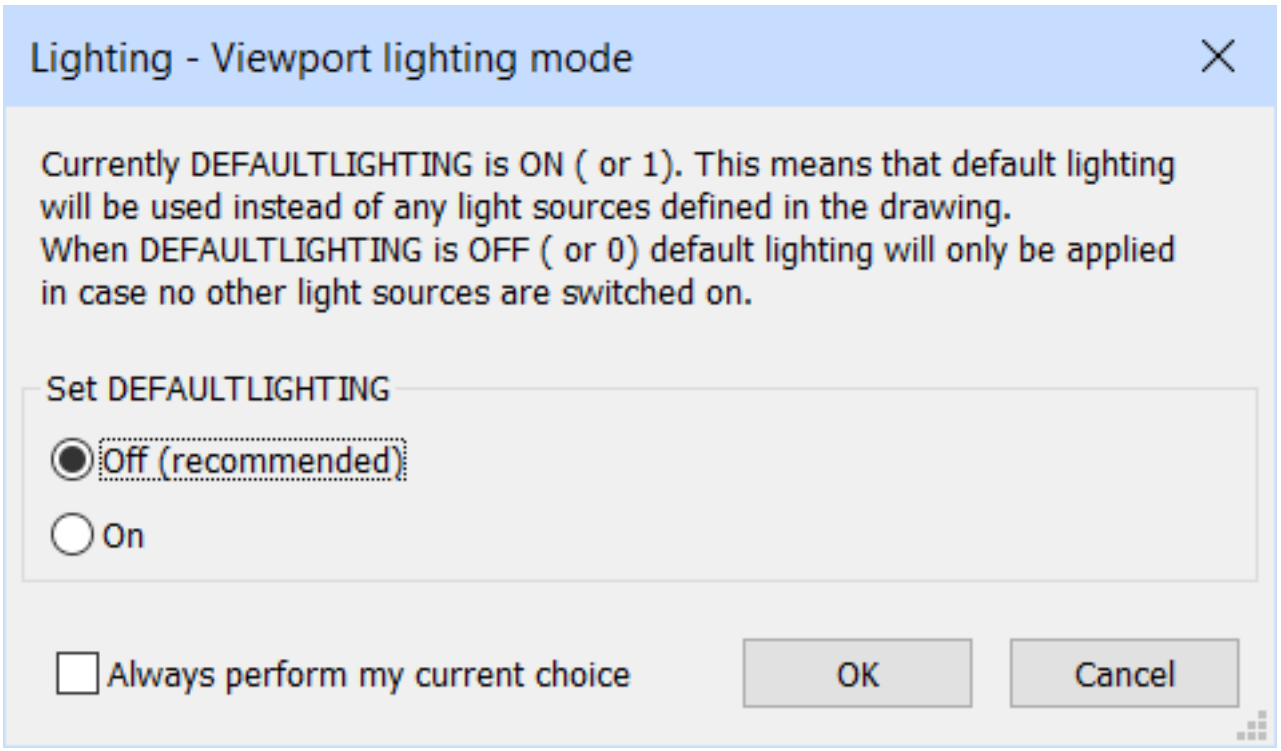
Accessing the Command

 command bar: **distantlight** menu bar: **View | Rendering | Lights | New distant light** toolbar: **Render |Lights |**  ribbon:**View | Lights | Create | Distant Light** : **Distantlight** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/distantlight-dlg.png}) To be able to use distant lights in drawings, you must turn off default lighting. (Default lighting is the light sources the program uses when you have defined no lights yet.)

1. Choose **Off**
2. Click **OK**

Prompts you in the command bar: Specify light direction or [Vector] <0,0,0>: **(Picka point, enter x,y,z coordinates, press Enter to accept the default, or else type V)** Specify light direction TO <1,1,1>: **(Specify a second point, enter x,y,z coordinates, or else press Enter to accept the default)** Enter an option to change [Name/Intensity factor/ Status/ Photometry/ shadoW/ filterColor/eXit] <eXit>: **(Enter an option, or else press Enter to exit the command)**

Figure 6.709



Command Options

Option	**Description**
Name	Specifies a name for the light; prompts you: **Enter light name <Distantlight1>** - **(enter a name, or else press Enter to accept the default name)**
Intensity Factor	Specifies the intensity of the light; prompts you: **Enter intensity (0.00 - max float) <1.0>** - **(enter an intensity value of between 0.00 and the maximum supported by your system)**
Status	Toggle the light; prompts you: **Enter status [ON/OFf] <ON>** - **(choose ON to include the light when calculating renderings)**

Photometry Options	**Description**
Photometric	The Photometric option defines intensity and color characteristics of the light; prompts you: **Enter a photometric option to change [Intensity/Color/eXit] <Intensity>** - **(enter an option.)**

Intensity

The Intensity option defines the brightness of the light by Cd, flux, or illuminance; prompts you: **Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0>** - **(enter an option:)**

- **Enter intensity (Cd)** - specifies the intensity expressed in Candela (Cd) The candela (symbol: cd) is the SI base unit of luminous intensity; that is, power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function). A common candle emits light with a luminous intensity of roughly one candela
- **Enter Flux (Lm)** - specifies the luminous flux expressed in Lumen (Lm) The lumen (symbol: Lm) is the SI unit of luminous flux, a measure of the power of light perceived by the human eye. Luminous flux differs from radiant flux in that luminous flux measurements (such as lumens) are intended to reflect the varying sensitivity of the human eye to different wavelengths of light, while radiant flux measurements (such as watts) indicate the total power of light emitted. The lumen is defined in relation to the candela as: 1 lm = 1 cd·sr
- **Enter Illuminance (Lx or Fc) or enter an option [Distance] <9424.8>**- **(specify the illuminance expressed in Lx (lux) or Fc (foot-candle) units) Distance** - **(enter a distance expressed in drawing units)** In photometry, illuminance is the total luminous flux incident on a surface, per unit area. It is a measure of the intensity of the incident light, wavelength-weighted by the luminosity function to correlate with human The foot-candle (Fc) is a non-metric unit of illuminance that is used in photography. The unit is defined as the amount of illumination the inside surface of a 1-foot radius sphere would be receiving if there were a uniform point source of one candela in the exact center of the sphere. Alternatively, it can be defined as the illuminance on a 1-square foot surface of which there is a uniformly distributed flux of one lumen. This can be thought of as the amount of light that actually falls on a given surface. The foot-candle is equal to one lumen per square foot. One foot-candle is equal to approximately 10.764 lux, although in the lighting industry, typically this is approximated as 1 foot-candle being equal to 10 lux. As a full sphere has a solid angle of 4·pi steradians, a light source that uniformly radiates one candela in all directions has a total luminous flux of 1 cd·4pi sr = 4pi ≈ 12.57 lumens. For example, if a candle emits light with a luminous intensity of about 1 candela, the total light emitted is about 12.6 lumens.

The Color option determines the color of the light by light type of degrees Kelvin; prompts you: **Enter color name or enter an option [?/Kelvin] <D65>** - **(enter an option:)**

- **Enter color name** - **(specifies the name of a color of light, as listed below)**
- **?** prompts you "**Enter color name(s) to list <*>**" - **(press Enter to see the valid names of light colors:)**

D65 Fluorescent Coolwhite Whitefluorescent daylightfluorescenT Incandescent Xenon Halogen Quartz Metalhalide mErcury Phosphormercury highpressureSodium Lowpressuresodium

- **Enter Kelvin temperature <3600.0>** - **(enter a color temperature expressed in degrees Kelvin (K))** Color temperature is a characteristic of visible light that has important applications in lighting, photography, videography, publishing, manufacturing, astrophysics, and other fields. The color temperature of a light source is the temperature of an ideal black-body radiator that radiates light of comparable hue to that light source. The temperature is conventionally stated in units of absolute temperature: degrees kelvin (K). Higher color temperatures (5,000 K or more) are called cool colors (blueish white); lower color temperatures (2,700 – 3,000 K) are called warm colors (yellowish white through red).

eXit

Exits this sub-level of command options, and returns to the previous level; prompts you: **Enter a photometric option to change [Intensity/Color/eXit] <Intensity>:(Enter x to exit photometric options) Enter an option to change [Name/Intensity factor/Status/Photometry/shadoW/filterColor/eXit] <eXit>:**

Shadow Options	**Description**
shadoW	Determines the look of shadows cast by lights; prompts you: **Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled]: <Sharp>**- **(enter an option)**
Off	Disables the calculation of shadows; this light will cast no shadows, which helps speed up renderings. The command prompt returns to the previous level.
Sharp	Displays shadows with sharp edges; this option renders lights with shadows more quickly than lights with soft (diffuse) shadows. The prompt returns to the previous level.

soFt mapped

Displays realistic shadows with soft edges.Prompts you:

1. **Enter map size [64/128/256/512/1024/2048/4096] <256>(enter a value between 64 and 256.)** The map size determines how diffuse the shadow is at its edges. A larger number is more diffuse (has softer edges) but takes longer to render.
2. **Enter softness (1-10) <1>** - **(enter a value between 1 and 10.)** The software determines the softness of the body of the shadow. A larger number is software, but takes longer to render.

soft sAmpled

Displays realistic shadows with shadows based on extensive options.Prompts you: **Enter an option to change Shape/sAmples/Visible/<eXit>** - **(enter an option:) Shape** - specifies the shape of the shadow; prompts you:

- **Enter shape [Disk/Rect] <Disk>** - **(enter d or r)**

When the shadow has a disk shape, prompts you: **Enter radius <0.0>** - **(enter the radius of the disk)** When the shadow has a rectangular shape, prompts you: **Enter Length <0.0>** - **(enter length of the rectangle) Enter Width <0.0>** - **(enter width of the rectangle) sAmples** - specifies the number of samples to take for the shadow:

- **Enter Shadow Sample <16>** - **(enter a value between0 and 2147483647)**; a larger samples slows down the rendering speed, but generates a more realistic-looking shadow

Visible - specifies the visibility of the disk or rectangular shadow shape:

- **Enter Shape Visibility [Yes/No] <No>** - (**enter y or n.)**

eXit -exits this sub-level of command options, and returns to the previous level.

filterColor	Specifies the color of the light based on RGB, ACI, or HSL systems; prompts you: **Enter true color (R,G,B) or enter an option [Index color/Hsl]:<255,255,255>** - **(enter an option)** • **True Color (RGB)** - **(enter three integers in the range of 0 to 255, where each number refers to the amount of red, green, and blue, such as 0,0,0 (black), 255,0,0 (red), 255,0,255 (purple), 128,128,128 (gray), or 255, 255,255 (white).Refer to the Color (6.9.33) command's dialog box for the meaning of the RGB numbers.)** • **Index color** - specifies the CAD Color Index color; prompts you: **Enter color name or number (1-255)** - **(enter a value between 1 and 255; refer to the Color (6.9.33) command's dialog box for the meaning of the color numbers.)** • **HSL** - specifies the color using three parameters that determine the amount of hue (color), saturation (strength of color), and lightness (brightness or darkness of color. **Enter HSL number (0,0,24)** - **(enter three values between 1 and 255; refer to the Color (6.9.33) command's dialog box for the meaning of the color numbers.)**
eXit	Exits the the command and places the light in the drawing.

To edit the properties of the light, double-click it.

Related Commands

Light (6.20.28) - creates a new light in the drawing. Lightlist (6.20.29) - displays the Drawing Explorer - Lights dialog window. Pointlight (6.24.42) - creates a point light. Render (6.26.22) - calculates a photorealistic image of the 3D model. Spotlight (6.27.42) - creates a spot light. Sunproperties (6.27.57) - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog. Weblight (6.33.6) - creates a web light.

Figure 6.710



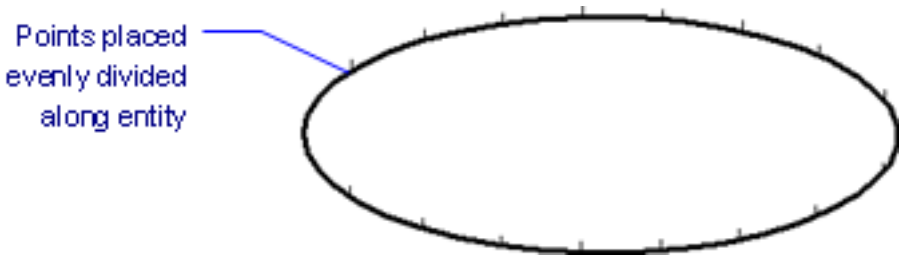
6.10.33 Divide

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places a specific number of points or blocks spaced equally along an entity.

Accessing the Command

 command bar: **divide** alias: **div** menu bar: **Modify | Divide** toolbar: **Modify |**  ribbon:**Draw | Draw | Points | Divide** : **Divide Prompts you in the command bar:** Select entity to divide: **(Choose a line, polyline, spline, arc, circle, or ellipse)** Number of segments [insert Block]: **(Enter a number, or type B)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image157.png})

Figure 6.711



Command Options

Option	Description
Select entity to divide	Choose a single entity; this command works with the following entities: • Line • Polyline • Spline • Arc • Circle • Ellipse
Number of segments	Specifies the number of ”segments,” which means equally-divided distances between points. Prompts you: • **Number of segments** - **(type an integer between 2 to 32767)**; the larger the number, the shorter the segments over the length of the entity Entering **2**for the number of segments places a single point or block at the midpoint of the entity, as the entity is divided into two segments. There is always one more segment than points placed, as no point or block is placed at the start or endpoints of the entity. Points on an entity are essentially invisible, so you can use the**DdPType (6.10.18)** commandto change their look and make them larger. After points are placed on the entity, you can use the **NODe** entity snap to attach other entities to them
Blocks	Places equally-spaced blocks along the entity, instead of points.Prompts you: 1. **Name of block to insert [? to list]** - **(type the name of the block, or else press ? to list the available blocks.)** 2. **Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-Align blocks>** - **(enter an option)**: • **Yes** - rotates blocks to match the local orientation of the entity, such as along an arc • **No** - does not rotate blocks, so that they maintain their orientation **Number of segments** - **(type an integerbetween 2 to 32767)**

Procedures

- To divide an entity using points (1.23.40)
- To divide an entity using blocks (1.23.40)

Related Commands

Measure (6.21.13) - places equi-distant points along entities. Point (6.24.31) - places points in drawings. DdPType (6.10.18) - specifies the look and size of point entities through the Settings dialog box.

Figure 6.712



6.10.34 Dome

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) This command is obsolete, and still exists in the program for backward compatibility only.Please use **Ai_Dome (6.5.8)** instead.

Figure 6.713



6.10.35 Donut

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws wide, filled polylines that look like donuts.

Accessing the Command



command bar: **donut** alias: **do, doughnut** menu bar: **Draw | Donut** toolbar: **Draw | Polygons |** ribbon:**Draw | Draw | Polygons | Donut** : **Donut Prompts you in the command bar:** Inside diameter of donut [2 Point/3 Point/Tangent Tangent Radius] <0.5>: **(Pick a point, enter a value, or enter an option)** Outside diameter of donut <1>: **(Specify a value, or pick two points)** Center of donut: **(Specify coordinates, or pick a point in the drawing)** Center of donut: **(Specify coordinates to draw another donut or pick a point in the drawing, or else press Enter to conclude the command)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image209.png} Donuts are made of two wide polyline arcs. This command was created originally to draw solder pads on printed circuit boards.

Command Options

Option	Description
2Point (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image207.png})	Defines the donut by two points on its circumference; prompts you: • **Width of donut** - **(enter a value, or pick two points)** • **First point on diameter** - **(enter coordinates, or pick a point)** • **Second point on diameter** - **(enter coordinates, or pick another point)** The abbreviation for this option is 2P.
3Point (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image208.png})	Defines the donut by three points on its circumference; prompts you: 1. **Width of donut** - **(enter a value, or pick two points)** 2. **First point on donut** - **(enter coordinates, or pick a point)** 3. **Second point** - **(enter coordinates, or pick another point)** 4. **Third point** - **(enter coordinates, or pick another point)** The abbreviation for this option is 3P .

Tangent Tangent Radius (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image210.jpg})	Defines the donut by two points tangent to other entities and a radius. It prompts you: 1. **Width of donut**- **(enter a value, or pick two points)** 2. **Diameter of donut** - **(enter a value, or pick two points)** 3. **Select first tangent point** - **(enter coordinates, or pick an entity)** 4. **Select second tangent point** - **(enter coordinates, or pick another entity)** The abbreviation for this option is R, and not TTR.
Diameter of donut (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image209.jpg})	Defines the donut by inner and outer diameter, and a center point. It prompts you: 1. **Outside diameter of donut** - **(specify a value, or pick two points)** 2. **Center of donut** - **(specify coordinates, or pick a point in the drawing)**

Procedures

Drawing donuts (1.17.22)

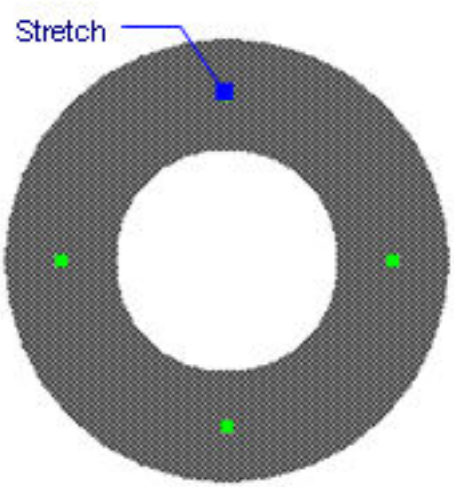
Grips Editing

Donuts can be edited directly through grips:

1. Select the donut. Notice that it has four grips.
2. Drag a grip to stretch the donut (increase or decrease its diameter).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image469.jpg})

Figure 6.714



Related Commands

Circle (6.9.20) - draws regular circles. PLine (6.24.27) - draws polyline segments and arcs. Rectang (6.26.6) - draws rectangles out of polylines. PEdit (6.24.20) - edits polylines. 3dPoly (6.3.10) - draws polylines with x,y,z coordinates. Polygon (6.24.43) - draws regular polygons out of polylines.

Figure 6.715

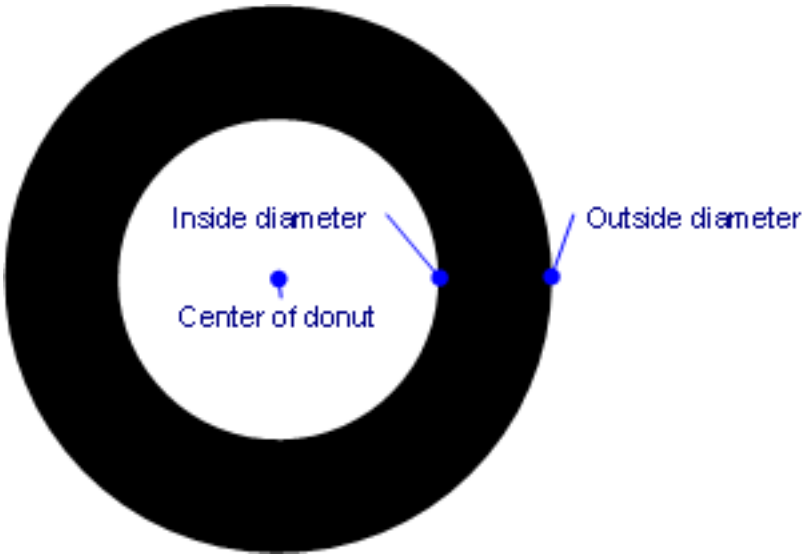


Figure 6.716

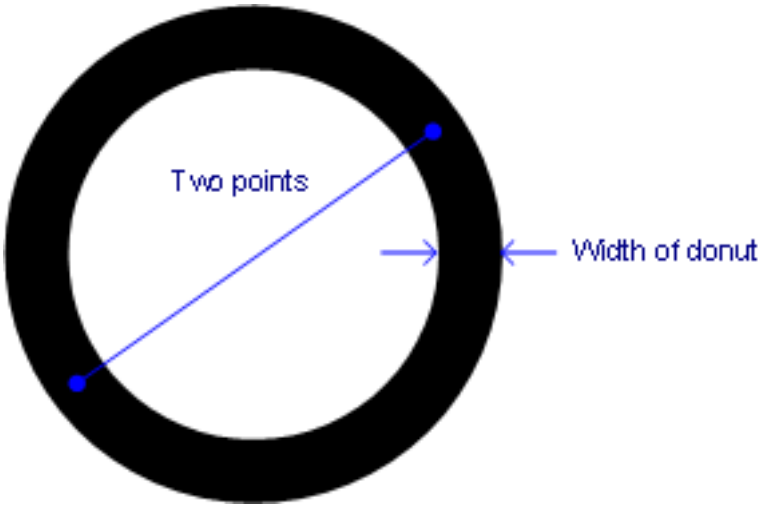


Figure 6.717

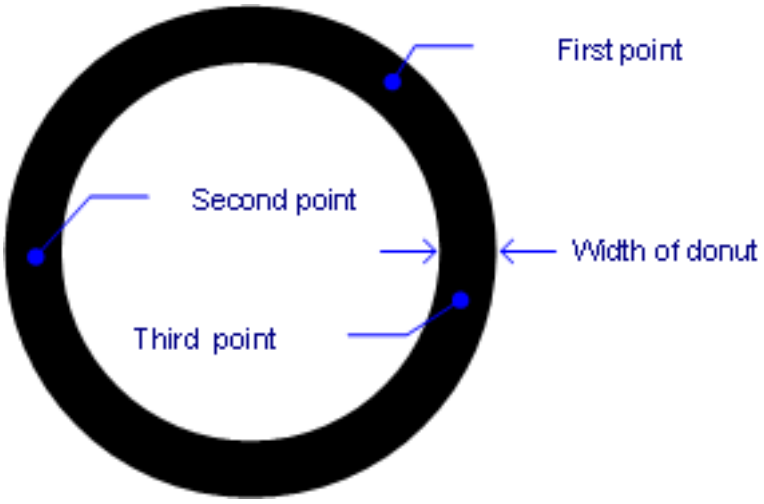


Figure 6.718

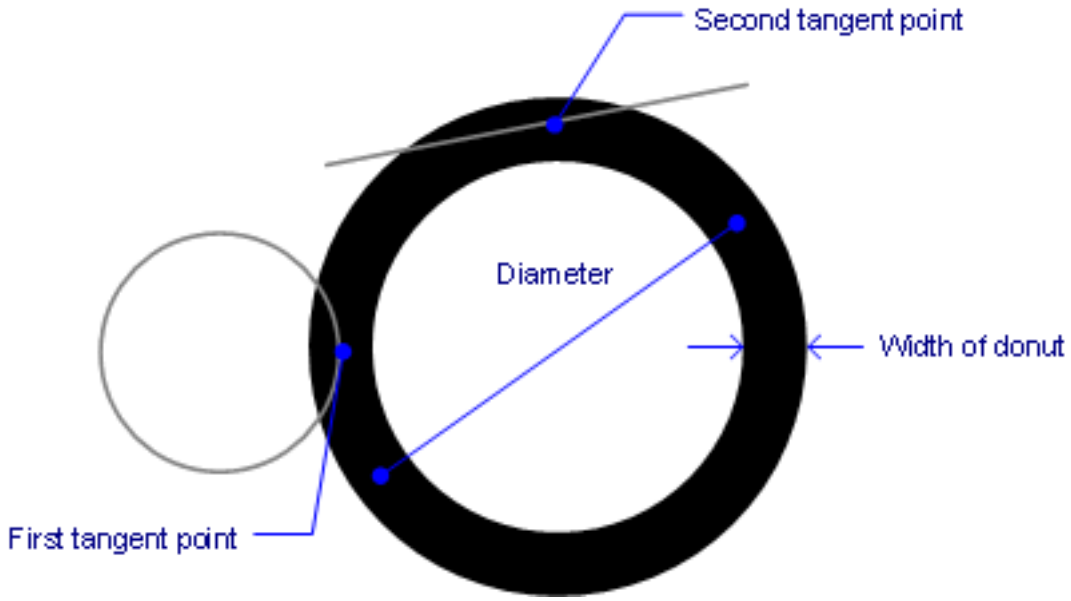


Figure 6.719

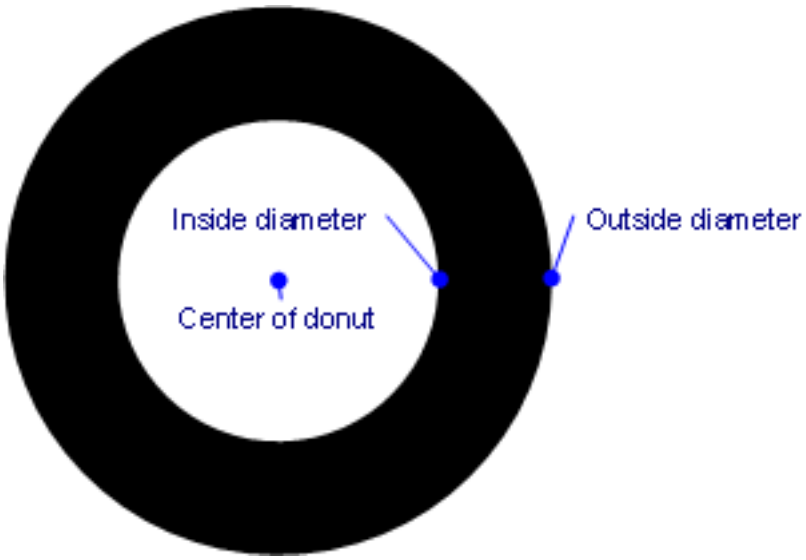


Figure 6.720



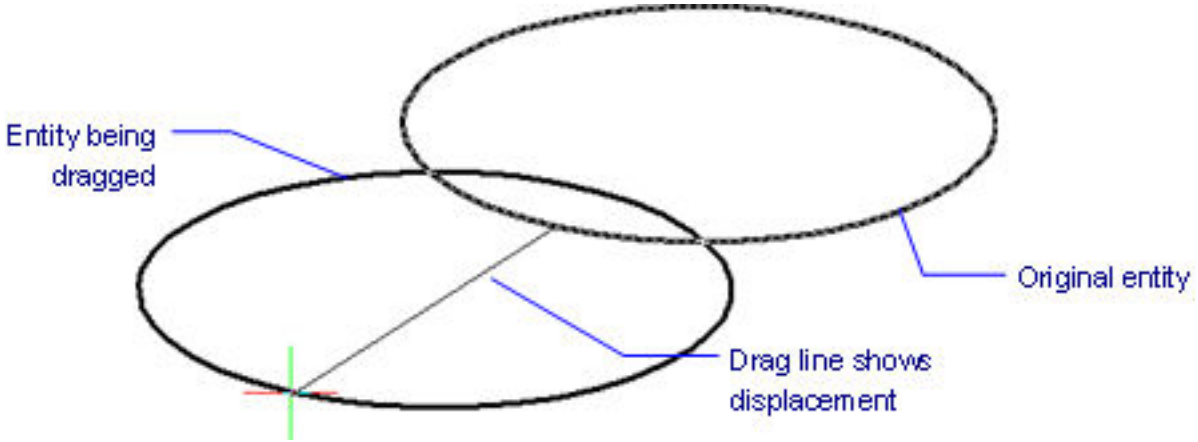
6.10.36 DragMode

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Controls the appearance of objects while being dragged.The command is no longer necessary, and is kept only for compatibility. This command was useful in the early days of CAD when computers of the 1980s and early 1990s were so slow at processing vector graphics that they had difficulty updating the screen quickly enough to show objects being dragged.

Accessing the Command

command bar: **dragmode** transparent: **'dragmode : DragMode Prompts you in the command bar:** [ON/OFf/Auto] <Auto>: **(Enter an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image138.

Figure 6.721



Command Options

Option	Description
ON	Turns on dragmode; same as Auto mode.
OFF	Turns off dragmode; dragged objects and the dragline are never shown on the screen.
Auto	Turns on drag mode; dragged objects are always shown.

Related Command

Settings (6.27.23) - sets drag mode through the Drawing Display section of the Settings dialog box.

Figure 6.722



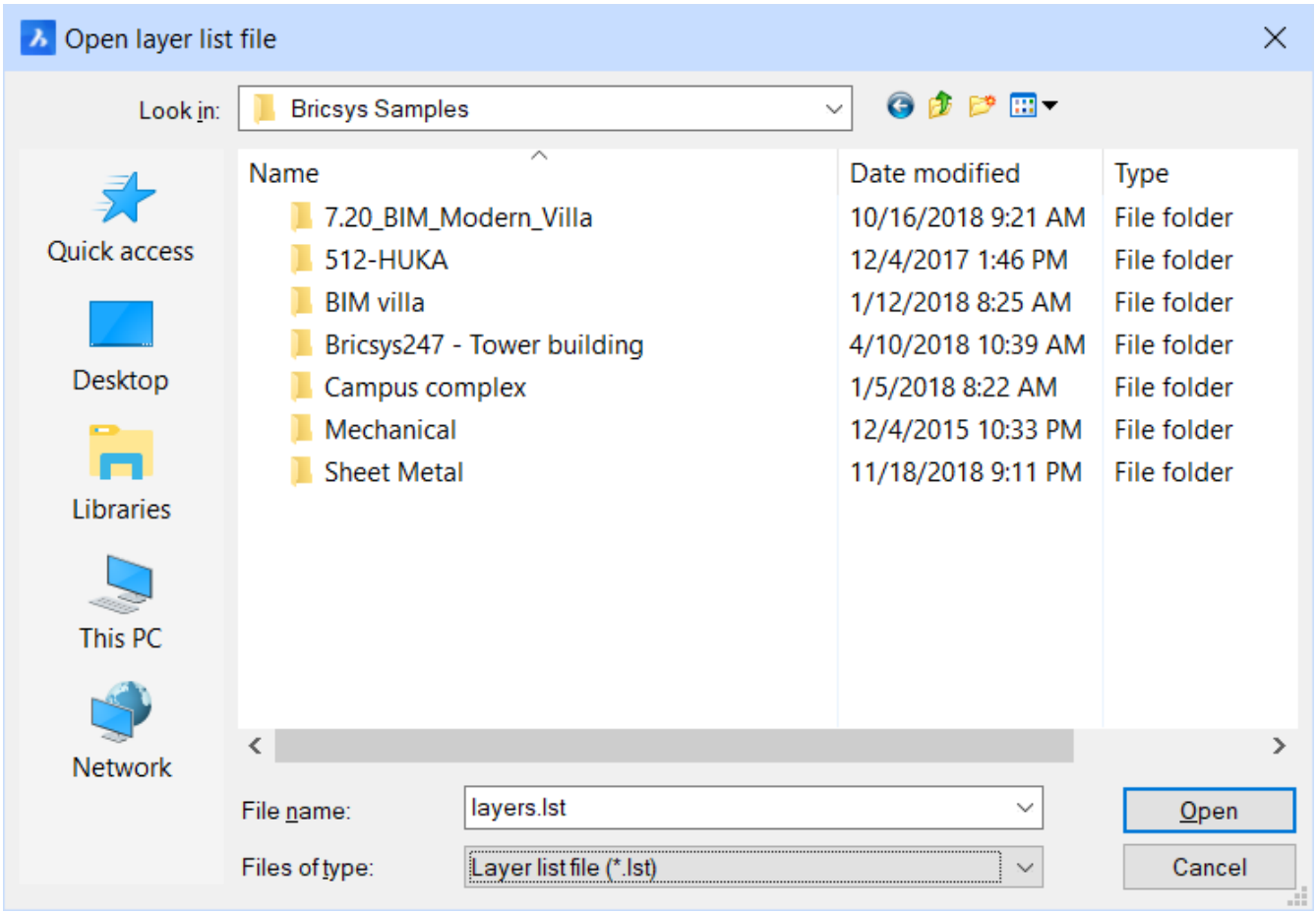
6.10.37 DrawOrderByLayer

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Controls the display order of overlapping objects through a list of layer names in a LST file.

Accessing the Command

command bar: **draworderbylayer** : **DrawOrderByLayer** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsy file-dlg.png}) Choose a *.lst layer list file, and then click **Open**. The *.lst file is an ASCII text file that you (or someone else) wrote with a text editor, such as Notepad, and holds a list of layer names, one per line, such as LayerName1 LayerName2 ... LayerNameN The command displays entities on layers in the order defined by the *.lst file. The display order is the reverse of the draw order: first the entities on layer LayerName1 are drawn, then the entities on LayerName2 are drawn on top of the first ones, and so on. The entities on LayerNameN end up on top. Wildcard characters (6.20.2) are supported in layer names. All entities on layers that match the selection criteria are drawn at the same display level. Caution: A specific filter preceded by a general filter does nothing, such as filter at* listed after filter a* has no effect because all entities on layers starting with 'a' are already drawn. But filter a* listed after *at first draws allentities on layers starting with 'at', and then all entities on the remaining layers starting with 'a'.

Figure 6.723



Command Options

Option	Description
Open	Opens the *.lst file.

Related Commands

DrawOrder (6.10.38) - sets draw order of overlapping objects through the command bar. BHatch (6.6.5) and Hatch (6.16.1) - set the draw order of hatch patterns relative to other entities through a dialog box. -BHatch (6.6.6) and -Hatch (6.16.2) - set the draw order of hatch patterns relative to other entities through the command bar.

Figure 6.724



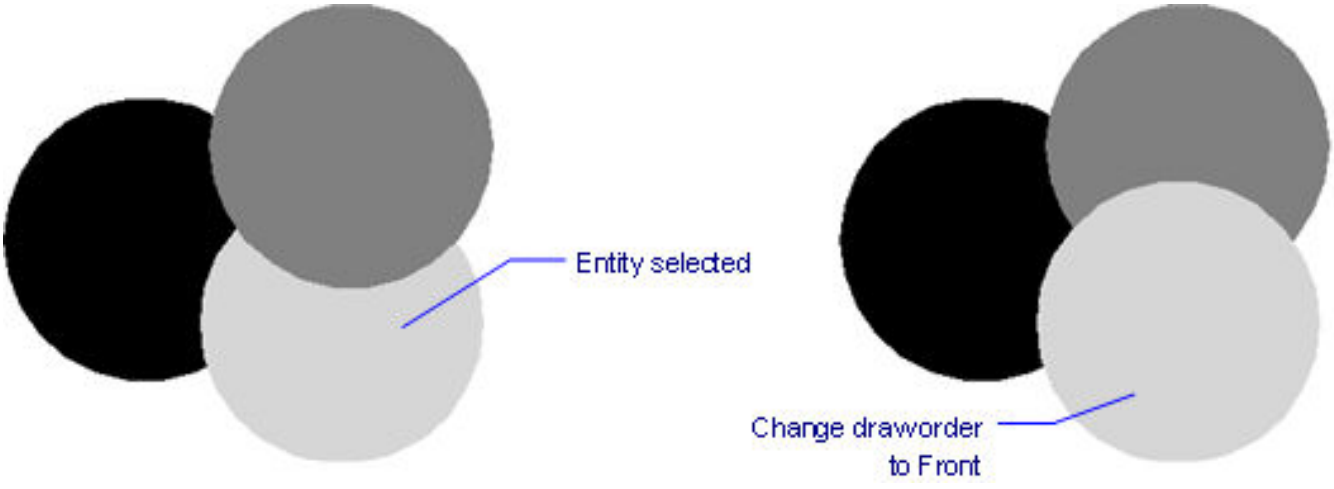
6.10.38 DrawOrder

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Changes the display order of overlapping entities. Use this command to displayed entities over top other entities that would otherwise overlap or even hide them.Hatches and text entities have handy commands that controlling their display order:**HatchToBack (6.16.6)** places all hatching below all other entities, and **TextToFront (6.29.14)** displays all text above them all entities.

Accessing the Command

command bar: **draworder** alias: **dr** menu bar: **Tools | Draw Order** toolbar: **Draw Order |** ribbon: **Change | Arrange | Draw Order |** **: DrawOrder Prompts you in the command bar:** Select entities to change draw order: **(Select one or more entities; when entities overlap, press the Tab key to select the one you want)** Entities in set: 1 Select entities to change draw order: **(Select one or more entities, or press Enter)** Change draw order [Above/Under/Clear all orders/bring to Front/send to Back] <Send to back>: **(Enter an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image139.jpg})

Figure 6.725



Command Options

Option	Description
Select Entities to Change Draw Order	Selects one or more entities. This command is meant to work with overlapping objects. While this command works with non-overlapping entities, it has no effect on them.
Above	Visually moves the display of selected entities "one level" above other entities; this option does not necessarily move them to the top (front) of display order.
Under	Moves the entities below other entities, but not necessarily to the bottom (back) of display order.
Clear all orders	Clears assigned display orders; entities are displayed as they were originally created. This means that the last created entity is displayed on top of one created earlier.
Bring to Front	Moves the selected entities' display order above all other overlapping entities.
Send to Back	Moves the selected entities' display order below all other overlapping entities.

Procedures

- Adjusting the Draw Order (1.23.9)

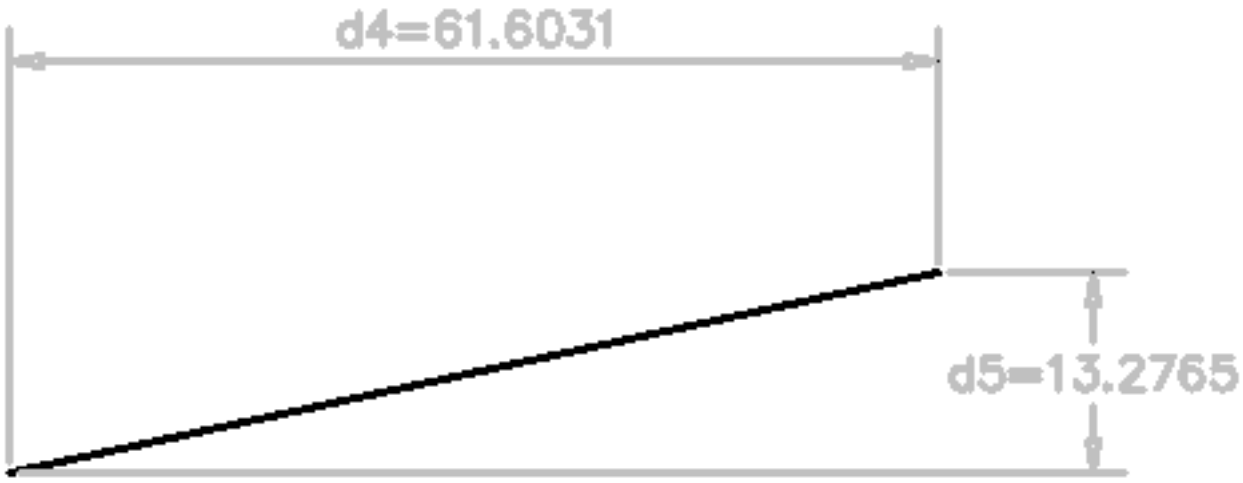
Related Commands

DrawOrderByLayer (6.10.37) - sets draw order of overlapping objects through layer names. HatchToBack (6.16.6) - sets the draw order of all hatch entities in the drawing to display behind all other entities. TextToFront (6.29.14) - sets the draw order of all texts and dimensions in the drawing to display in front of all other entities. BHatch (6.6.5) and Hatch (6.16.1) - set the draw order of hatch patterns relative to other entities through a dialog box.

Figure 6.726



Figure 6.727



6.10.39 DcLinear

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains the distance between two points, or the length of a single entity, to the horizontal (X direction) or vertical distance (Y direction), with respect to the current coordinate system. (This command is equivalent to the Linear (6.10.29) option of the DIMCONSTRAINT (6.10.29) command.) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dclinear-eg.png})

Accessing the Command

 command bar: **dclinear** menu bar: **Parametric | 2D Constraints | Linear** toolbar: **2D Constraints |**  ribbon:**Parametric | 2D Constraints | Linear** : **dclinear Prompts you in the command bar:** Specify first constraint point or [Entity] <Entity>: **(Select a valid constraint point (6.15.16) or, else press Enter to select an Entity)** Specify second constraint point: **(Select another valid constraint point (6.15.16))** Specify dimension line location: **(Move the cursor horizontally or vertically, and then pick a point to position the dimension line)** Dimension text <current>: **(Type the dimension text, or else press Enter to accept the current dimension text)** The horizontal or vertical distance (parallel to the X- or Y-axis of the current coordinate system) between the two points is constrained.

Command Options

Option	Description

Entity	Constrains the horizontal or vertical distance between the endpoints of an entity.Prompts you: 1. **Select an entity** - **(select a line, arc or polyline segment)** 2. **Specify dimension line location** - **(Move the cursor horizontally or vertically, and then pick a point to position the dimension line)** 3. **Dimension text <xx.xx>** - **(type a value, or ekse press Enter to accept the current value)** The entity is constrained horizontally or vertically (parallel to the X- or Y-axis of the current coordinate system), but can continue change its size in other directions.

Procedures

- To edit dimensional constraints (1.15.6)
- Using expressions to define dimensional constraints (1.15.6)

Related Commands

DimConstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GeomConstraint (6.15.16) - applies geometric relationships between entities and/or valid constraint points on entities. DcAligned (6.10.5) - constrains the distance between two points. DcAngular (6.10.6) - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. DcConvert (6.10.7) - converts an associative dimension to a dimensional constraint. DcDiameter (6.10.8) - constrains the diameter of a circle or an arc. DcHorizontal (6.10.10) - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. DcRadius (6.10.11) - constrains the radius of a circle or an arc. DcVertical (6.10.12) - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. DcDisplay (6.10.9) - allows to set dimensional constraints to be displayed or hidden. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity. CleanUnusedVariables (6.9.23) - purges variables not used in constraint expressions and not linked to dimensions. -Parameters (6.24.5) - manages 2D dimensional constraint parameters and user variables in the current drawing.

Figure 6.728



6.10.40 DwgCodePage

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Changes the language code-page for text in drawings.

Accessing the Command

command bar: **dwgcodepage** transparent: **'dwgcodepage** : **DwgCodePage Prompts you in the command bar:** This command sets DWGCODEPAGE to a new value. Characters will not be converted to unicode. ?=List/New code page <ANSI_1252> : **(Enter a code page number, or type ?)** Code pages included with BricsCAD: For Unix: 0: undefined 1: ascii $20-$7F,%%c,%%d,%%p Only 2: iso8859_1 Western Europe (Unix) $20-$FF,%%c,%%d,%%p 3: iso8859_2 Central Europe (Unix) 4: iso8859_3 Eastern Europe (Unix) 5: iso8859_4 Baltic (Unix) 6: iso8859_5 Cyrillic (Unix) 7: iso8859_6 Arabic (Unix) 8: iso8859_7 Greek (Unix) 9: iso8859_8 Hebrew (Unix) 10: iso8859_9 Turkish (Unix) For DOS/Mac: 11: dos437 DOS USA 12: dos850 DOS Western Europe 13: dos852 DOS Eastern Europe 14: dos855 IBM Russian 15: dos857 IBM Turkish 16: dos860 DOS Portuguese 17: dos861 DOS Icelandic 18: dos863 DOS Canadian French 19: dos864 DOS Arabic 20: dos865 DOS Norwegian 21: dos869 DOS Greek 22: dos932 DOS Japanese 23: mac-roman Mac 24: big5 DOS Traditional Chinese 25: ksc5601 Korean Wansung 26: johab Korean Johab 27: dos866 DOS Russian 31: gb2312 DOS Simplified Chinese For Windows: 28: ansi_1250 Win Eastern Europe 29: ansi_1251 Win Russian 30: ansi_1252 Win Western Europe (ANSI) 32: ansi_1253 Win Greek 33: ansi_1254 Win Turkish 34: ansi_1255 Win Hebrew 35: ansi_1256 Win Arabic 36: ansi_1257 Win Baltic 37: ansi_874 Win Thai 38: ansi_932 Win Japanese 39: ansi_936 Win Simplified Chinese GB 40: ansi_949 Win Korean Wansung 41: ansi_950 Win Tradissional Chinese big5 42: ansi_1361 Win Korean Johab 43: ansi_1200 Unicode (reserved) --: ansi_1258 Win Vietnamese (reserved)

Command Options

Option	Description
? = List	Lists code pages available in the program.
New Code Page	Specifies the number of another code page.

Figure 6.729



6.10.41 DwgCompare

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Compares a selected drawing with the current drawing, showing the differences between the two.

Accessing the Command

command bar: **dwgcompare** menu bar: **Tools | DWG Compare** toolbar: **Compare |** : **DwgCompare**
Prompts you in the command bar: CMPDIFFLIMIT = 10000000. Select the file to compare with or [Limit]: **(Press Enter to open a File dialog box or enter the Limit option)** A file dialog box is displayed: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dwgcompare-file-dwg.png}) Select a drawing with which to compare the current drawing, then click the **Open** button. The command looks for differences in the same entities between the current drawings and the drawing opened for comparison. When no differences are found, the program reports: **No differences found.** When the program finds differences, entities are shown in these colors:

- Entities added - green (color is set by variableCMPCLRNEW)
- Entities removed - red(set by variableCMPCLRMISS)
- Entities changed in the original drawing - gray(by variableCMPCLRMOD1)
- Entities changed in the comparison drawing - yellow(variableCMPCLRMOD2)

The colors are set by the Settings dialog box; search for variables that begin with CMPCLR. As well, you can see a list of differences in theStructure (6.27.50) panel. Open it with the **StructurePanel** command, and ensure the Default-CST configuration file is loaded (see Load a configuration file (1.23.8).)Expand the **Comparison** node in the Structure panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DwgCompareStructurePanel.
You must run the **ENDCOMPARE (6.13.6)** command to unload the comparison drawing.

Figure 6.730

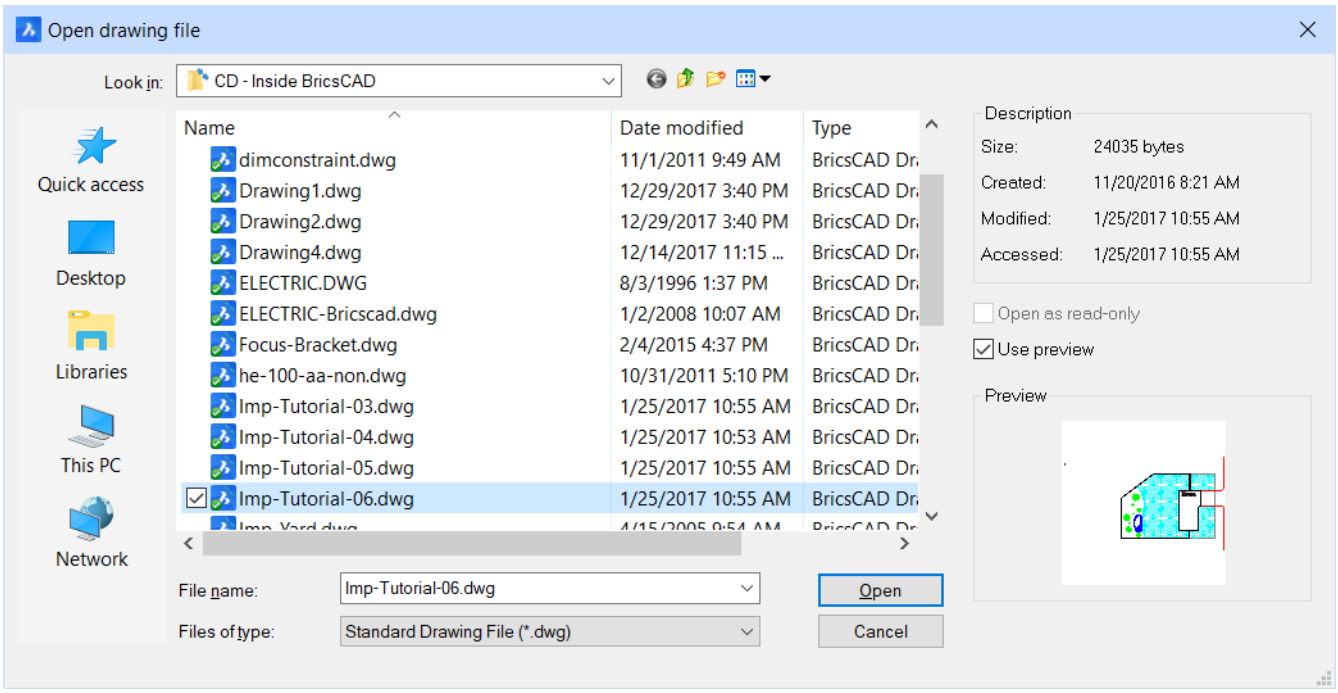


Figure 6.731



Command Options

Option	Description
Limit	Sets the limit on how many entities to compare in the drawings. Prompts you: **New value for CMPDIFFLIMIT <10000000>** - **(enter a value between 1 and 10000000 (ten million), or press Enter to accept the current value.)** The current value is held in the **CMPDIFFLIMIT**variable; initial value is10000000

Procedures

- Setting the comparison system variables (1.14.8)
- Executing a comparison session (1.14.8)
- Analyzing the comparison in the Structure panel (1.14.8)
- Merging the compared drawings (1.14.8)

Related Commands

3dCompare (6.3.3) - executes a geometrical comparison between solids and surfaces in 2 drawing files. The result is shown in a new drawing and reported in the Mechanical Browser panel. EndCompare (6.13.6) - exits a drawing comparison setting, started by the DwgCompare command. KeepMe (6.19.1) - allows to merge the compared drawings visually. StructurePanel (6.27.50) - opens the Structure panel.

Figure 6.732



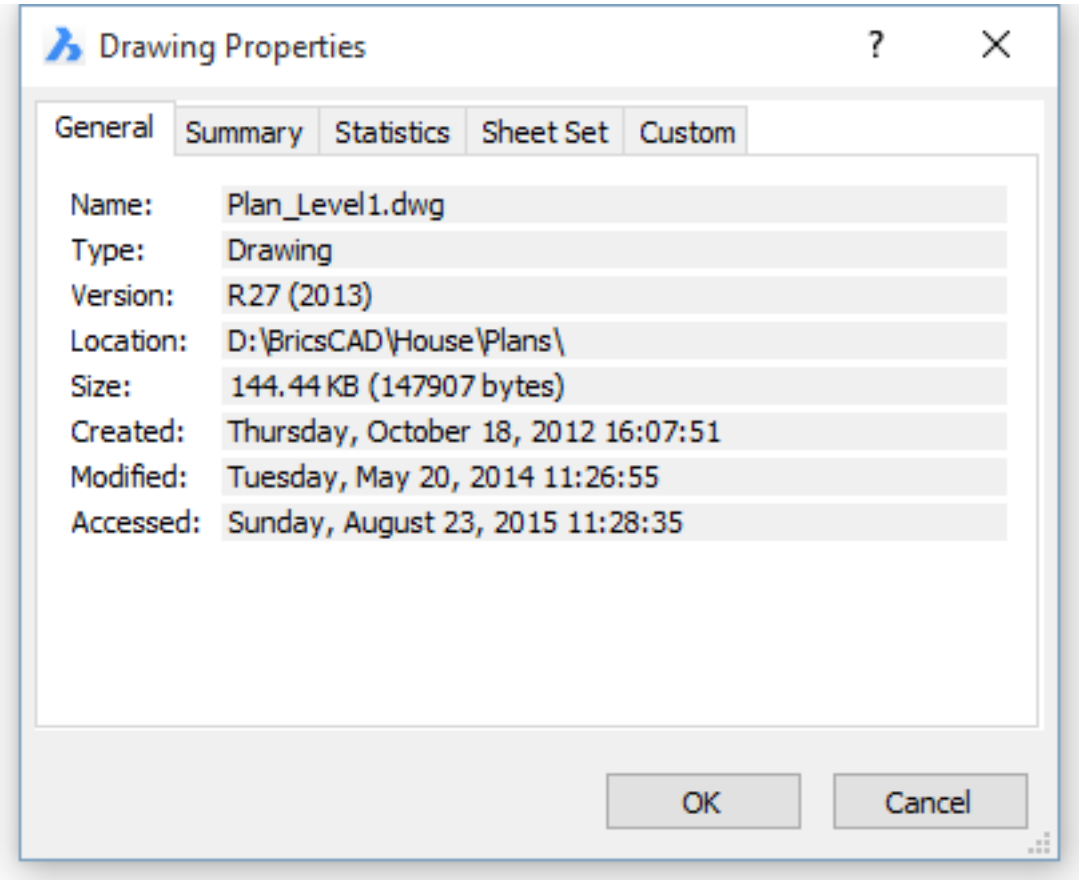
6.10.42 DwgProps

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Opens the Drawing Properties dialog box, showing the general information about the drawing and user-defined properties.

Accessing the Command

command bar: **dwgprops** menu bar: **File| Drawing Properties** : **DwgProps** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DrawingPropertiesGeneral.png})

Figure 6.733



Command Options

Option	Description
General	Displays the general properties of the drawing, such as creation date and time.
Summary	Lets you define the summary properties of the drawing, such as subject and keywords. This data can be searched for by EDMS (engineering data management software) to find drawings in large collections. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dw summary-tab.png})
Statistics	Displays statistical information about the drawing, such as when it was last saved. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dw statistics-tab.png})
SheetSet	Displays sheetset data when the drawing is associated with a sheetset. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dw sheetset-tab.png}) Click **Remove Association** button to remove the association of this drawing with the sheetset collection.
Custom	Displays the custom properties of the drawing.Click the **Add** button to add a new property. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/D Click the **Edit** button to edit the selected property.Click the **Delete** button to delete the selected property.

Procedures

- To open the Drawing Properties dialog (1.14.4)
- Defining default properties (1.14.4)

Figure 6.734

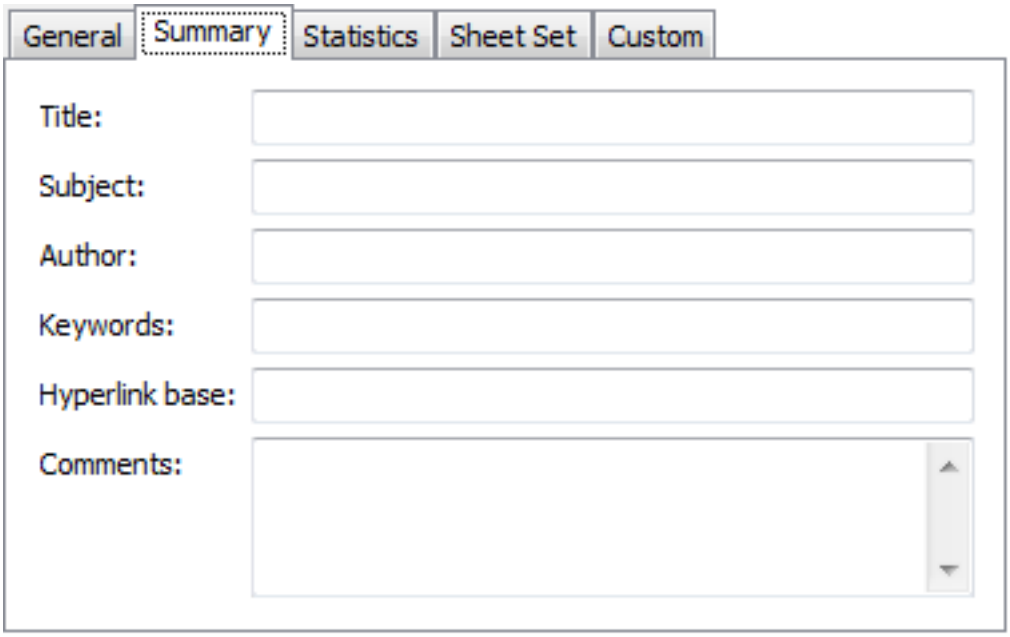


Figure 6.735

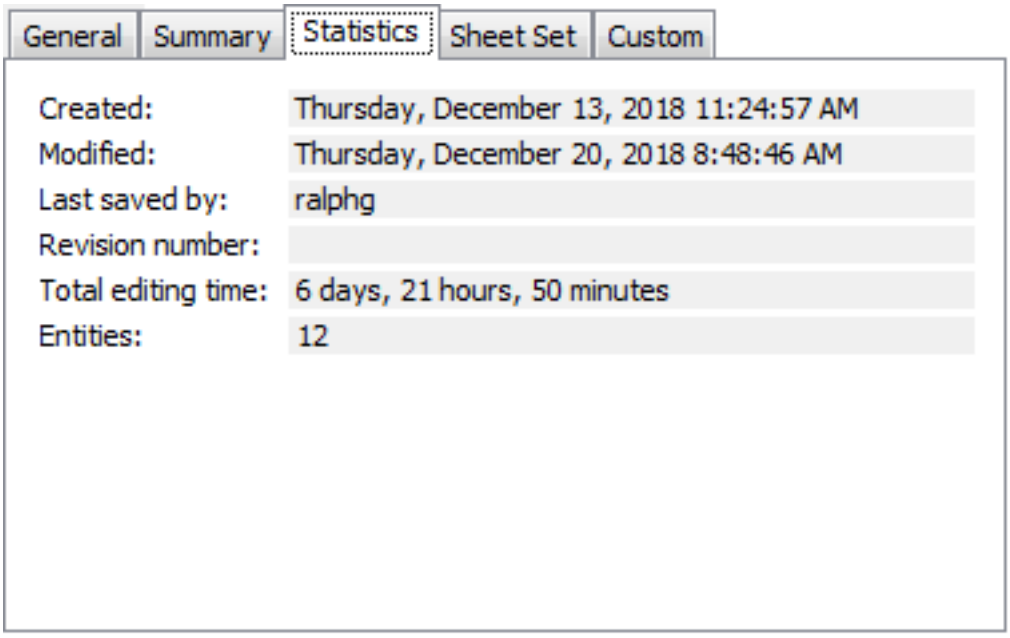


Figure 6.736

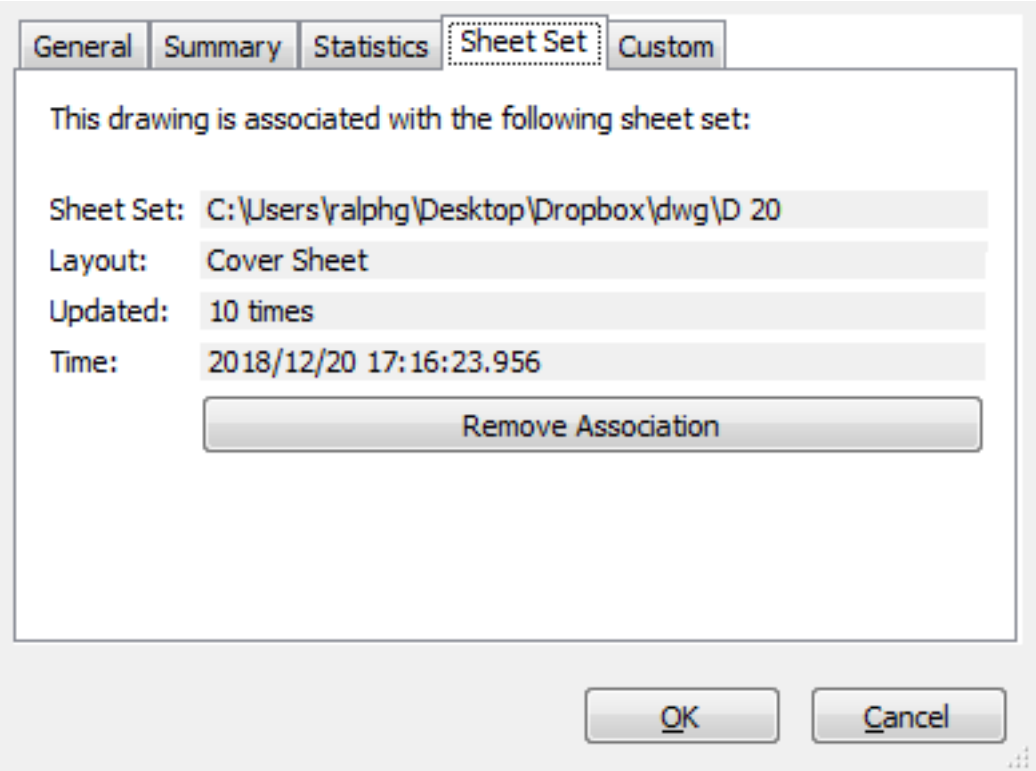


Figure 6.737

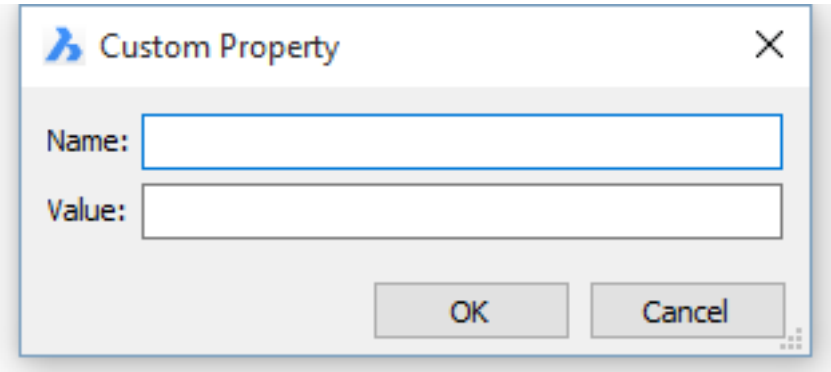


Figure 6.738



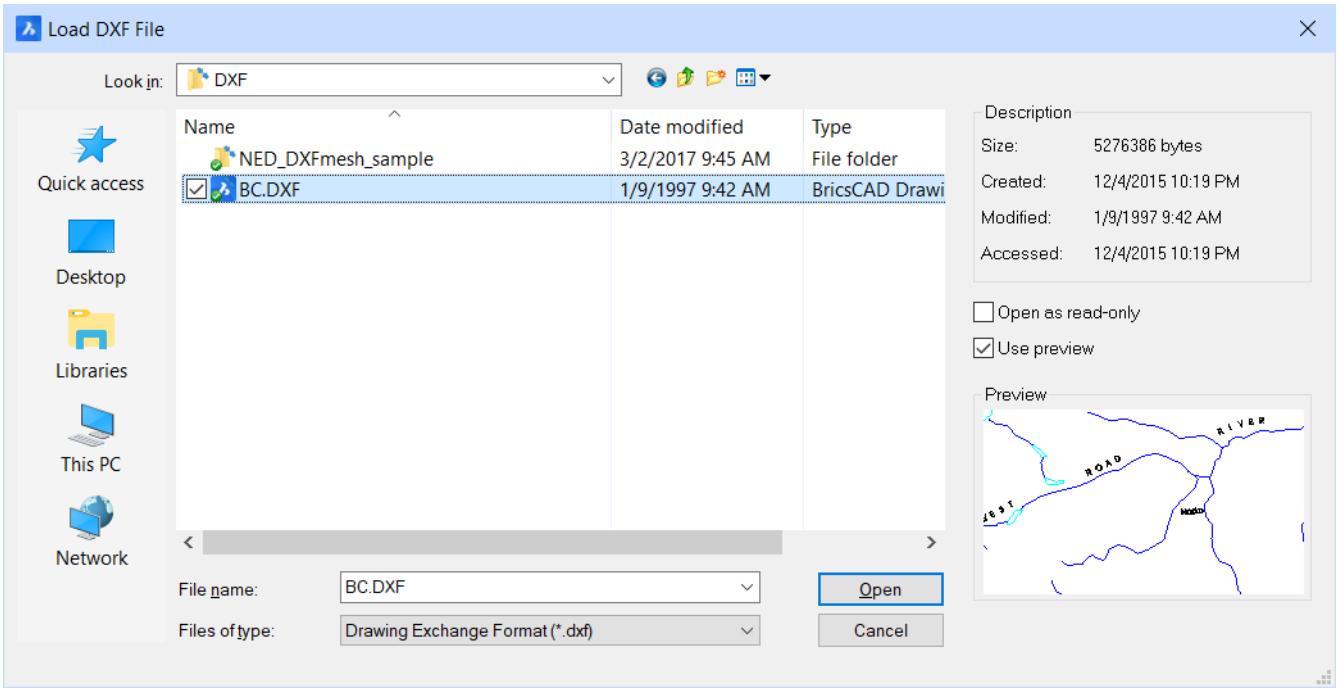
6.10.43 DxfIn

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Imports DXF files (short for "drawing exchange format input"), which is the one of the earliest exchange formats.

Accessing the Command

command bar: **dxfin** : **DxfIn** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Cor filedlg.png}) Choose a DXF file, and then click **Open**. The file is opened in a new drawing.

Figure 6.739



Related Commands

Export (6.13.15) - exports drawings in a variety of formats. DxfOut (6.10.44) - exports drawings in DXF format.

Figure 6.740



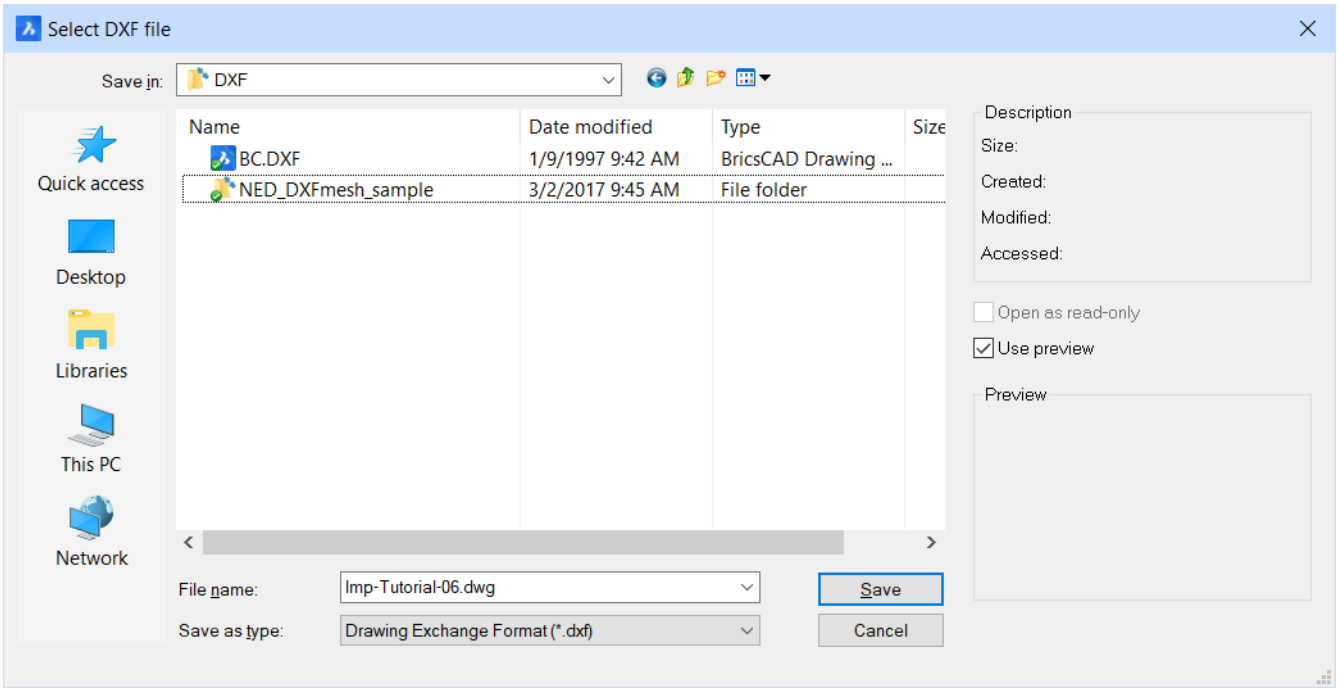
6.10.44 DxfOut

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Exports selected entities from the current drawing in DXF format (short for "drawing interchange format output").

Accessing the Command

command bar: **dxfout** alias: **dx** : **DxfOut** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
file-dlg.png}) Enter a file name, and then click **Save**. **Prompts you in the command bar:** Enter decimal places of accuracy (0-16) [Entities/Binary/Version] <6>:**(Enter an option)**

Figure 6.741



Command Options

Option	Description
Enter decimal places of accuracy(0-16)	Specifies the accuracy of real numbers in the DXF file; default = 6. **(enter a number between 0 (no decimal places) and 16 or press Enter to accept the default value.)** Note: Some CNC (computerized numerical control) machines require 4 decimal places in DXF files.
Entities	Select the entities you want exported to the DXF file; prompts you: **Select entities to export** - **(choose the entities to export; press Ctrl+A to select all entities in the drawing)**
Binary	Exports the drawing in binary DXF format; otherwise, the DXF file is in ASCII (text) format.
Version	Specifies the DXF version number in which to save the drawing; prompts you: **File version R2.5/R2.6/R9/R10/R11/R12/R13/R14/A2000/A2004/A2007 -(enter a version number.)** Caution: exporting to an older version can cause some entities or properties to be lost.

Related Commands

Export (6.13.15) - exports drawings in a variety of formats. DwfOut (6.10.47) - exports drawings in DWF format.

Figure 6.742



6.10.45 DView

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Changes the 3D viewpoint interactively and turns on perspective mode (short for "dynamic view").

Accessing the Command

 command bar: **dview** alias: **dv** menu bar: **View | Define View** toolbar: **View |**  ribbon: **View | View | Viewpoint | Define View** : **DView Prompts you in the command bar:** Select entities or <use DVIEWBLOCK>: **(Press Enter to display the DVIEWBLOCK, or select entities)** DView: CAmera/ TArget/ TWist/ Distance/ POints/ CLip/ Off/ Hide/ PAn/ Zoom/ Undo/<eXit>: **(Enter an option, or press Enter to exit the command)**

Command Options

Option	Description
Use DVIEWBLOCK	Displays a built-in 3D block: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Select Entities To See In Preview	Select one or more entities to be previewed during this command. This command first appeared when computers were slow, and so could not interactively redraw many vector entities.
CAmera	Specifies the camera (eye) location; prompts you: **Specify camera location, or enter angle from XY plane, or [Toggle (angle in)] <current angle>** - **(enter an option)**: • Move the mouse to specify the camera location, click to confirm • Enter a value for the angle from the XY plane • Press **T** to toggle to angle in XY plane
TArget	Specifies the target (look at) location; prompts you: **Specify camera location, or enter angle from XY plane, or [Toggle (angle in)] <current angle>** - **(enter an option)**: • Move the mouse to specify the target location, and then click to confirm • Enter a value for the angle from the XY plane • Press **T** to toggle to angle in XY plane
TWist	Sets the twist (view rotation about Z axis) angle; prompts you: **New view twist <current angle>** - **(move the mouse to specify the camera location, click to confirm; or specify the angle.)**
Distance	Specifies the camera to target distance; prompts you: **New camera-to-target distance <current distance>** - **(move the mouse to specify the camera to target distance, click to confirm; or key in the distance.)** This option turns on perspective mode.

POints	Specifies the camera and target location; prompts you: 1. **Enter target point** - **(specify the point using the mouse, or enter x,y,z coordinates)** 2. **Enter camera point** - **(specify the point using the mouse, or enter x,y,z coordinates)**
CLip	Sets the clipping distances, which cut off the front and back parts of the model; prompts you: **Enter clipping option: Front/Back/<Off>** - **(type F, B, or O.)**
Off	Turns off perspective mode. Use the Distance option to turn perspective mode on.The **Perspective** system variable toggles the perspective property of a view.
Hide	Removes hidden lines from the model.
PAn	Interactively pans around the drawing; prompts you: 1. **Displacement base point** - **(specify the first point to pan from, or enter x,y,z coordinates.)** 2. **Second point** - **(specify another point to pan towards, or enter x,y,z coordinates.)**
Zoom	Zooms in and out interactively; prompts you: **View size** - **(enter a number to change the zoom level)**
Undo	Undoes the last view change
eXit	Exits the command

Related Commands

Camera (6.9.2) - creates perspective views. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. RtRot (6.26.38) - rotates 3D drawings in real-time.

Figure 6.743

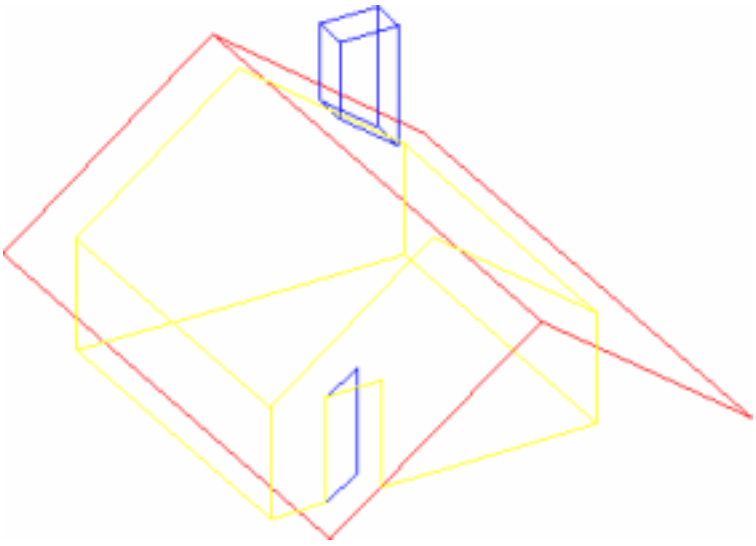


Figure 6.744



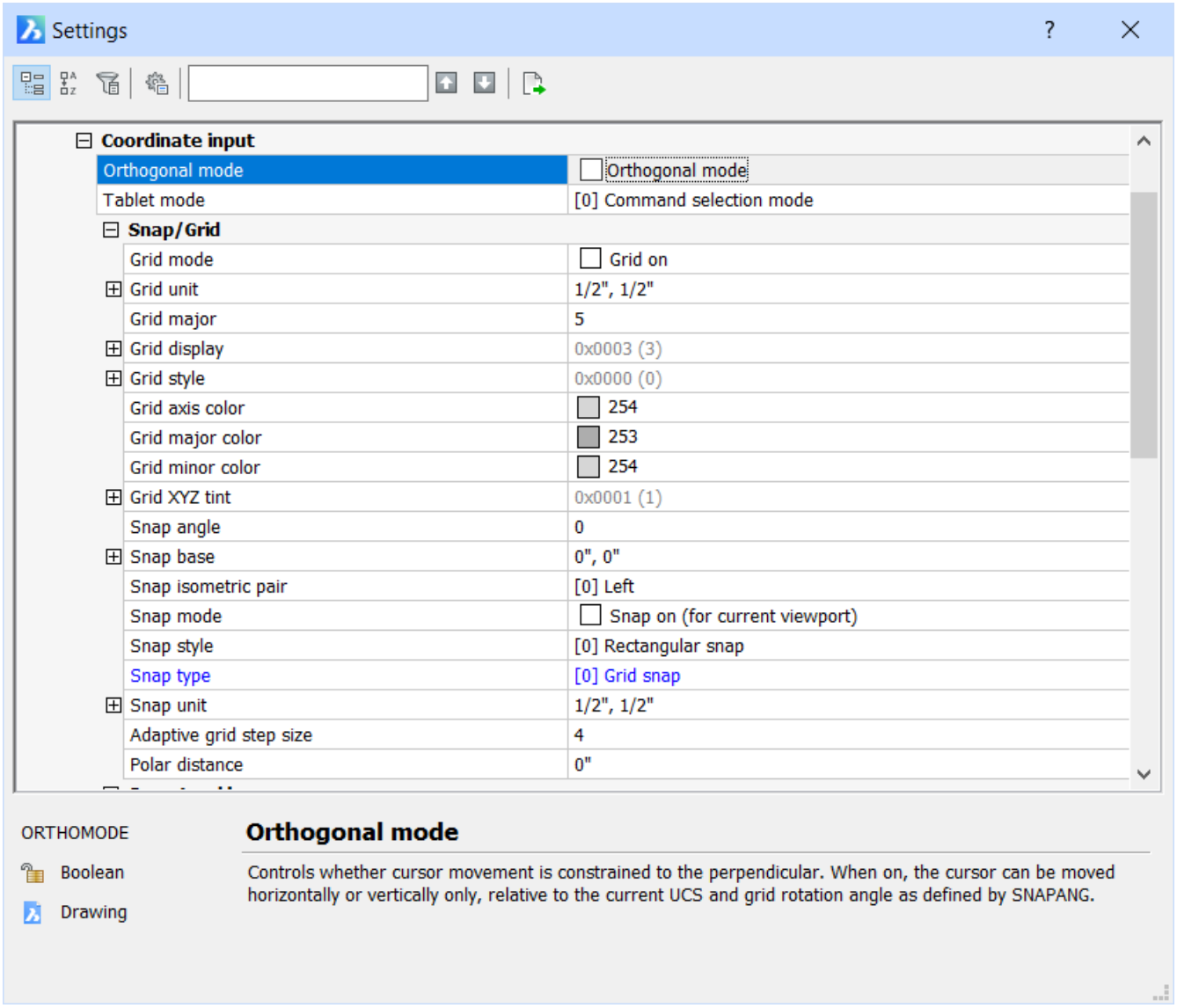
6.10.46 DSettings

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the Settings dialog box at the drafting settings (short for "drafting settings") section. This command is superseded by the Settings command, and remains in the program for compatibility only.

Accessing the Command

command bar: **dsettings** transparent: **'dsettings** alias: **ddrmodes, rm : DSettings** Displays the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dsettings-dlg.png}) Change one or more settings, and then click **X**.

Figure 6.745


Settings
Coordinate input
Orthogonal mode
Orthogonal mode
Tablet mode
[0] Command selection mode
Snap/Grid
Grid mode
Grid on
Grid unit
1/2", 1/2"
Grid major
5
Grid display
0x0003 (3)
Grid style
0x0000 (0)
Grid axis color
254
Grid major color
253
Grid minor color
254
Grid XYZ tint
0x0001 (1)
Snap angle
0
Snap base
0", 0"
Snap isometric pair
[0] Left
Snap mode
Snap on (for current viewport)
Snap style
[0] Rectangular snap
Snap type
[0] Grid snap
Snap unit
1/2", 1/2"
Adaptive grid step size
4
Polar distance
0"
ORTHOMODE
Boolean
Drawing
Orthogonal mode
Controls whether cursor movement is constrained to the perpendicular. When on, the cursor can be moved horizontally or vertically only, relative to the current UCS and grid rotation angle as defined by SNAPANG.

Command Options

Coordinate Input Description Orthogonal Mode Toggles orthogonal mode:

- **On** - constrains cursor movement to the vertical and horizontal
- **Off** - moves cursor freely

Hold down the **Shift** key to turn on ortho mode temporarily; alternatively, double-click **ORTHO** on the status bar. Tablet Mode Toggles tablet mode:

- **Digitizing mode** - uses tablet for drawing input
- **Command selection mode** - uses tablet for command input

Snap/Grid Options Description Grid Mode Toggles the grid display:

- **On** - displays the grid
- **Off** - does not display the grid

To toggle the grid display, you can click GRID on the status bar. Grid Unit Specifies the spacing between grid dots:

- **X** - specifies the distance in the x direction
- **Y** - specifies the distance in the y direction

Grid Major Specifies the frequency of manor gridlines versus minor gridlines. Grid Display Specifies how the grid is displayed.

- **Display beyond limits area -** when checked, the grid displays on the entire drawing
- **Adaptive grid display** - turn on or off: **On** - (checked) the number of grid lines is adapted to the zoom **Off** - (unchecked) the grid does not display if the grid is too dense, and aA warning displays in the command bar: ”The grid spacing is too small to display in this view”.
- **Allow subdivision below grid spacing** - when checked, extra grid lines display when zooming in closely
- **Follow dynamic UCS**- when checked, the grid displays in the XY-plane of the dynamic UCS

Grid Style Specifies whether the grid is displayed with dots or lines in model space, block editor, and layouts. Grid Axis Color Sets the color of the grid lines on the X-axis and Y-axis of the current coordinate system; default = index color 251. Grid Major Color Sets the color of the major grid lines; default = index color 252. Grid Minor Color Sets the color of the minor grid lines: default = index color 253. Grid XYZ tint Specifies whether the grid axis lines, the major grid lines and/or the minor grid lines display in the color of the X-axis (**COLORX**) and the Y-axis (**COLORY**). Snap Angle Rotates the snap and grid relative to the current UCS. Snap Base Locates the origin of the snap and grid relative to the current UCS:

- **X** - specifies the x coordinate.
- **Y** - specifies the y coordinate.

Snap Isometric Pair Specifies the current isometric plane:

- Left
- Top
- Right

To switch isoplanes in the drawing, press **Ctrl+E**. Snap Mode Toggles snap mode:

- **On** - constrains cursor movement to the snap distance
- **Off** - allows the cursor to move freely

To toggle snap mode, click **SNAP** on the status bar. Snap Style Specifies the snap style:

- **Rectangular snap** - sets snap and grid to rectangular.
- **Isometric snap** - sets snap and grid to isometric; adds the Isocircle option to the Ellipse (6.13.5) command.

Snap Type Specifies the snap type:

- **Grid snap** - restricts cursor movement to x and y snap distances.
- **Polar snap** - restricts cursor movement to angles.

Snap Unit Specifies the spacing between snap points:

- **X** - specifies the distance in the x direction
- **Y** - specifies the distance in the y direction

Adaptive Grid Step Size Sets the adaptive grid spacing in pixels for the Manipulate (6.21.2) command, so that the snap varies with the zoom level; default = 4 pixels and range to 2 to 100. Polar Distance Specifies the polar distance; effective only when Snap Type is set to Polar Snap.

Related Commands

DdSetVar (6.10.19) - changes the values of system variables through a dialog box. SetVar (6.27.26) - changes system variables through the command bar.

6.10.47 DwfOut

   Saves the drawing in DWF and other formats (short for ”drawing Web format output”).DwfOut is an alias for the Export (6.13.15) command.

Accessing the Command

command bar: **dwfout** alias: **Export** menu bar: **File | Export** : **DwfOut** See the Export (6.13.15)command. DWF files can be viewed with the DesignReview software, available free from www.autodesk.com/designreview.

Related Commands

Export (6.13.15) - exports drawings in a variety of formats. DxfOut (6.10.44) - exports drawings in DXF format.

6.11 Dimensions

6.11.1 Dim

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates multiple dimension types in a single workflow.

Figure 6.746



Accessing the Command



keyboard: dim **alias:** dimension **menu**: Dimension | Dimension **toolbar**: Dimensions | **ribbon**: Annotate | Dimensions | Dimension : **dim Prompts you:** Dimensioning command [HORizontal/VErtical/ALigned/ANgular/Leader/OBlique/ dimensions/variable STatus/OVerride/SEttings.../LAyer]: Choose an option or do the following:

1. Hover over an entity, then click when the entity highlights. Depending on the entity under the cursor, an appropriate dimension entity displays dynamically.
2. Click to position the dimension entity.
3. Repeat the previous steps to create more dimension entities.

Command Options

Dimension Options	**Description**
HORizontal	Places horizontal linear dimensions; seeDimLinear (6.11.14).
VErtical	Places vertical dimensions; seeDimLinear (6.11.14).
ALigned	Places linear dimensions that are aligned to entities; seeDimAligned (6.11.3).
ANgular	Places angular dimensions that measure angles; seeDimAngular (6.11.4).
Leader	Places leaders; seeDimLeader (6.11.13) command.
CEnter	Places marks at the centers of circles and arcs; seeDimCenter (6.11.8).
Diameter	Places diameter dimensions on circles and arcs; seeDimDiameter (6.11.10).
RAdius	Places radial dimensions on circles and arcs; seeDimRadius (6.11.17).
Baseline	Places multiple linear and angular dimensions from the same base point; seeDimBaseline (6.11.6).
COntinue	Continues linear and angular dimensions from the last endpoint; seeDimContinue (6.11.9).
ORdinate	Places x and y ordinate measurements from an origin point; seeDimOrdinate (6.11.15).
Dimension Editing Options	**Description**
OBlique	Changes the angle of extension lines; seeDimEdit (6.11.12).
ROtated	Places linear dimensions at an angle; seeDimLinear (6.11.14).
Position	Repositions the dimension text; see DimTEdit (6.11.23).
DIStribute	Spaces selected dimensions equally; prompts you: **Specify option to distribute dimensions** [Equal/Offset] <Offset> - **enter an option**: • Equal - prompts youSelect dimensions to distribute - select one or more dimensions, and then press Enter • Offset - prompts you 1. Select base dimension or [Offset] - pick the dimension from which the other ones will be distributed, or type o to specify the offset distance 2. Select dimensions to distribute - select one or more dimensions to be spaced

UPdate dimensions	Applies the current dimension style to a selection of dimension entities; see **Apply** option of -DimStyle (6.11.21).
variable STatus	Lists the status of all dimension variables in the Prompt History (6.29.13) window.
OVerride	Overrides the values of the current dimension style; see DimOverride (6.11.16).
SEttings...	Displays the Drawing Explorer | Dimension Styles dialog box; see DimStyle (6.11.20).
LAyer	Sets a different layer for the default layer on which dimensions are drawing; prompts you: **New value for DIMLAYER, or . for use current /<"use current">** - enter an option: • New value for DIMLAYER - enter a new name for the layer • . for use current - use the current layer

Related Commands

DimStyle (6.11.20)- creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21)- saves and applies dimensions styles at the command line. Style (6.27.53)- sets the text style used by dimensions. DimOverride (6.11.16)- overrides the values of the current dimension style. DimLinear (6.11.14)- places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3)- places aligned dimensions. DimArc (6.11.5)- Places arc length dimensions. DimBaseline (6.11.6)- places additional dimensions from a common baseline. DimContinue (6.11.9)- continues dimensions. DimDiameter (6.11.10)- places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8)- places center marks. DimAngular (6.11.4)- places angular dimensions. DimLeader (6.11.13)andLeader (6.20.20)- places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15)- places x and y ordinate dimensions. Tolerance (6.29.17)- places tolerance tags in drawings. DimEdit (6.11.12)- changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23)- changes the position of dimension text. DdEdit (6.10.14)- edits dimension text. Properties (6.24.49)- edits the properties of dimensions through a palette.

Figure 6.747



6.11.2 Dim1

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Executes a single dimension command at the 'Dimensioning command:' prompt.

Accessing the Command

command bar: **dim1** : **dim1 Prompts you in the command bar:** Dimensioning command [HORizontal/VErtical/ALigned/ANgula dimensions/variable STatus/OVerride/SEttings.../LAyer]:**(Enter a dimension command)**

Command Options

Dimension Drawing Options	**Description**
ALigned	Places linear dimensions that are aligned to entities; see **DimAligned (6.11.3)**.
ANgular	Places angular dimensions that measure angles; see **DimAngular (6.11.4)**.
Baseline	Places multiple linear and angular dimensions from the same base point; see **DimBaseline (6.11.6)**.
CEnter	Places marks at the centers of circles and arcs; see **DimCenter (6.11.8)**.
COntinue	Continues linear and angular dimensions from the last endpoint; see **DimContinue (6.11.9)**.
Diameter	Places diameter dimensions on circles and arcs; see **DimDiameter (6.11.10)**.
HORizontal	Places horizontal linear dimensions; see **DimLinear (6.11.14)**.
Leader	Places leaders; see **DimLeader (6.11.13)**command.
ORdinate	Places x and y ordinate measurements from an origin point; see **DimOrdinate (6.11.15)**.
RAdius	Places radial dimensions on circles and arcs; see **DimRadius (6.11.17)**.
ROtated	Places linear dimensions at an angle; see **DimLinear (6.11.14)**.
VErtical	Places vertical dimensions; see **DimLinear (6.11.14)**.
Dimension Editing Options	
DIStribute	Spaces selected dimensions equally; prompts you: **Specify option to distribute dimensions [Equal/Offset] <Offset>** - enter an option: • Equal - prompts youSelect dimensions to distribute - select one or more dimensions, and then press Enter • Offset - prompts you 1. Select base dimension or [Offset] - pick the dimension from which the other ones will be distributed, or type o to specify the offset distance 2. Select dimensions to distribute - select one or more dimensions to be spaced

LAyer	Sets a different layer for the default layer on which dimensions are drawing; prompts you: **New value for DIMLAYER, or . for use current /<"use current">** - enter an option: • New value for DIMLAYER - enter a new name for the layer • . for use current - use the current layer
Oblique	Changes the angle of extension lines; see **DimEdit (6.11.12)**.
POsition	Moves and reorients dimension text; see **DimTEdit (6.11.23)**.
Dimensioning Utility Options	
OVerride	Overrides settings of dimension variables; see **DimOverride (6.11.16)**.
Settings	Displays the Dimension Style section of the Drawing Explorer; see **DimStyle (6.11.21)**
variable STatus	Reports the current values of dimension variables, including those overriding the current style; see **-DimStyle (6.11.21)**.

Related Commands

DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimArc (6.11.5) - Places arc length dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette.

Figure 6.748



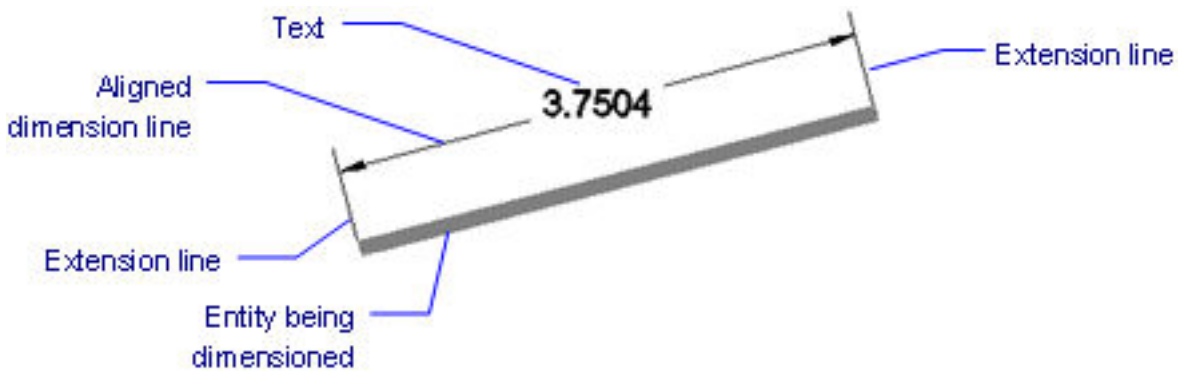
6.11.3 DimAligned

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws dimensions parallel to (aligned with) lines and polyline segments. it also works with polyarcs, arcs, and circles, but dimensions their diameters.

Accessing the Command

 command bar: **dimaligned** alias: **dal, dimali** menu bar: **Dimension | Aligned** toolbar: **Dimensions |**  ribbon:**Draw | Dimensions | Aligned** : **dimaligned Prompts you in the command bar:** Origin of first extension line <Select entity>: **(Pick a point, or else press Enter to select an entity to dimension without specifying extensionlines)** Origin of second extension line: **(Pick another point)** Location of dimension line [Angle/Text]: **(Pick a point, or else choose an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image109.jpg})

Figure 6.749



Command Options

Option	Description
Select entity	Dimensions the length of the selected entity.Prompts you: 1. **Select entity to dimension** - **select an entity. Lines, polyline segments, arcs, circles and edges of 3D solids in the XY-plane of the current UCS are accepted.** 2. **Location of dimension line [Angle/Text]** - **pick a point or choose an option.**
Angle	Specifies the angle of the text; useful for keeping text horizontal with inclined dimension lines.Prompts you: **Angle of dimension text** - **type a value**.
Text	Specifies the text; useful for overriding the default measurement generated by the program.Prompts you: **Dimension text <xx.xx>** - **type a text or press Enter to accept the default text**. You can use two angle brackets -- like these **<>** ones -- to located the program-generated dimension value in amongst other text on the dimension text, such as ”Length of line = <>”.

Procedures

- To create an aligned linear dimension (1.21.8)
- To create a horizontal or vertical linear dimension (1.21.8)
- To create a rotated linear dimension (1.21.8)

Grips Editing

Aligned dimensions can be edited directly through grips:

1. Select the aligned dimension. Notice the grips.
2. Drag the grips to perform the following actions:
    - Drag either grip on the dimension line to move the dimension line.
    - Drag the grip on the text to relocate it.
    - Drag either grip on extension lines to rotate and stretch the dimension.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image503.jpg}) Move the dimension line by dragging either grip near the arrowhead: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
Relocate the text and dimension line by dragging the grip centered on the dimension line (the leader is added automatically, if required): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image508.jpg}
Rotate and stretch the dimension by dragging either grip located at the base of the extension lines: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image512.jpg})

Figure 6.750

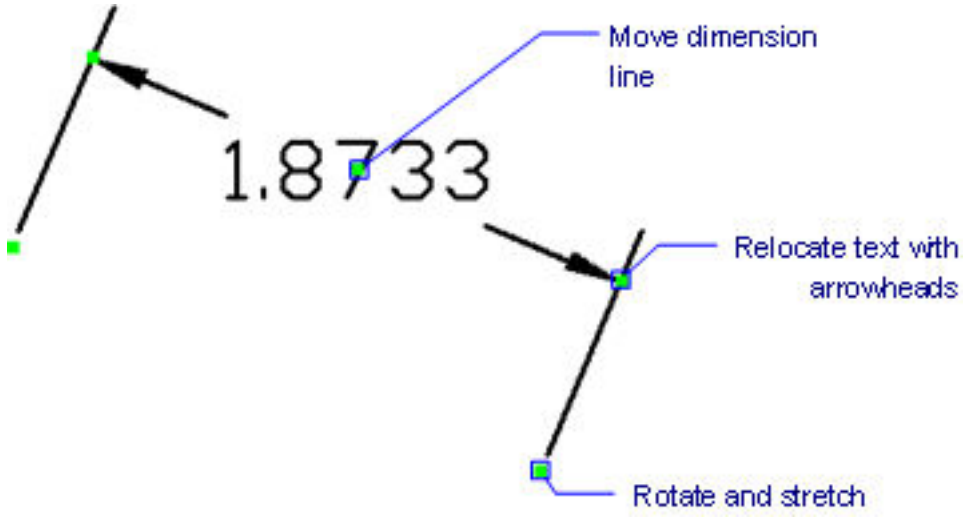


Figure 6.751

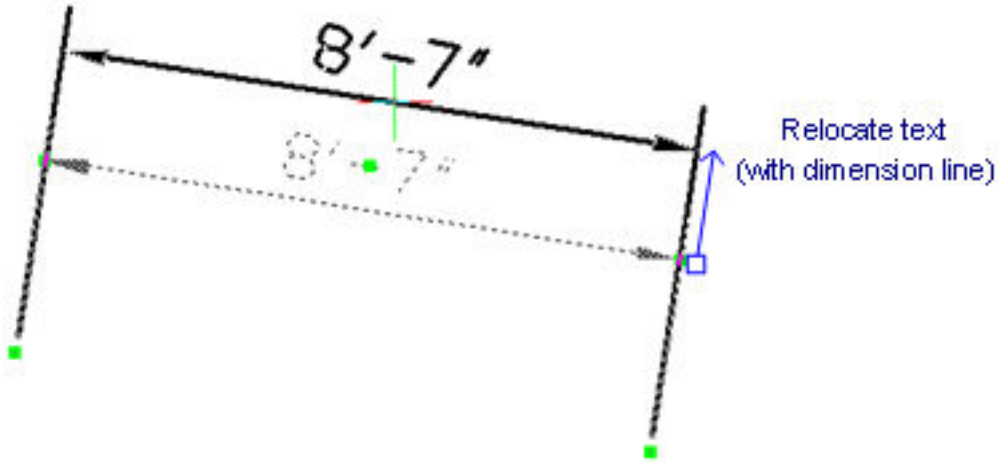


Figure 6.752

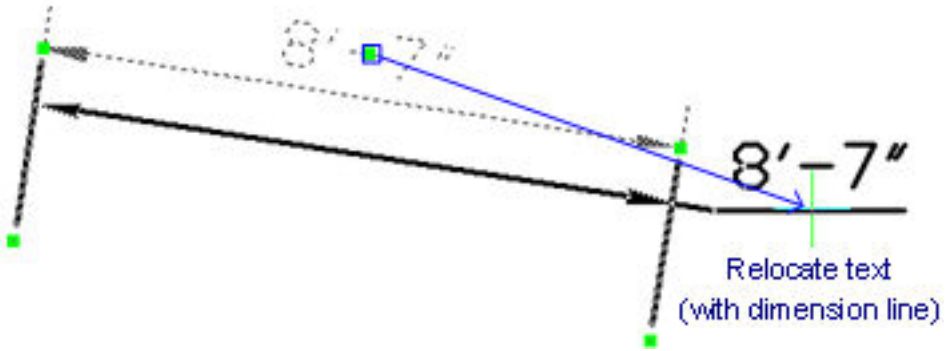
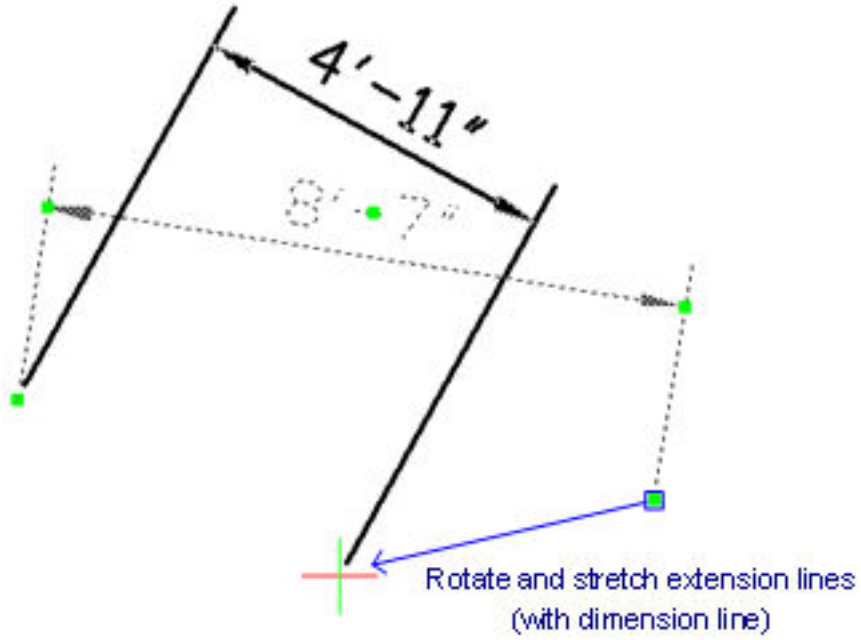


Figure 6.753



Related Commands

DimArc (6.11.5) - Places arc length dimensions. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. AiDimFlipArrow (6.5.2) - flips individual arrows of a dimension. AiDimPrec (**??**) - changes the displayed precision of existing dimensions. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.754



6.11.4 DimAngular

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Dimensions the angle between two lines or two points. Also find the angle of arcs, circles, and polyarcs.

Accessing the Command

 command bar: **dimangular** alias: **dan, dimang** menu bar: **Dimension | Angular** toolbar: **Dimensions |**  ribbon:**Draw | Dimensions | Circle | Angular** : **dimangular Prompts you in the command bar:** Select line, arc, or circle <Press ENTER to specify angle>: **(Select a line, polyline segment, arc, polyarc, or circle; or else press Enter to specify the angle by three points)**

Command Options

Option	Description
Select line, arc, or circle	Dimensions the angle between two line segments, or of arcs and circles. The prompt you see depends on the entity you selected: Picking a line or polyline segment prompts you: 1. **Other line for angular dimension** - **pick another line segment** 2. **Location of dimension arc [Angle/Text]** - **pick a point or enter an option** Picking a circle prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **Other side of angle** - **pick another point** 2. **Location of dimension arc [Angle/Text]** - **pick a point or enter an option** Picking an arc or polyarc prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **Angle/Text/<Location of dimension arc>** - **pick a point to locate the dimension arc, or else enter an option:** • **Angle** specifies the angle of the dimension text. • **Text** specifies different text for the dimension line. The program end the command by reporting the angle: **Dimension text: = 242**

Press ENTER to Specify Angle	Specifies the angular dimension by three points. Prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **Vertex of Angle** - **pick a point to specify the first point, which is the vertex of the angle** 2. **First Side of Angle** - **pick a point to specify the second point, which is one leg of the angle** 3. **Other Side of Angle** - **pick a point to specify the third point, which is the second leg of the angle** 4. **Location of Dimension Arc** - **pick a point to locate the position of the dimension arc**

Procedures

- To dimension an angle between two lines (1.21.9)
- To dimension an angle encompassed by an arc (1.21.9)
- To create an angular dimension defined by a vertex and two endpoints (1.21.9)

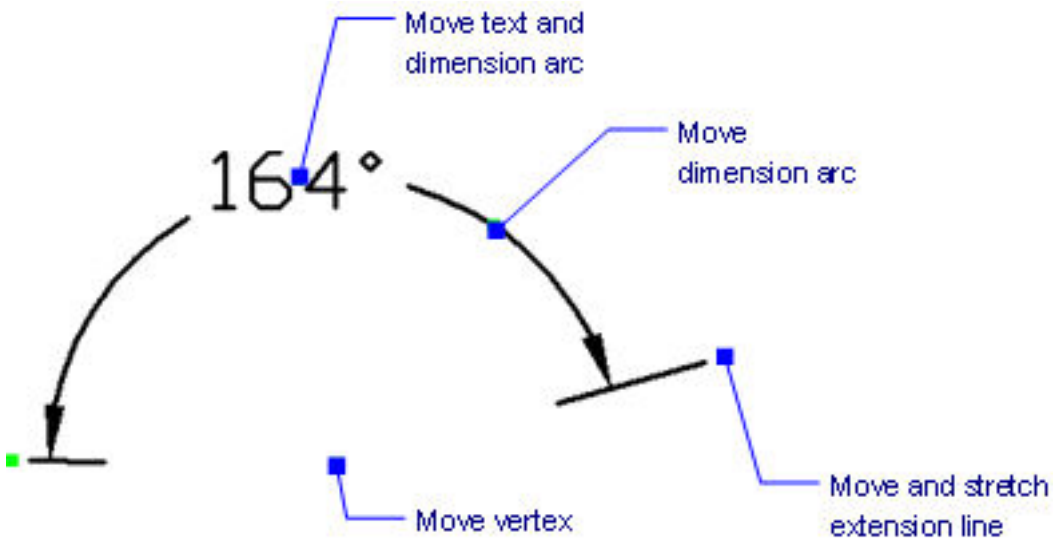
Grips Editing

Angular dimensions can be edited directly through grips:

1. Select the angular dimension. Notice the grips.
2. Drag the grips to perform the following actions:
   - Drag the grip on the text to relocate it.
   - Drag the grip on the dimension arc to move it.
   - Drag either grip on extension lines to move and stretch the dimension.
   - Drag the grip at the vertex to move it.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image497.jpg})

Figure 6.755



Related Commands

DimArc (6.11.5) - Places arc length dimensions. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. AiDimFlipArrow (6.5.2) - flips individual arrows of a dimension. AiDimPrec (**??**) - changes the displayed precision of existing dimensions. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.756

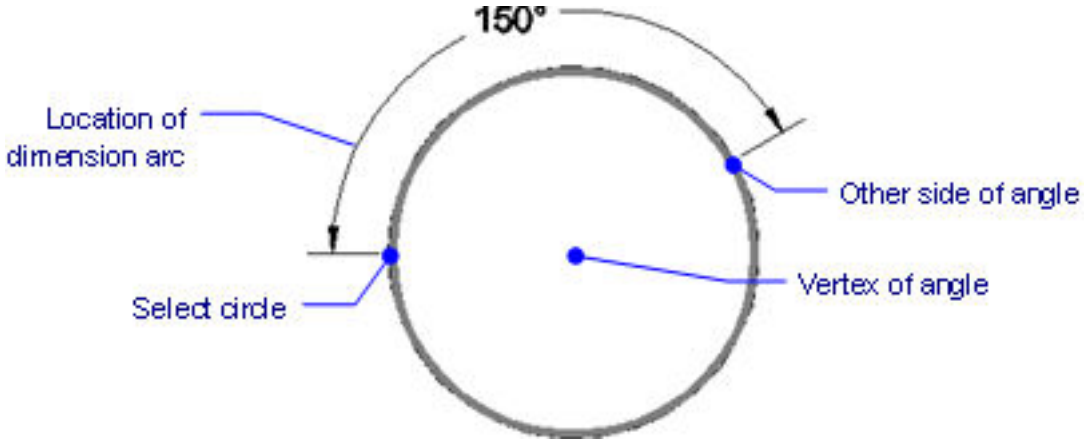


Figure 6.757

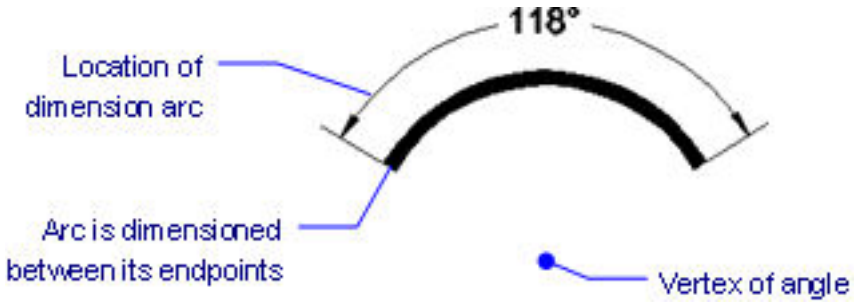


Figure 6.758

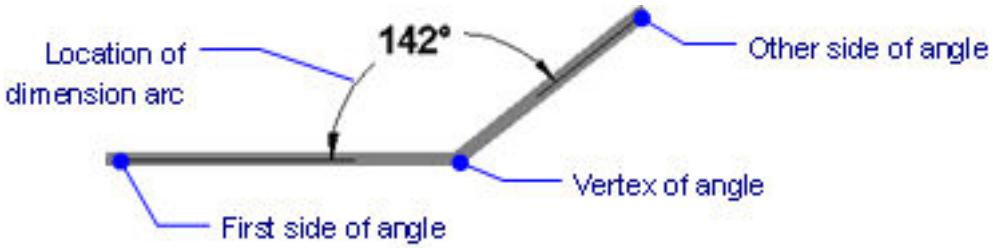


Figure 6.759



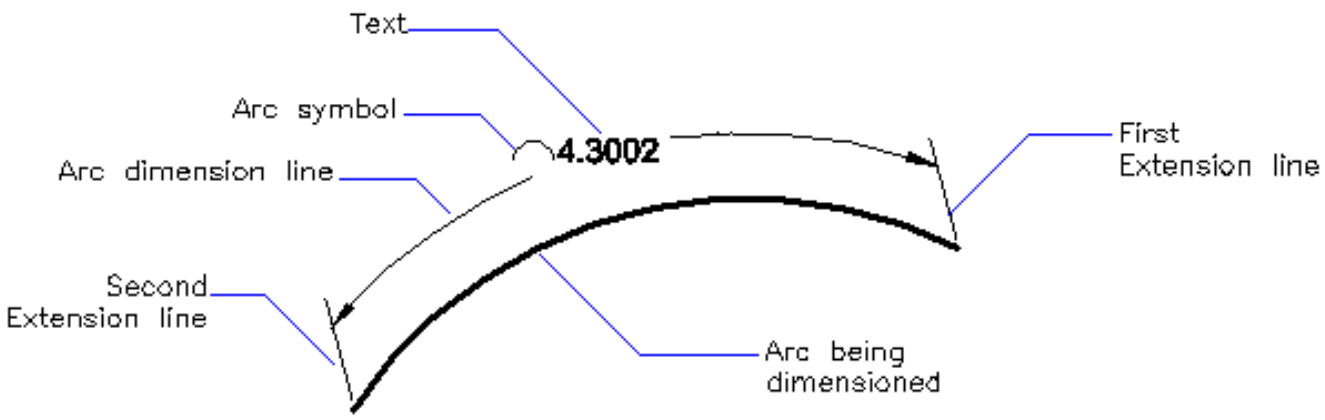
6.11.5 DimArc

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places dimensions that measure the lengths of arcs and polyarcs.

Accessing the Command

 command bar: **dimarc** menu bar: **Dimension | Arc** toolbar: **Dimensions |** ribbon:**Draw | Dimensions | Circle | Arc** : **dimarc Prompts you in the command bar:** Select arc or polyline arc segment: **(Click an arc or polyline arc segment)** Location of dimension arc [Angle/Text/Partial/Leader]: **(Pick a point to locate the dimension arc away from the entity; or else choose an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so

Figure 6.760



Command Options

Option	Description
Select arc or polyline arc segment	Select one entity, either an arc or a polyarc.
Angle	Specifies the angle of the text. Prompts you: **Angle of dimension text** - **enter the rotation angle of the text, or else pick two points**
Text	Specifies the text; useful for overriding the default measurement generated by the program. Prompts you: **Dimension text** - **enter the replacement text**
Partial	Dimensions part of the selected arc or polyline arc segment. Prompts you: 1. **Specify first point for arc length dimension**- **pick a point to position the first extension line** 2. **Specify second point for arc length dimension**- **pick another point on the arc or polyline segment to position the second extension line** It can be helpful to use entity snaps, such as **NEArest** or **MIDpoint** to snap right on the arc.
Leader	Adds a leader, which points from the dimension text to the arc. Prompts you: **Angle/Text/Partial/No leader/<Location of dimension arc>** - **pick a point to locate the dimension arc, or else choose an option**
No leader	Removes the leader; this option works only during the command, after you chose the Leader option. Returns to the previous prompt: **Angle/Text/Partial/Leader/<Location of dimension arc>** - **Pick a point to locate the dimension arc or choose an option.**

Procedures

- To create an arc length dimension (1.21.8)

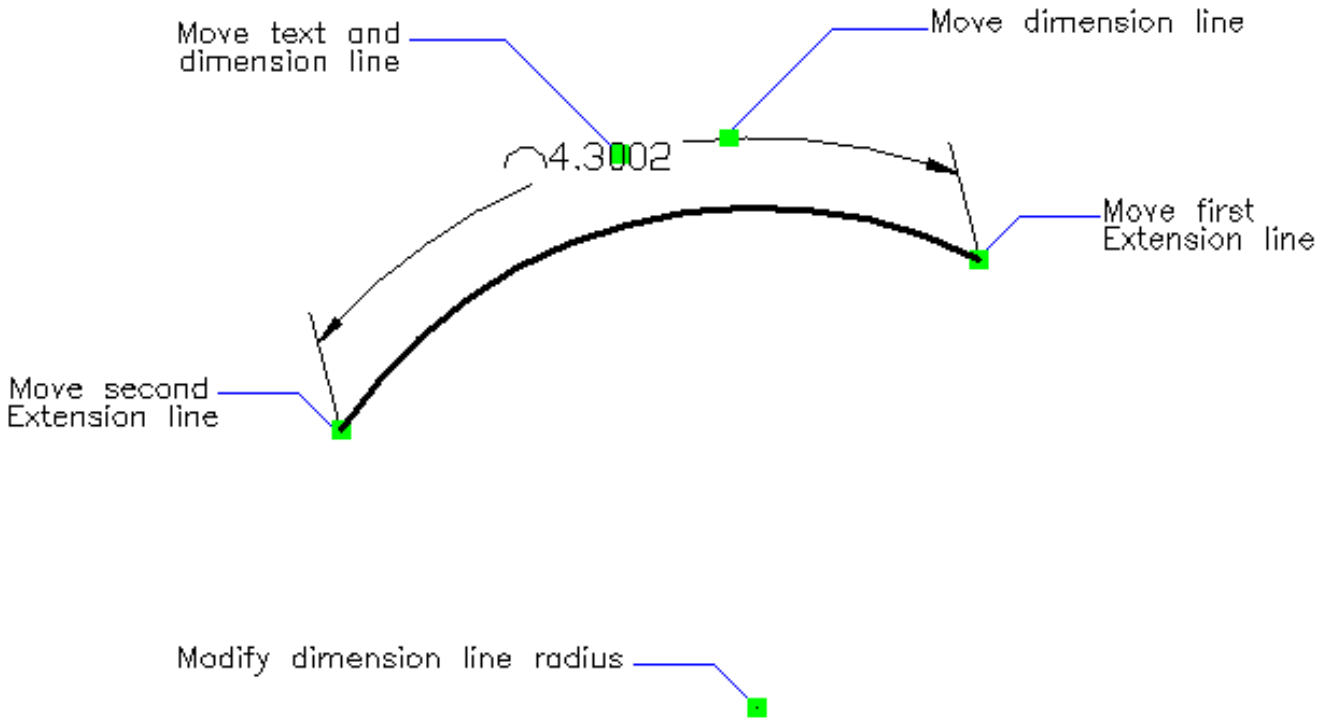
Grips Editing

Arc length dimensions can be edited directly through grips:

1. Select the arc length dimension. Notice the grips.
2. Drag the grips to perform the following actions:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ArcDimensionEditGrips.png})

Figure 6.761



Related Commands

DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. AiDimFlipArrow (6.5.2) - flips individual arrows of a dimension. AiDimPrec (**??**) - changes the displayed precision of existing dimensions. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.762



6.11.6 DimBaseline

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places multiple linear and angular dimensions, all starting at the same base point of an existing dimension. This command works onlywhen at least one other dimension already exists in the drawing.

Accessing the Command


command bar: **dimbaseline** aliases: **dba, dimbase** menu bar: **Dimension | Baseline** toolbar: **Dimensions |** 
ribbon: **Draw | Dimensions | Continue | Baseline** : **dimbaseline Prompts you in the command bar:** Baseline: ENTER to select starting dimension/<Origin of next extension line>: **(Pick a point to locate the second extension line, or else press Enter to pick a new starting point)** Dimension text = xx.xx Baseline: ENTER to select starting dimension/<Origin of next extension line>: **(Pick another point, or else press Esc to exit the command)** Baseline dimensions applied to a linear dimension: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Baseline dimensions applies to an angular dimension: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18

Figure 6.763

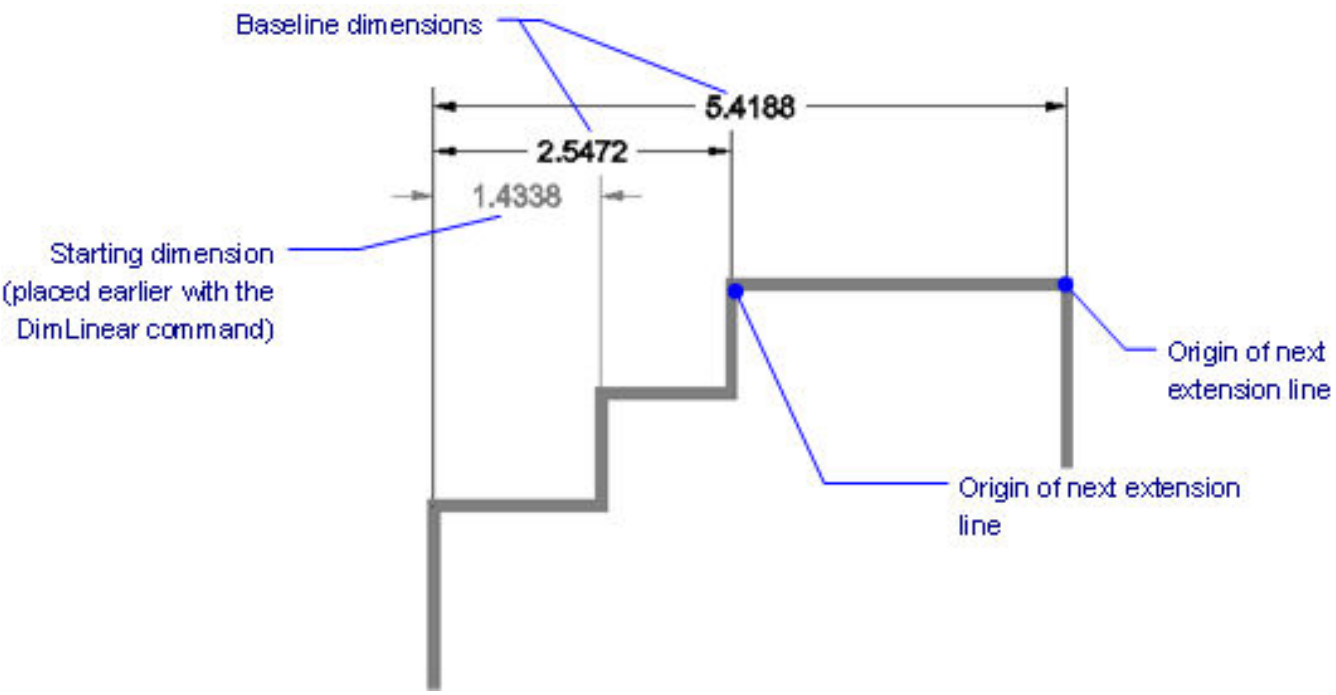
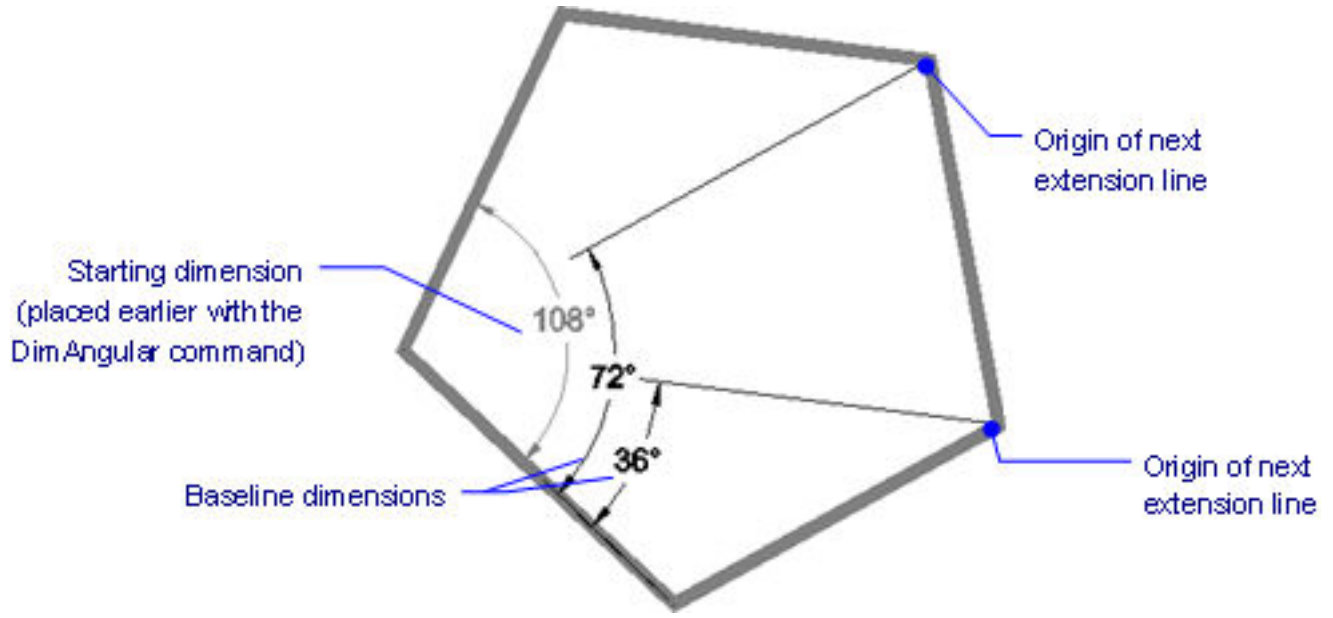


Figure 6.764



Command Options

Option	Description
ENTER to Select Starting Dimension	Chooses the linear or angular dimension from which to continue dimensioning.
Origin of Next Extension Line	Locates the endpoint of the second extension line; the first extension line is co-located with the first extension of the starting dimension. The command repeats this prompt until you press Esc.

Procedures

- To create baseline dimensions (1.21.8)

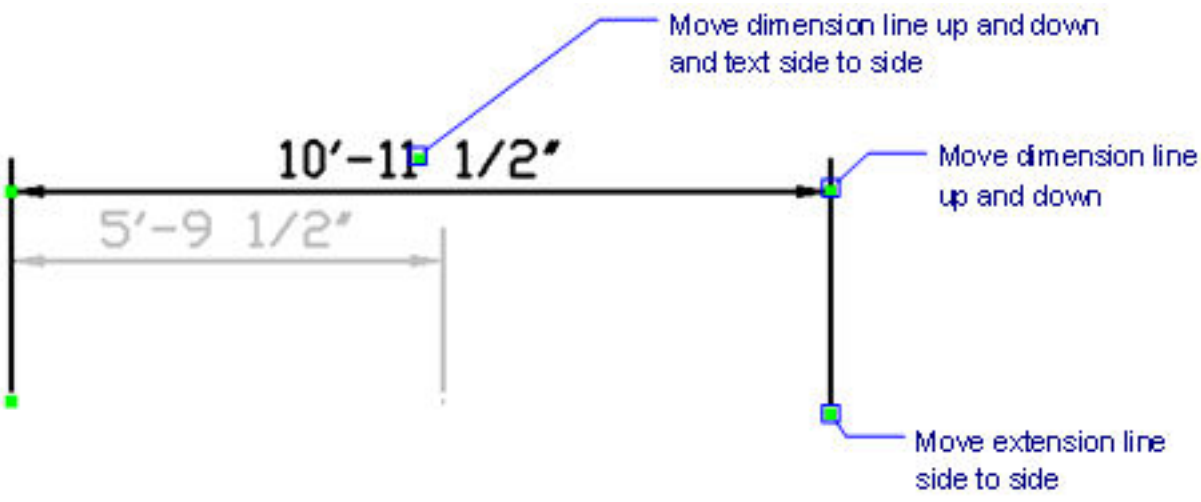
Grips Editing

Baseline dimensions can be edited directly through grips:

1. Select the baseline dimension. Notice the grips.
2. Drag the grips to perform the following actions:
   - Drag either grip on the dimension line to move the dimension line; the extension lines change length.
   - Drag the grip on the text to relocate it; the dimension line moves along, and a leader line is added, if required.
   - Drag either grip on the extension lines to move the extension line; the dimension line changes length.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image519.jpg})

Figure 6.765



Related Commands

DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.766



6.11.7 DimBreak

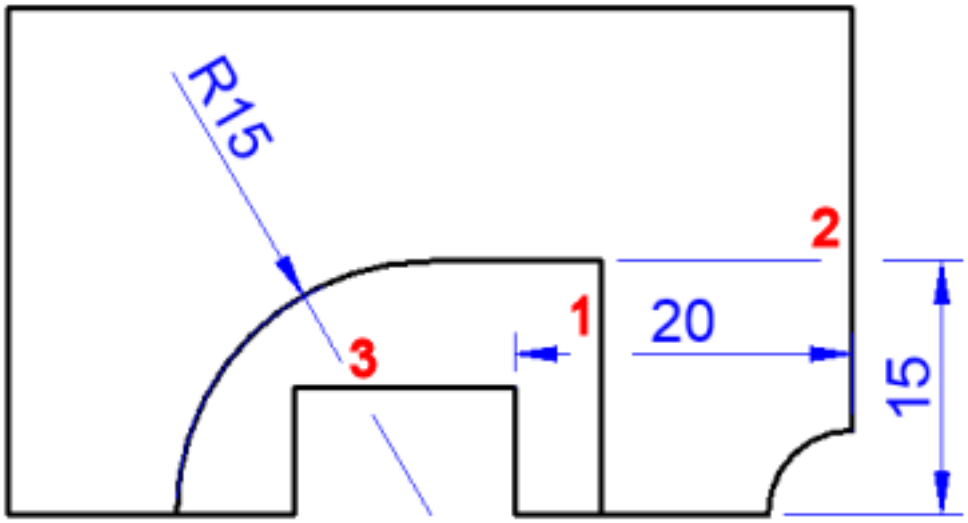
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Breaks dimension lines, extension lines, and leaders at locations where they cross other entities; also removes breaks.

Accessing the Command


command bar: **dimbreak** menu bar: **Break Dimension** toolbar: **Dimensions** |: **dimbreak Prompts you in the command bar:** Select dimension to add/remove break or [Multiple]: **(Select a dimension entity)** Select object to break dimension or [Auto/Manual/Remove]: **(Select an entity that crosses the dimension)** Select object to break dimension: (**Select an entity, or else press Enter to end the command)** The dimension line and extension lines of the selected dimension are broken at the selected entity or entities. The size of the gap is defined by the **DimBreakSize** variable of the current dimension style. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sourc

1. Dimension break on dimension line
2. Dimension break on extension line
3. Dimension break on leader

Figure 6.767



Command Options

Option	Description
Multiple	Selects multiple dimension entities.Prompts you: 1. **Select dimensions to add/remove break- select a dimension entity, or enter all to select all dimensions in the drawing Entities in set: 1** 2. **Select dimensions to add/remove break- select more dimension entities, and then press Enter to start selecting breaking entities**
Auto	Breaks selected dimensions at all intersections found with other entities.
Manual	Manually defines the width of the dimension break.You are prompted to specify two points: 1. **Specify first break point - pick a point on the dimension** 2. **Specify second break point - pick another point on the dimension** This option is not available in Auto mode.
Remove	Removes all dimension breaks from the selected dimension(s).

Tutorial: Using the DimBreak Command in BricsCAD

Click here (1.4.13) to watch

Related Commands

Break (6.6.18) - removes a portion of an entity.

Figure 6.768



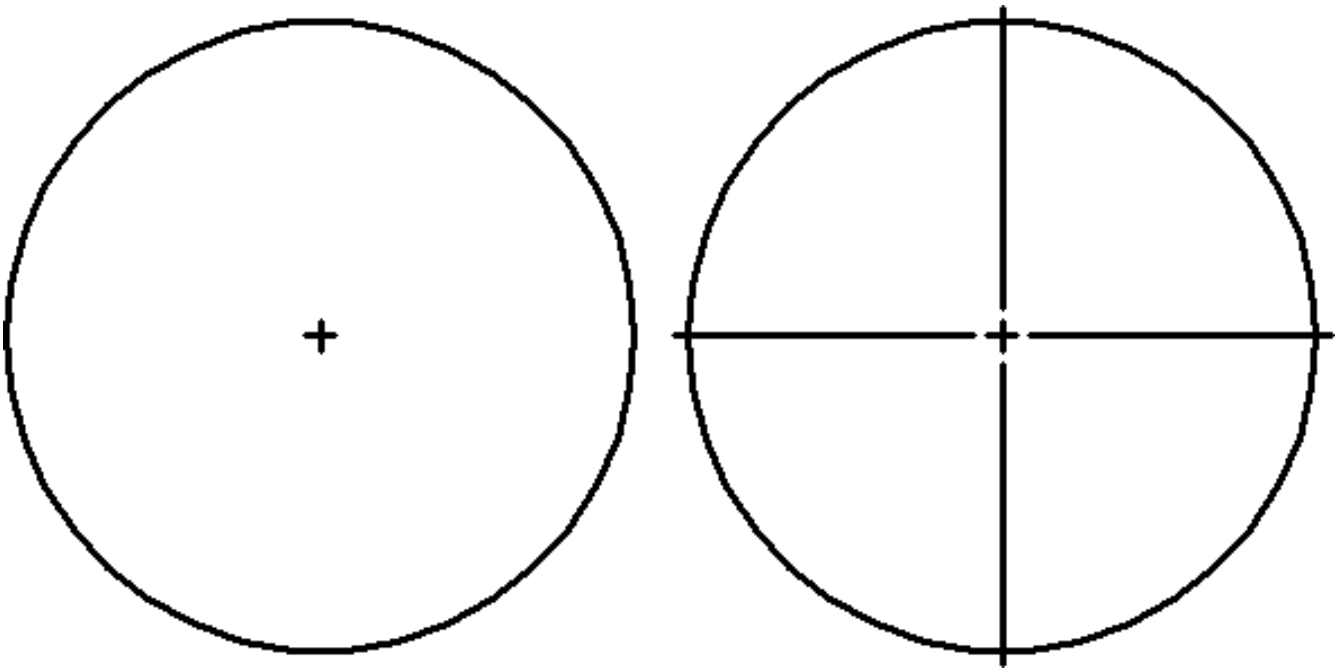
6.11.8 DimCenter

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places center marks and lines at the center points of circles. arcs, and polyarcs.

Accessing the Command


command bar: **DimCenter** alias: **dce** menu bar: **Dimension | Center Lines** toolbar: **Dimensions | Circles |** 
ribbon:**Draw | Dimensions | Circle | Center Mark** : **DimCenter Prompts you in the command bar:** Select arc or circle to dimension: **(Choose an arc, polyline arc segment, or circle)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
The DIMCEN (1.13.11) system variable controls the length of center mark lines. Left: DimCen = 0.09; right: DimCen = -0.09

Figure 6.769



Command Options

Option	Description
Select arc or circle to dimension	Choose an arc, polyline arc segment, or circle

Procedures

- Creating center marks or center lines (1.21.10)

Related Commands

CenterMark (6.9.6) - creates a center mark entity associated with a selected circle or arc. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.770



6.11.9 DimContinue

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places a chain of dimensions by continuing linear or angular dimensions from the endpoint of the previous dimension.

Accessing the Command


command bar: **dimcontinue** aliases: **dco, dimcont** menu bar: **Dimension | Continue** toolbar: **Dimensions** |  ribbon:**Draw | Dimensions | Continue** : **DimContinue Prompts you in the command bar:** Continue: ENTER to select starting dimension/<Origin or next extension line>: **(Choose a point for the endpoint of the second extension line, or press Enter to start at another dimension)** Dimension text = xx.xx Continue: ENTER to select starting dimension/<Origin or next extension line>: **(Choose another point, or press Enter to exit command)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image117.jpg}) Continue dimensions applied to a linear dimension (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou Continue dimensions applies to an angular dimension

Figure 6.771

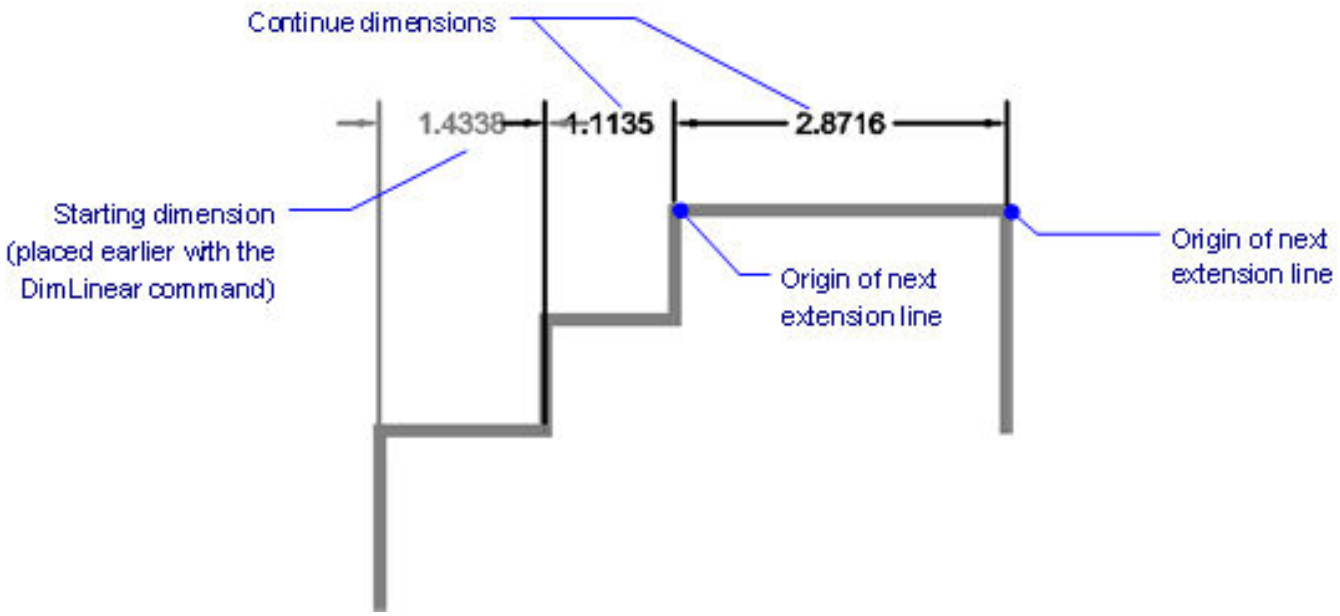
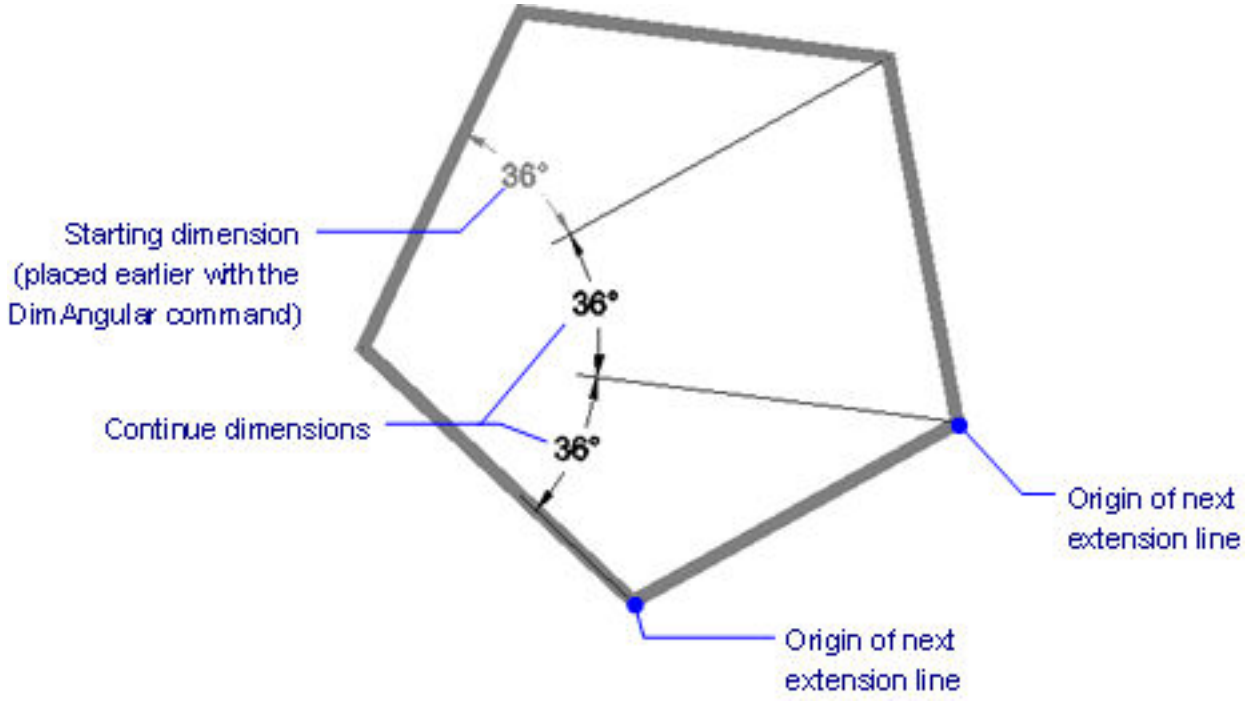


Figure 6.772



Command Options

Option	Description
ENTER to Select Starting Dimension	Choose the linear or angular dimension from which to continue dimensioning.
Origin of Next Extension Line	Locates the endpoint of the second extension line; the first extension line is co-located with the second extension of the previous dimension. The command repeats this prompt until you press Esc.

Procedures

- To create continued dimensions (1.21.8)

Grips Editing

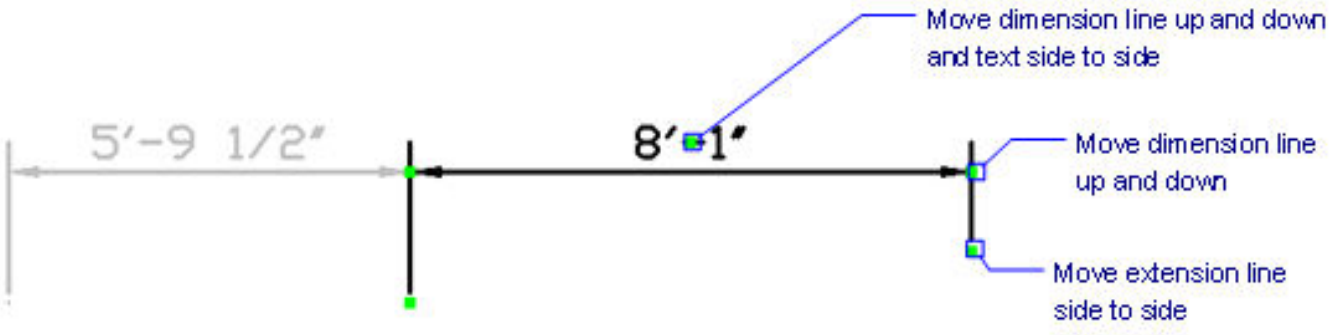
Continued dimensions can be edited directly through grips:

1. Select the continued dimension. Notice the grips.
2. Drag the grips to perform the following actions:

- Drag either grip on the dimension line to move the dimension line; the extension lines change length.
- Drag the grip on the text to relocate it; the dimension line moves along, and a leader line is added, if required.
- Drag either grip on the extension lines to move the extension line; the dimension line changes length.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image518.jpg})

Figure 6.773



Related Commands

DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.774



6.11.10 DimDiameter

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Dimensions the diameter of circles, arcs,and polyarcs, and places the center mark.

Accessing the Command


command bar: **dimdiameter** aliases: **ddi, dimdia** menu bar: **Dimension | Diameter** toolbar: **Dimensions |** ribbon: **Draw | Dimensions | Circle | Diameter** : **DimDiameter Prompts you in the command bar:** Select arc or circle to dimension: **(Choose an arc, polyline arc segment, or circle)** Dimension text = xx.xx Location of dimension line [Angle/Text]: **(Position the dimension test, or else enter an option to change the text)** A circle with diameter dimension placed outside: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image120.jpg}) An arc with diameter dimension placed inside: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRe

Figure 6.775

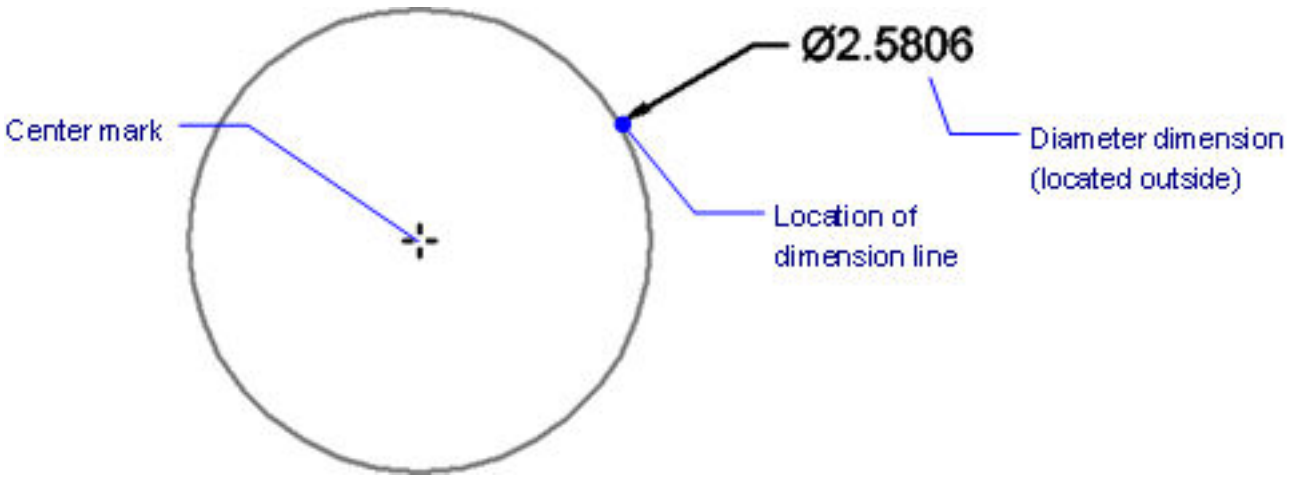
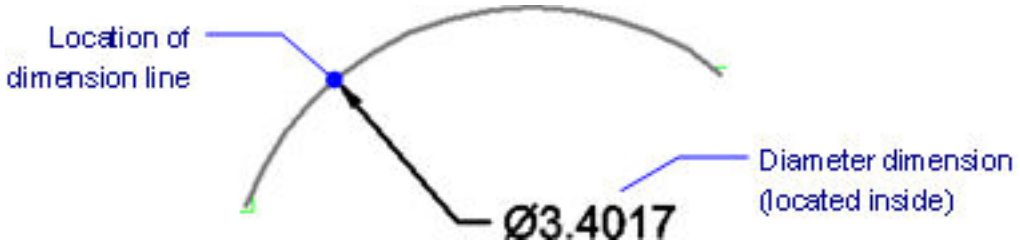


Figure 6.776



Command Options

Option	Description
Select arc or circle to dimension	Choose an arc, polyline arc segment, or circle
Angle	Changes the angle of the text.
Text	Edits the the default dimension text that had been generated by the program.

Procedures

- To create a diametrical dimension (1.21.10)
- Creating center marks or center lines (1.21.10)
- To create a radial dimension (1.21.10)

Grips Editing

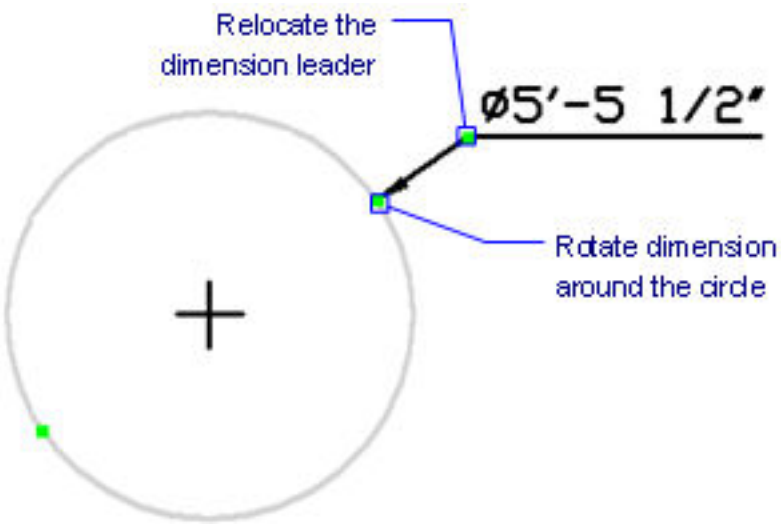
Diameter dimensions can be edited directly through grips:

1. Select the diameter dimension. Notice the grips.
2. Drag the grips to perform the following actions:

- Drag either grip on the circle to rotate the dimension around the circle.
- Drag the grip on the leader to relocate it and the associated text.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image513.jpg})

Figure 6.777



Related Commands

DimArc (6.11.5) - Places arc length dimensions. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. AiDimFlipArrow (6.5.2) - flips individual arrows of a dimension. AiDimPrec (**??**) - changes the displayed precision of existing dimensions. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.778



6.11.11 DimDisassociate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Removes the associativity from selected dimension entities.

Accessing the Command

command bar: **dimdisassociate** menu: **Dimension | Disassociate Dimensions** : **DimDisassociate Prompts you in the command bar:** Select dimensions to disassociate: **(Select one or more dimensions, or enter all to select all dimensions in the current drawing)** Entities in set: 1 Select dimensions to disassociate: **(Select more dimensions, or else press Enter to exit the command)** xx disassociated.

Command Options

Option	Description
Select dimensions to disassociate	Enter one of the following • Select one or more dimensions • Enter **all** to select all dimensions in the current drawing • Press Enter to exit the command

Related Commands

DimReassociate (6.11.18) - reassociates or associates dimensions to entities or points on entities.

Figure 6.779



6.11.12 DimEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits the position, angle, and wording of dimension text, and changes the angle of extension lines.

Accessing the Command

command bar: **dimedit** aliases: **ded, dimed** menu bar: **Dimension | Make Oblique Dimension | Edit Dimension Text Dimension | Rotate Dimension Text Dimension | Reposition Dimension Text Dimension | Restore Text Position** toolbar: **Dimensions |** ribbon:**Draw | Dimensions | Edit : DimEdit Prompts you in the command bar:** Edit dimension [Edit/Oblique/Rotate/REstore] <REstore text>: **(Enter an option, or press Enter to restore the dimension text to its default position)**

Command Options

Option	Description
Edit	Changes the text; prompts you: 1. **New dimension text <>** - **(specify the new text.)** 2. **Selection dimensions to replace with new text** - **(choose one or more dimensions; prompt repeats until you press Enter.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Use the **<>** symbols indicate the default text, so that you can insert text in front or behind the default, such as "Adjust to <> inches."
Oblique	Rotates (obliques or slants) the extension lines; useful for creating isometric dimensions. Prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **Select linear dimensions to make oblique** - **(choose one or more linear dimensions; prompt repeats until you press Enter.)** 2. **Enter obliquing angle** - **(specify the angle.)** The angle is measured counter clockwise from the positive x-axis.
Rotate	Rotates dimension text; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **Angle of dimension text** - specifies the angle. The angle is measured counter clockwise from the positive x-axis. 2. **Select entities to rotate text** - **(choose one or more dimensions; prompt repeats until you press Enter.)**

Restore Text	Restores the text to its original position; this option does not restore edited text or obliqued extension lines. Prompts you: **Select dimensions to restore text to default position** - **(choose one or more dimensions; prompt repeats until you press Enter.)**

Procedures

- Using grips to edit a linear dimension (1.21.15)
- Editing the dimension text (1.21.15)
- To rotate the dimension text (1.21.15)
- To reposition the dimension text (1.21.15)
- To restore the text position (1.21.15)
- To make the extension lines oblique (1.21.15)

Related Commands

AiDimFlipArrow (6.5.2) - flips individual arrows of a dimension. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimArc (6.11.5) - Places arc length dimensions. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.780

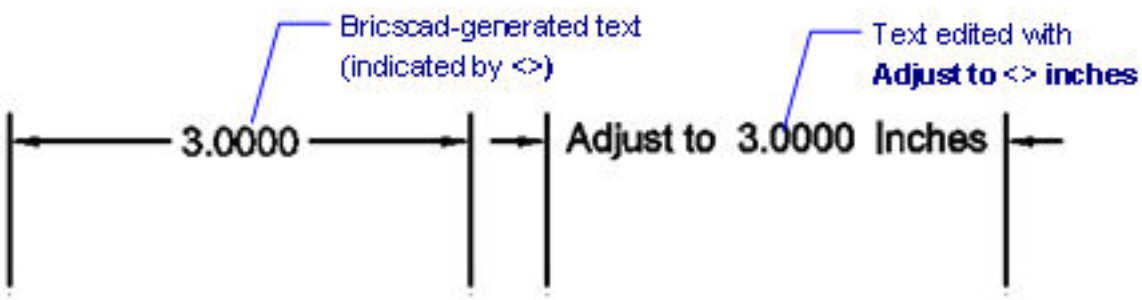


Figure 6.781

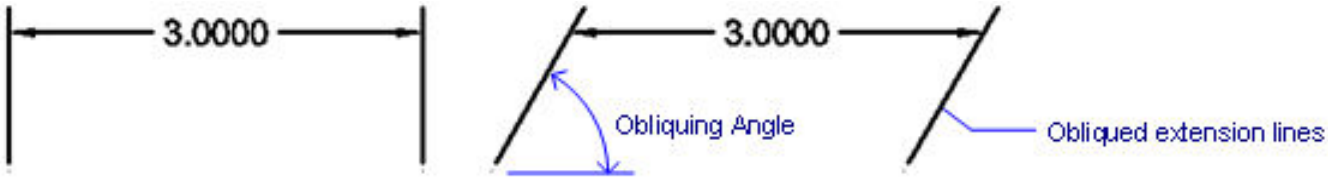


Figure 6.782

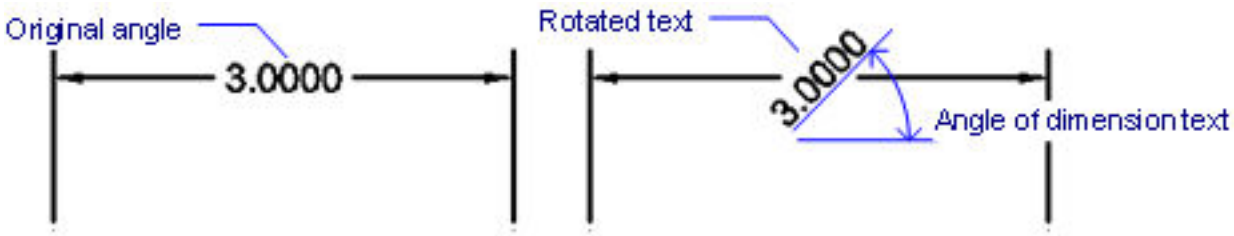
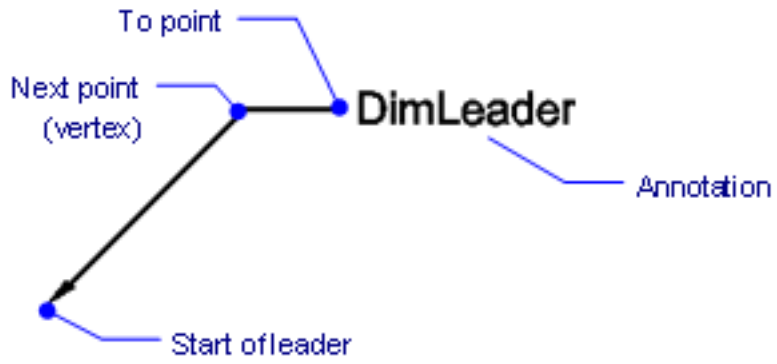


Figure 6.783



Figure 6.784



6.11.13 DimLeader

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws leaders. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdF

Accessing the Command


command bar: **dimleader** aliases: **lead, leader** ribbon: **Annotate | Dimension | Leader** : **DimLeader Prompts you in the command bar:** Start of leader: **(Choose the starting point of the leader)** Next point: **(Choose the location of the first vertex)** To point [Format/Undo/Annotation] <Annotation>: **(Choose another vertex, enter an option, or press Enter to enter text)** First line of annotation text <options>: **(Enter text, or press Enter to specify annotation options)** Next line of annotation text: **(Enter additional text, or else press Enter to exit the command)**

Command Options

Option	Description
Start of Leader	Locates the leader's arrowhead, and starts the leader line.
Next Point	Locates the first end point of the leader, or vertex.
To Point	Locates the next end point.
Format	Formats the leader line.Prompts you: **[Arrow/None/SPline/STraight/Exit] <Exit>** - **(choose an option)**: • **Arrow** - draws the arrowhead • **None**- does not draw the arrowhead • **Spline**- draws the leader line as a spline • **Straight**- draws the leader line as straight segments • **Exit**- exits the Format options
Undo	Undoes the last segment.

Annotation	Adds the leader's annotation.Prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **First line of annotation text <options>** - **(type the annotation text at the command prompt or press Enter to specify the annotation options.)** 2. **Next line of annotation text** - **(type the next line of the annotation text or press Enter to exit the command)**
Options	Specifies the type of annotation.Prompts you: **Dimension text options [Block/Copy/None/Tolerance/Mtext] <Mtext>**- **(enter an option)**: • **Block** - uses a block as the annotation; prompts you: **? to list blocks in drawing/~ to open the file dialog/<Block to insert>**- **(type the name of a block in the drawing or enter ? (to list the names of blocks in the current drawing, or enter~ to displays a file dialog box from which you can choose a .dwg file to use as the block for the annotation.)** • **Copy**- copies an entity in the drawing to use as the annotation; prompts you: **Select an entity** - **(select an mtext, text, block reference or tolerance entity.)** • **None**- eaves out the annotation • **Tolerance**- uses tolerance specification as the annotation; displays the Geometric Tolerance dialog box; see the**Tolerance (6.29.17)** command • **Mtext** - uses MText as the annotation and displays the Multiline Text dialog box; see the**MText (6.21.35)** command

Procedures

- To create a leader and an annotation (1.21.13)
- To create a leader only (1.21.13)

Grips Editing

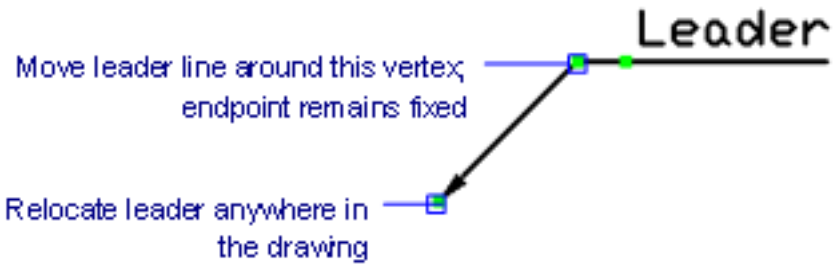
Leaders can be edited directly through grips:

1. Select the leader. Notice the grips.
2. Drag the grips to perform the following actions:

- Drag a grip on a vertex of the leader line to stretch the line segments on either side of the vertex.
- Drag the grip at the arrowhead to move the leader anywhere in the drawing. (The text is independent of the leader line.)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image788.png})

Figure 6.785



Related Commands

Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. QLeader (6.25.1) - draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders. MLeader (6.21.20) - creates multileader entities using the current multileader style. DimArc (6.11.5) - Places arc length dimensions. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.786

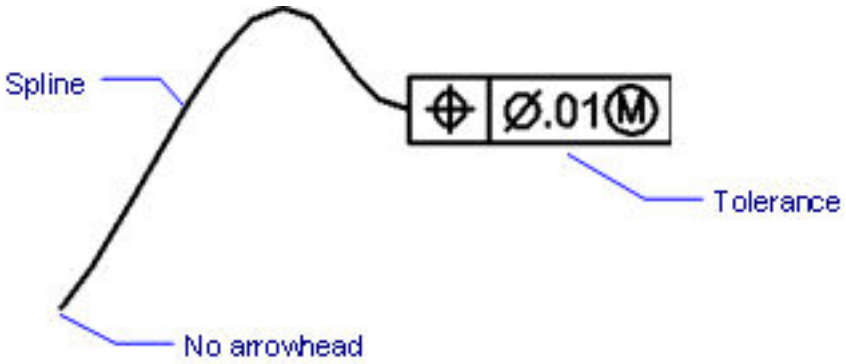


Figure 6.787



6.11.14 DimLinear

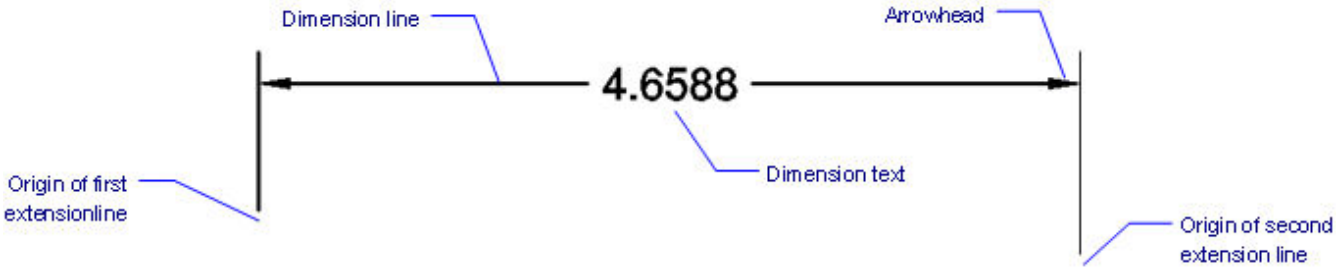
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places linear dimensions horizontally, vertically, or rotated.

Accessing the Command



command bar: **dimlinear** aliases: **dimlin, dli, dimhorizontal, dimrotated, dimvertical** menu bar: **Dimension | Linear** toolbar: **Dimensions |** ribbon:**Draw | Dimensions | Leader** : **DimLinear Prompts you in the command bar:** Origin of first extension line <Select entity>: **(Pick a point, or else press Enter to dimension an entity directly)** Origin of second extension line: **(Pick another point)** [Angle/Text/Horizontal/Vertical/Rotated]: **(Position the dimension line, or else enter an option)** Dimension text = xx.xx (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cr

Figure 6.788



Command Options

Option	Description
Select entity	Dimensions an entity.Prompts you: 1. **Select entity to dimension** - **(select a line, polyline segment, arc or circle to dimension; other entities cannot be dimensioned with this option.)** 2. **[Angle/Text/Horizontal/Vertical/Rotated]** - **(position the dimension line or choose an option.)** (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Angle	Specifies the angle of the text; useful for keeping text horizontal with inclined dimension lines.Prompts you: **Angle of dimension text** - **(specify the rotation angle of the text.)**
Text	Specifies the text; useful for overriding the default measurement generated by the program.Prompts you: **Dimension text** - **(specifies the replacement text.)**
Horizontal	Forces the dimension to be horizontal.prompts you: **Location of dimension line [Angle/Text]** - **(positions the dimension line or choose an option.)**
Vertical	Forces the dimension to be vertical.prompts you: **Location of dimension line [Angle/Text]** - **(positions the dimension line or choose an option.)**
Rotated	Rotates the dimension by a specified angle.Prompts you: **Angle of dimension line <0.0>** - **(specifies the angle of the dimension line.)**

Procedures

- To create a horizontal or vertical linear dimension (1.21.8)
- To create an aligned linear dimension (1.21.8)
- To create a rotated linear dimension (1.21.8)

Grips Editing

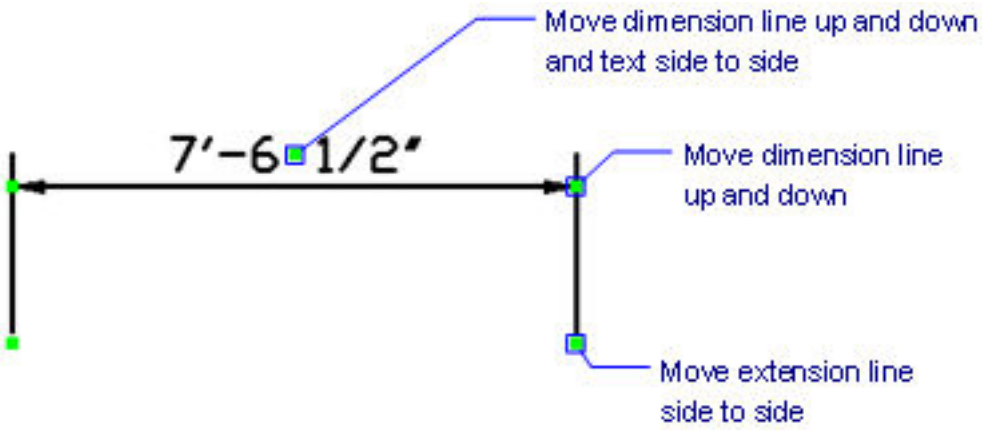
Linear dimensions can be edited directly through grips:

1. Select the linear dimension. Notice the grips.
2. Drag the grips to perform the following actions:

- Drag either grip on the dimension line to move the dimension line; the extension lines change length.
- Drag the grip on the text to relocate it; the dimension line moves along, and a leader line is added, if required.
- Drag either grip on the extension lines to move the extension line; the dimension line changes length.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image516.jpg})

Figure 6.789



Related Commands

DimArc (6.11.5) - Places arc length dimensions. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. AiDimFlipArrow (6.5.2) - flips individual arrows of a dimension. AiDimPrec - changes the displayed precision of existing dimensions. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.790

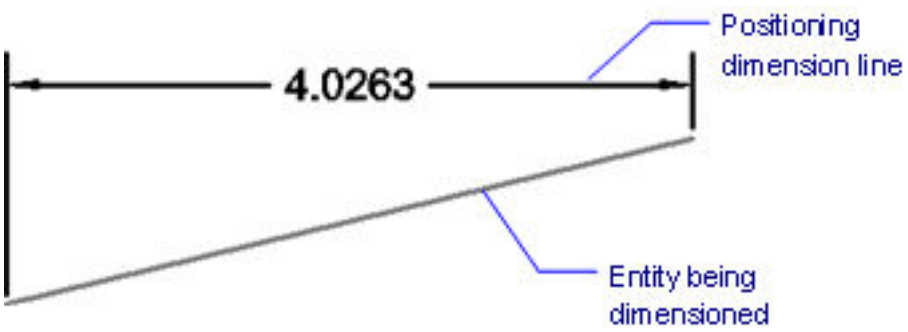


Figure 6.791



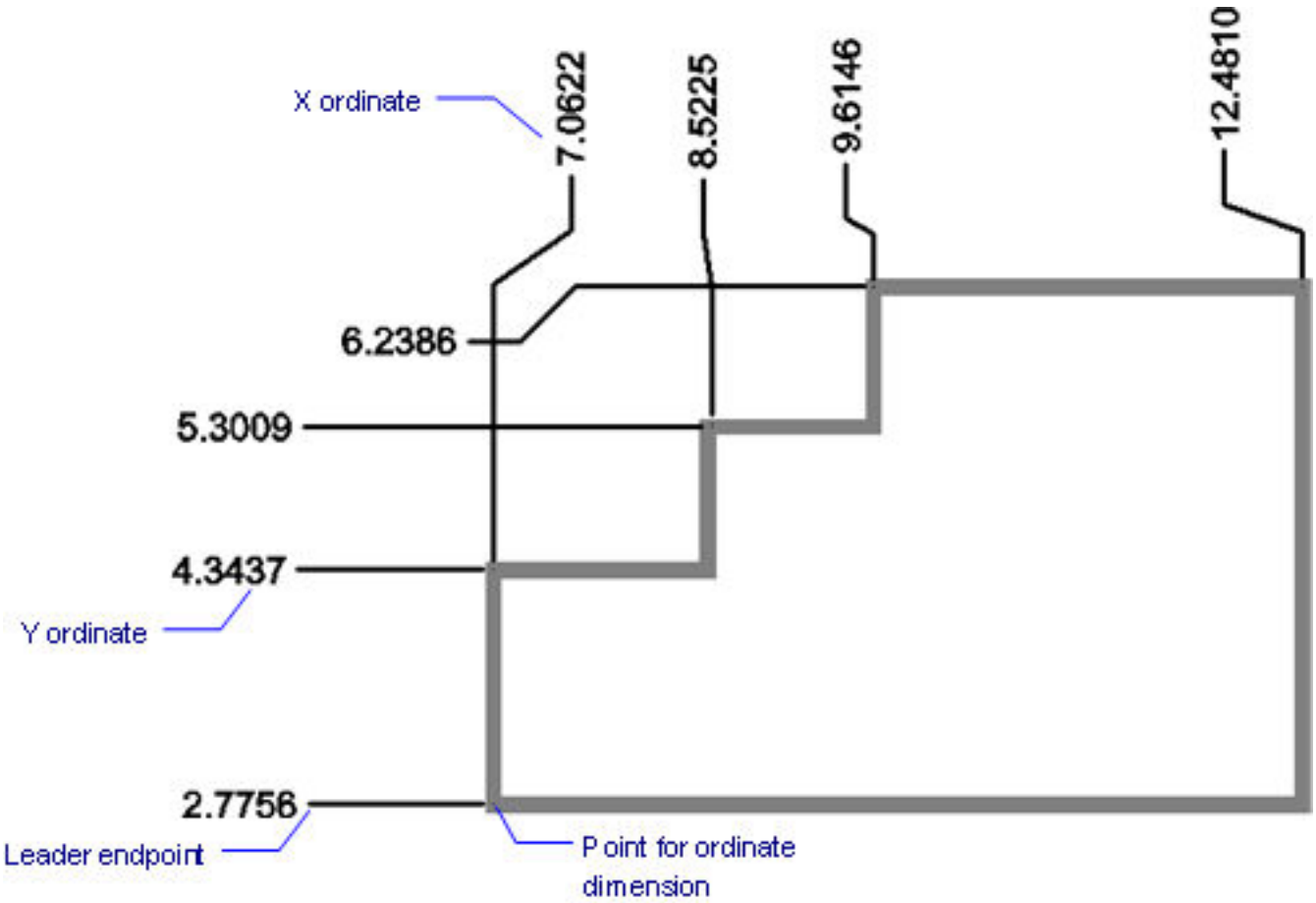
6.11.15 DimOrdinate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Dimensions in the x and y distances from the current UCS origin to the specified point in the drawing.

Accessing the Command


command bar: **dimordinate** aliases: **dor, dimord** menu bar: **Dimension | Ordinate** toolbar: **Dimensions |** 
ribbon:**Draw | Dimensions | Continue | Ordinate** : **DimOrdinate Prompts you in the command bar:** Select point for ordinate dimension: **(Pick a point)** Leader endpoint [Xdatum/Ydatum/Text/Angle]: **(Pick an endpoint for the leader, or enter n option.)** Dimension text: = xx.xx The distance is measured from the origin (0,0) of the current UCS. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image128.jpg})

Figure 6.792



Command Options

Option	Description
X datum	Forces the ordinate dimension to measure the x distance; the distance is measured from the origin of the current UCS.
Y datum	Forces the ordinate dimension to measure the y distance; the distance is measured from the origin of the current UCS.
Angle	Specifies the angle of the text; useful for keeping text horizontal with inclined dimension lines.Prompts you: **Angle of dimension text -(specify the rotation angle of the text.)**
Text	Specifies the text; useful for overriding the default measurement generated by BricsCAD.Prompts you: **Dimension text** - **(specify the replacement text.)**

Procedures

- To create an ordinate dimension (1.21.12)

Grips Editing

Ordinate dimensions can be edited directly through grips:

1. Select an ordinate dimension. Notice the grips.
2. Drag the grips to perform the following actions:
   - Drag the grip at the end point to relocate the endpoint; the leader and text remain fixed.
   - Drag the grip on the leader to relocate it and the associated text.
   - Drag the grip at the center of the text to relocate the entire dimension anywhere in the drawing.

{\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image515.jpg}}

Figure 6.793

Related Commands

DimArc (6.11.5) - Places arc length dimensions. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.794



6.11.16 DimOverride

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Overrides the values of the current dimension style. This command is superseded by the Override function of the **DimStyle (6.11.20)** command.

Accessing the Command

command bar: **dimoverride** aliases: **dor, dimover** : **DimOverride Prompts you in the command bar:** Select Dimension variable to override or [Clear]: **(Enter the name of a dimension variable, or else type C)** DimVar is now <"xx.xx">. New value: **(Enter a new value the dimensionvariable)** Dimension variable to override: **(Enter the name of another dimension variable, or else press Enter to continue)** Select dimensions to use new value(s): **(Choose one or more dimensions)** Entities in set: 1 Select dimensions to use new value(s): **(Choose more dimensions, or eles press Enter to exit command)**

Command Options

Option	Description
Clear Overrides	Resets the values of dimension variables, clearing the overrides. Prompts you: 1. **Select dimensions to clear overrides - (choose the dimensions whose settings should be reset.) Entities in set: 1** 2. **Select dimensions to clear overrides - (choose more dimensions or press Enter to exit the command.)**

Related Commands

DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimArc (6.11.5) - Places arc length dimensions. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. AiDimFlipArrow (6.5.2) - flips individual arrows of a dimension. AiDimPrec - changes the displayed precision of existing dimensions. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette.

Figure 6.795



6.11.17 DimRadius

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Dimensions the radii of arcs. polyarcs, and circles.

Accessing the Command

 command bar: **dimradius** aliases: **dimrad, dra** menu bar: **Dimension | Radius** toolbar: **Dimensions |**  ribbon:**Draw | Dimensions | Circle | Radius** : **DimRadius Prompts you in the command bar:** Select arc or circle to dimension: **(Choose an arc, a polyline arc, or a circle)** Dimension text = xx.xx Location of dimension line [Angle/Text]:**(Position the dimension, or enter an option)** A circle with radius dimension placed outside: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image123.jpg}) An arc with radius dimension placed inside: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image

Figure 6.796

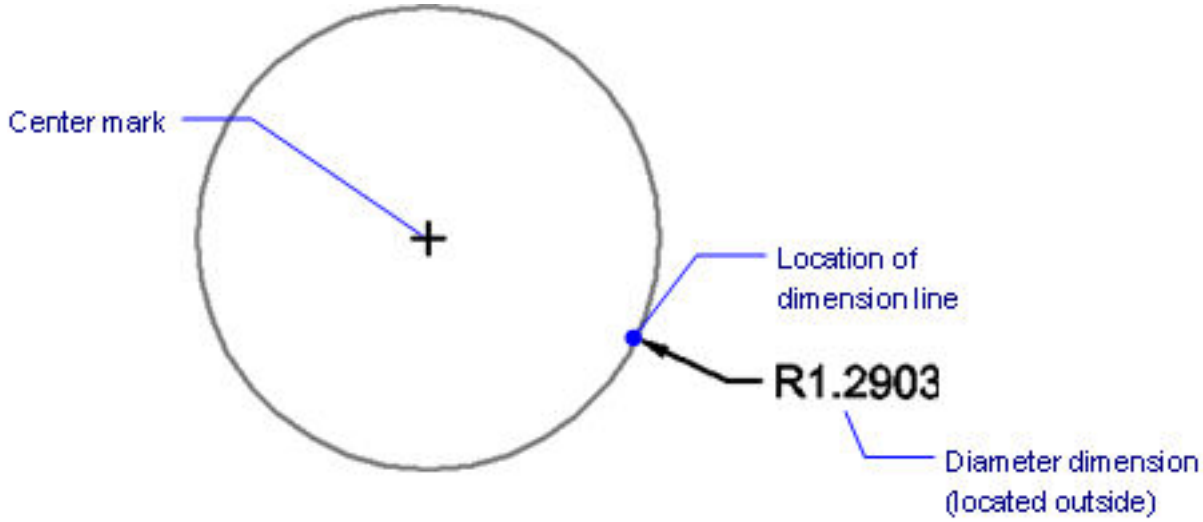
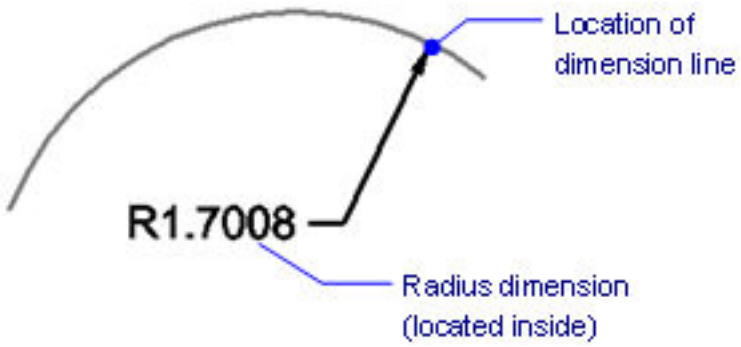


Figure 6.797



Command Options

Option	Description
Angle	Specifies the angle of the text. Prompts you: **Angle of dimension text** - **(specify the rotation angle of the text.)**
Text	Specifies the text; useful for overriding the default measurement generated by the program. Prompts you: **Dimension text** - **(specifies the replacement text.)**

Procedures

- To create a radial dimension (1.21.10)
- Creating center marks or center lines (1.21.10)
- To create a diametrical dimension (1.21.10)

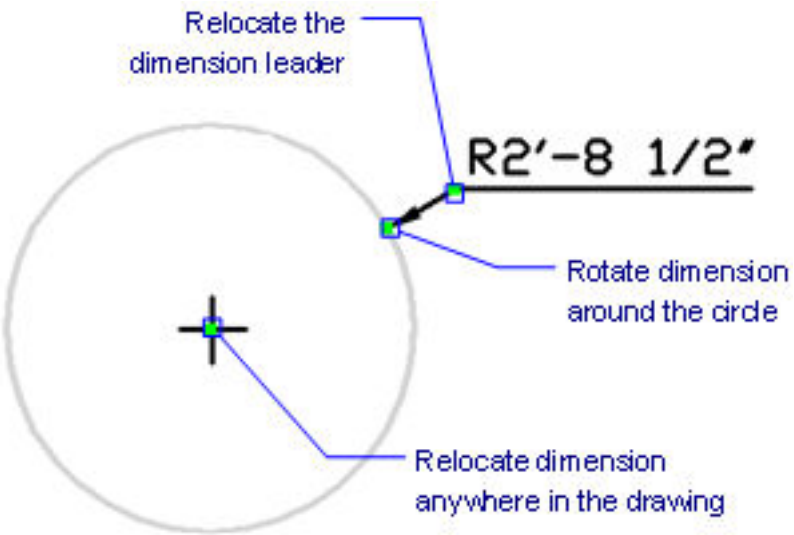
Grips Editing

Radial dimensions can be edited directly through grips:

1. Select the radius dimension. Notice the grips.
2. Drag the grips to perform the following actions:
   - Drag the grip on the circle to rotate the leader and text around the circle.
   - Drag the grip on the leader to relocate it and the associated text.
   - Drag the grip at the center of the circle to relocate the dimension anywhere in the drawing.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image514.jpg})

Figure 6.798



Related Commands

DimArc (6.11.5) - Places arc length dimensions. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. AiDimFlipArrow (6.5.2) - flips individual arrows of a dimension. AiDimPrec - changes the displayed precision of existing dimensions. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.799



6.11.18 DimReassociate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Reassociates or associates dimensions to entities or points on entities.

Accessing the Command

command bar: **dimreassociate** menu: **Dimension | Reassociate Dimensions** : **DimReassociate Prompts you in the command bar:** Select entities or [Disassociated]: **(Select one or more dimension entities or type D, then press Enter.)** Entities in set: 1 Select entities or [Disassociated]: **(Select more dimension entities or type D, then press Enter or press Enter to stop selecting.)** Dimension entities in the selection set highlight one by one and you are prompted to specify association points or entities for the currently highlighted dimension entity. A marker displays at the current position of the edited dimension point:

- Indicates a non-associated dimension point.
- Indicates an associated dimension point.

Depending on the dimension entity types in the selection set, the following prompts display:

Linear (rotated and aligned dimensions): Specify first extension line origin or [Select entity] <next>: **(Use an appropriate entity snap to specify the first extension line origin, type S then press Enter to select an entity or press Enter to skip to the next prompt.)** Specify second extension line origin <next>: **(Use an appropriate entity snap to specify the second extension line origin or press Enter to skip to the next prompt.)**

Diameter Select arc, circle or polyline arc <next>: (**Select an arc, circle or polyline arc segment or press Enter to skip to the next prompt.)**

Radius Select arc, circle or polyline arc <next>: **(Select an arc, circle are polyline arc segment or press Enter to skip to the next prompt.)**

Angular 2 Lines: Select first line <next>: **(Select a line or press Enter to skip to the next prompt.)** Select second line <next>: **(Select a line or press Enter to skip to the next prompt.)** 3 Points: Specify angle vertex or [Select arc, circle or polyline arc] <next>: **(Use an appropriate entity snap to specify the angle vertex or press Enter to skip to the next prompt.)** Specify first angle endpoint <next>: **(Use an appropriate entity snap to specify the first angle endpoint or press Enter to skip to the next prompt.)** Specify second angle endpoint <next>: **(Use an appropriate entity snap to specify the second angle endpoint or press Enter to skip to the next prompt.)** Arc or Circle: Specify angle vertex or [Select arc, circle or polyline arc] <next>: **(Type S, then press Enter or press Enter to skip to the next prompt.)** Select arc, circle or polyline arc <next>: **(Select an arc, polyline arc segment or circle or press Enter to skip to the next prompt.)**

Ordinate Specify feature location <next>: **(Use an appropriate entity snap to specify the new dimension point or press Enter to skip to the next prompt.)**

Leader Specify leader association point <next>: **(Use an appropriate entity snap to specify the new leader association point or press Enter to skip to the next prompt.)**

Command Options

Option	Description
Disassociated	Prompts to reassociate all disassociated dimension entities in the drawing. All disassociated dimensions highlight one by one.

Related Commands

DimDisassociate (6.11.11) - removes the associativity from the selected dimension entities. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimAngular (6.11.4) - places angular dimensions. DimOrdinate (6.11.15) - places x and y ordinate dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. BmBalloon (6.8.1) - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette.

Figure 6.800



6.11.19 DimRegen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Updates associative dimensions (short for ”dimension regeneration”).

Accessing the Command

command bar: **dimregen** menu bar:**Dimension | Reassociate Dimenions** : **DimRegen** Displays no prompt in the command bar.

Command Options

No options.

Figure 6.801



6.11.20 DimStyle

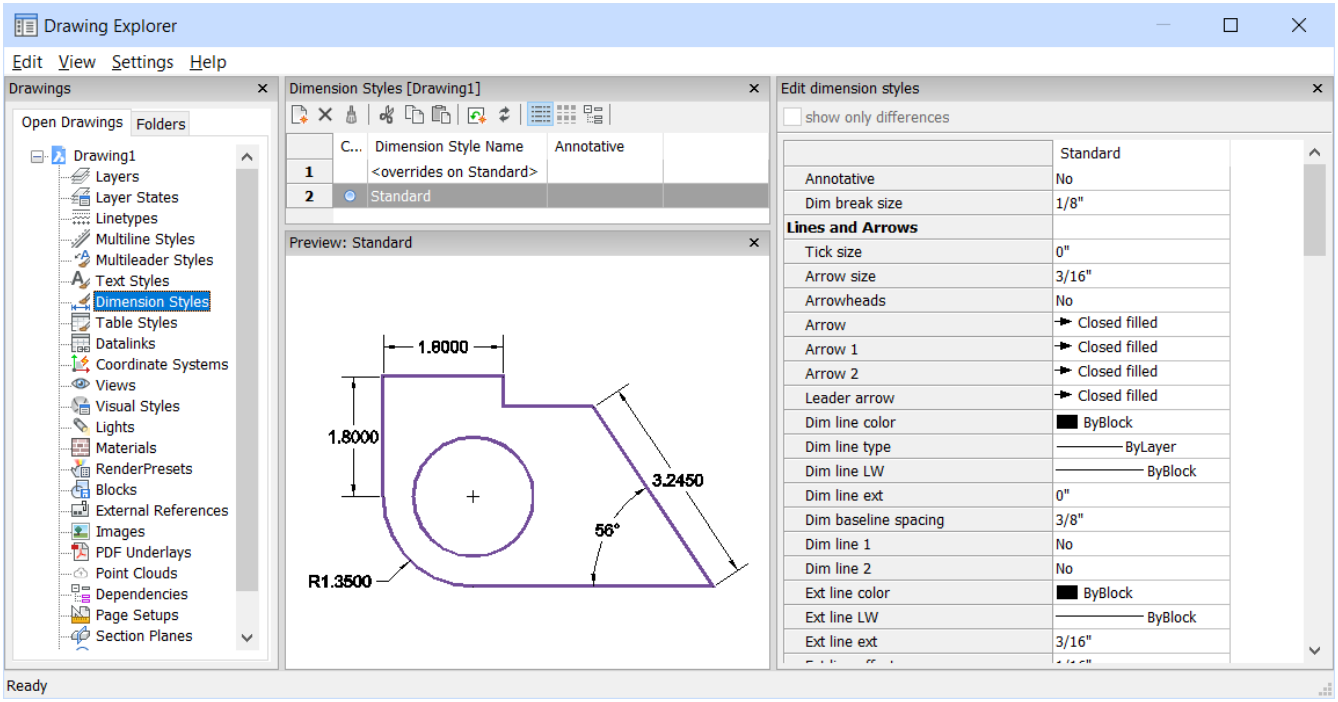
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and modifies dimension styles, through the Drawing Explorer.

Accessing the Command


command bar: **dimstyle** aliases: **d, ddim, dimsty, ds, dst, expdimstyles, setdim** menu bar: **Tools | Drawing Explorer | Dimension Styles...** status bar: **Standard** (or name of other current dimension style) toolbar: **Dimensions** | ribbon:**Draw | Dimensions | Dimension Variable Status** : **DimStyle** Displays the Drawing Explorer at the Dimension Styles section: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/dimstyledlg.png}) Create or edit dimension styles, and then click **X**.

- **<Overrides>** - overrides the values of individual dimension variables without creating a new style
- **Standard** - lists the dimension settings for the dimension style named ”Standard”

Figure 6.802



Command Options

Option	Description
Name	Names of the dimension style; all styles can be renamed and deleted, except for the Standard style.
Current	Indicates whether the style is current or not.
Annotative	Sets the annotative property of the dimension style.
Dim break size	Sets the value the gap created by the DimBreak (6.11.7) command.
Show only differences	If multiple styles are selected, compares the selected styles (1.21.4) and shows the differences only.

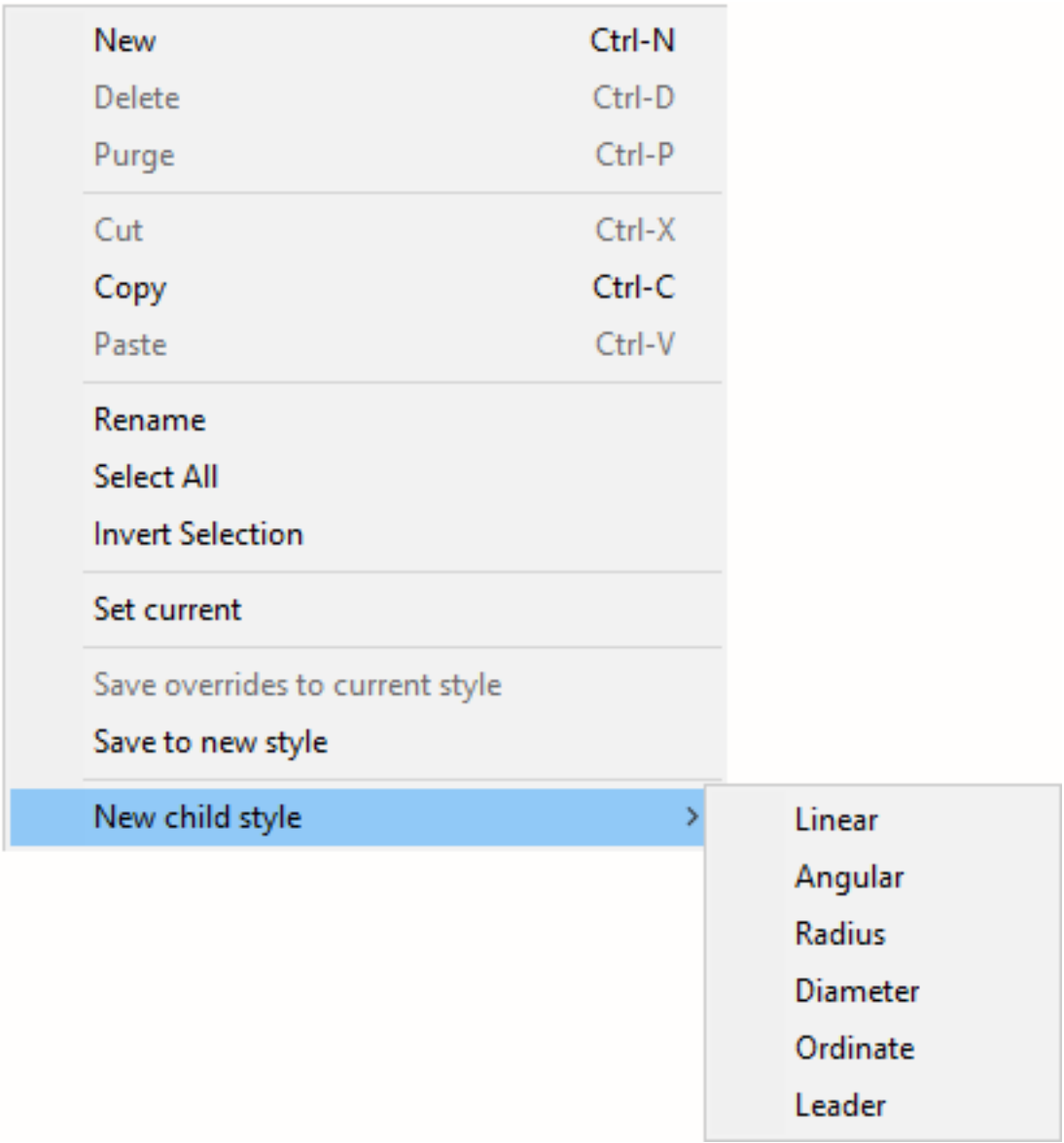
Dimension Child Styles A dimension style can have up to 6 child styles: **Linear**, **Angular**, **Radius**, **Diameter**, **Ordinate**, and **Leader**. If defined, then the child style is used for the corresponding dimension type. Right click the dimension style name, then choose **New child** style in the context menu and select a child style type. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DimStyleChildStyles.png}) Dimension child styles share all settings with the parent style, except for the properties that are defined explicitly differently.

Lines and Arrows Options
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DimLinesAndArrowsSettings.png})

Arrow 1	Specifies the style of arrowhead for the starting end of the dimension line; choose from one of the following types: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/D
Arrow 2	Specifies the style of arrowhead for the other end of the dimension line.
Arrow Size	Specifies the length of the arrowhead; the width is scaled proportionately.
Leader Arrow	Specifies the style of arrowhead for the starting end of the leader line.
Dim Line Color	Specifies the color of the dimension line; choose from: • Any Index Color • Any True Color • ByBlock • ByLayer

Ext Line Color	Specifies the color of the dimension line; choose from: • Any Index Colo. • Any True Color • ByBlock • ByLayer See the Color (6.9.33) command.
Ext Line type	Specifies the line type of both extension lines; choose from any line type loaded into the current drawing. To access additional linetypes, click **Load**, and then choose one from the Load Linetypes dialog box.
Ext Line LW	Specifies the lineweight of the extension line; choose from any lineweight supported by the program.
Ext Line Ext	Specifies the distance that the extension lines extend beyond the dimension line.
Ext Line Offset	Specifies the offset distance between the object and the start of the extension lines.
Ext Line 1	Toggles the display of the first extension line.
Ext Line 2	Toggles the display of the second extension line.
Ext line type 1	Specifies the line type of the first extension line; choose from any line type loaded into the current drawing. To access additional linetypes, click **Load**, and then choose one from the Load Linetypes dialog box.
Ext line type 2	Specifies the line type of the second extension line; choose from any line type loaded into the current drawing. To access additional linetypes, click **Load**, and then choose one from the Load Linetypes dialog box.
Ext line fixed length	Specifies the length of fixed-length extension lines.
Center Mark	Specifies the type of center mark: • Mark • Line • None
Center Mark Size	Specifies the size of the center mark.
Arc length symbol	Specifies the location of the arc length symbol: • Preceding dimension text • Above dimension text • Not displayed
Jog angle	Specifies the angle of the jog; default is 45 degrees.Enter another angle.

Figure 6.803


New Ctrl-N
Delete Ctrl-D
Purge Ctrl-P
Cut Ctrl-X
Copy Ctrl-C
Paste Ctrl-V
Rename
Select All
Invert Selection
Set current
Save overrides to current style
Save to new style
New child style
Linear
Angular
Radius
Diameter
Ordinate
Leader

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DimTextSettings.png})

Text Style	Specifies the style for dimension text; can use only styles created with the Style (6.27.53) command.
Text Color	Specifies the color of the text; choose from: • Any Index Color • Any True Color • ByBlock • ByLayer
Text Fill	Determines whether the dimension text has a rectangular background filled with color. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Choose from: • **No fill** - does not apply a background fill • **Background** - uses the background color of the drawing, usually white or black • **Color** - uses the color specified by the Text Fill Color option
Text Fill Color	Specifies the background fill color when Text Fill is set to Color.Choose a color from the drop list, or click **Select Color** to access the Select Color dialog box.
Text Height	Specifies the height of the text.
Draw Frame Around Text	Toggles a rectangle around the text.
Text Pos Vert	Justifies the text vertically relative to the dimension line: • **Centered** - centers the text on the dimension line • **Above** - places the text above the dimension line • **Outside** - places text outside of the extension lines • **JIS** - generally places text above the dimension line, according to the Japanese Industry Standard
Text Pos Hor	Justifies the text horizontally relative to the extension lines: • **Centered** - centers the text between the extension lines • **First Extension Line** - places the text near the first extension line • **Second Extension Line** - places the text near the second extension line • **Over First Extension** - places the text over the first extension line • **Over Second Extension** - places the text over the second extension line
Text Offset	Specifies the size of gap between the dimension line and the text.

Text Inside Align	Justifies text when located between extension lines: • **Aligned with Dimension Line** - aligns text with dimension line • **Horizontal** - forces text to be always horizontal
Text Outside Align	Justifies text when located outside of the extension lines: • **Aligned with Dimension Line** - align text with dimension line • **Horizontal** - forces text to be always horizontal

Fit Options
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DimFitSettings.png})

Arrow and Text Fit	Specifies where text and arrows should be located when there is insufficient space for both between the extension lines: • **Text and Arrows** - forces both text and arrows between the extension lines • **Arrows Only** - forces arrows to stay within extension lines; moves text outside when there is insufficient space • **Text Only** - forces text to stay within extension lines; moves arrows outside when there is insufficient space • **Best F**it - requires the program to figure out where to place text and arrows, depending on the space between extension lines
Text Inside	Toggles whether the text is forced between extension lines: • **On** - forces text between extension lines • **Off** - draws text outside, if insufficient space between extension lines
Dim Line Inside	Toggles whether the dimension line is forced between extension lines: • **On** - forces dimension line always between extension lines • **Off** - draws dimension line outside, if insufficient space between extension lines
Text Movement	Specifies what happens when text is moved away from its default location: • **Keep dim line with text** - moves dimension line with the text, stretching the extension lines. • **Move text, add leader** - draws a leader between the text and the dimension line. • **Move text, no leader** - does not draw a leader.

Dim Scale Overall	Specifies the overall scale factor for dimensions; this affects the size of arrows and text only. This setting is not editable for annotative dimension styles. It is recommended to set Dim Scale Overall = 1 for annotative dimension styles.
Place Text Manually	Toggles whether the user must always specify the text location when creating dimensions.
Dim Line Forced	Forces the dimension line to be always drawn; forces leaders to be drawn with the **DimDiameter** and **DimRadius** commands.

Primary Units Options

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DimPrimaryUnitsSettings.png})

Dim Units	Specifies the display units for dimensions: • **Scientific** - scientific notation, such as 1.2345E+01 • **Decimal** - metric, such as 1.2345 • **Engineering** - feet and decimal inches, such as 1'-2.3456" • **Architectural** - feet and fractional inches, such as 1'-2 1/16" • **Fractional** - fractional inches only; no feet, such as 14 1/16" • **Windows Desktop** - uses the units set by Windows
Dim Precision	Specifies the precision of units, either decimal places or fractional accuracy.
Fractional Type	Specifies how fractions are stacked: • **Horizontal** - stacks fractions vertically, with a horizontal separator line • **Diagonal** - stacks diagonally, with a diagonal separator line • **None** - does not stack fractions, placed horizontally with a slash separator
Decimal Separator	Specifies the character used to indicate the decimal point; can be any character. North American countries use the period; European countries use the comma.
Dim Round	Specifies the rounding of decimal numbers; range is none to 8 decimal places.
Dim Prefix	Specifies prefix text that appears in front of the dimension text, if any.
Dim Suffix	Specifies suffix text that appears after the dimension text, if any.
Dim sub-units suffix*	Specifies suffix text in case the sub unit is used. For example, enter 'cm' to display .96 as 96cm.
Dim Scale Linear	Specifies the multiplier for dimension values, such as 25.4 for changing inches to millimeters.
Dim sub-units scale*	Sets the scale factor for the sub-units. E.g. enter 10 if the drawing unit is cm, and the sub-unit suffix is mm.
Suppress Leading Zeros	Toggles the display of zeros in front of the decimal point; for example, 0.23 is displayed as: • **On** - suppresses the leading zero, such as .23 • **Off** - permits the leading zero, such as 0.23

Suppress Trailing Zeros	Toggles the display of zeros after the decimal point; for example, 1.2300 is displayed as: • **On** - suppresses trailing zeros, such as 1.23 • **Off** - allows trailing zeros, such as 1.2300
Suppress Zero Feet	Toggles the display of zero feet; for example, 0'-3" is displayed as: • **On** - suppresses zero feet, such as 3" • **Off** - allows zero feet, such as 0'-3"
Suppress Zero Inches	Toggles the display of zero inches; for example, 1'-0" is displayed as: • **On** - suppresses the display of zero inches, such as 1' • **Off** - allows zero inches, such as 1'-0"
Dim Angle Units	Specifies the format of units in angular dimensions: • **Decimal Degrees** - 360 degrees in a circle; for example, 123.45 degrees • **Deg/Min/Sec** - degrees, minutes, seconds; for example, 123d 12' 45.67" • **Grads** - 400 grads in a circle; for example, 230g • **Rads** - 2pi radians in a circle; for example, 1.5r
Dim Angle Precision	Specifies the number of decimal places; range is from 0 to 8.
Suppress Angle Leading Zeros	Toggles the display of zero degrees; for example, 0.1234 degrees is displayed as: • **On** - suppresses zero degrees, such as .1234 • **Off** - allows zero degrees, such as 0.1234
Suppress Angle Trailing Zeros	Toggles the display of zeros after degrees; for example, 0.1200 degrees is displayed as: • **On** - suppresses trailing zeros, such as 0.12 • **Off** - allows trailing zeros, such as 0.1200

* Suppress leading zeros must be set to **Yes** to display dimension distances less than one unit in sub units

Alternate Units Options

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DimAlternateUnitsSettings.png})

Alternate Enabled	Toggles the display of alternative units: • **On** - second, alternative units displayed to the right of primary units • **Off** - only primary units are displayed

Alt Units	Specifies the display units for dimensions: • **Scientific** - scientific notation, such as 1.2345E+01. • **Decimal** - metric, such as 1.2345 • **Engineering** - feet and decimal inches, such as 1’-2.3456” • **Architectural** - feet and fractional inches, such as 1’-2 1/16” • **Fractional** - fractional inches only; no feet, such as 14 1/16” • **Windows Desktop** - uses the units set by Windows
Alt Precision	Specifies the precision of units, either decimal places or fractional accuracy.
Alt Scale Factor	Specifies the multiplier for alternate values, such as 25.4 for showing millimeters (alternate units) next to inches (primary units).
Alt Round	Specifies the rounding of alternate numbers; range is none to 8 decimal places.
Alt Prefix	Specifies prefix text that appears in front of the alternate text, if any.
Alt Suffix	Specifies suffix text that appears in after the alternate text, if any.
Alt Leading Zeros	Toggles the display of zeros in front of the decimal point; for example, 0.23 is displayed as: • **On** - suppresses the leading zero, such as .23 • **Off** - permits the leading zero, such as 0.23
Alt Trailing Zeros	Toggles the display of zeros in front of the decimal point; for example, 1.2300 is displayed as: • **On** - suppresses trailing zeros, such as 1.23 • **Off** - allows trailing zeros, such as 1.2300
Alt Zero Feet	Toggles the display of zero feet; for example, 0’-3” is displayed as: • **On** - suppresses zero feet, such as 3” • **Off** - allows zero feet, such as 0’-3”
Alt Zero Inches	Toggles the display of zero inches; for example, 1’-0” is displayed as: On • **On** - suppresses the display of zero inches, such as 1’ • **Off** - allows zero inches, such as 1’-0”

Tolerances Options

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DimTolerancesSettings.png})

Tolerance Display	Toggles the display of tolerance text.
Tolerance Precision	Specifies the display precision of tolerance text; range is 0 to 8 decimal places or 1/1 to 1/256 inch.
Tolerance Limit Lower	Specifies the value of the upper tolerance.
Tolerance Limit Upper	Specifies the value of the lower tolerance.

Tolerance Text Height	Specifies the height of tolerance text.
Tolerance Pos Vert	Locates the tolerance text relative to the dimension text: • **Bottom** - aligns tolerance text with the bottom of the dimension text • **Middle** - aligns tolerance text with the middle of the dimension text • **Top** - aligns tolerance text with the top of the dimension text
Tolerance Suppress Leading Zeros	Toggles the display of zeros in front of the decimal point.
Tolerance Suppress Trailing Zeros	Toggles the display of zeros behind the decimal point.
Tolerance Suppress Zero Feet	Toggles the display of zero feet.
Tolerance Suppress Zero Inches	Toggles the display of zero inches.
Alt Tolerance Precision	Specifies the number of decimal places for tolerances in alternate units.
Alt Tolerance Suppress Leading Zeros	Toggles the display of zeros in front of the decimal point.
Alt Tolerance Suppress Trailing Zeros	Toggles the display of zeros behind the decimal point.
Alt Tolerance Suppress Zero Feet	Toggles the display of zero feet.
Alt Tolerance Suppress Zero Inches	Toggles the display of zero inches.

Status Bar Menu

Right-click Standard (or other dimension style name) on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image792.png}) (\vref {fig:C:/Dev/Task-2448/images/static

Shortcut Menu	Description
Standard	Chooses the default dimension style.When the drawing contains additional dimension styles, their names are listed here.
Properties	Opens the Dimension Styles section of the Drawing Explorer.

Figure 6.804

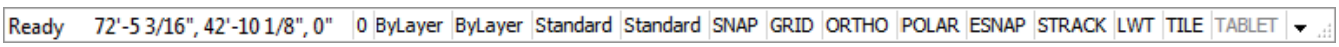
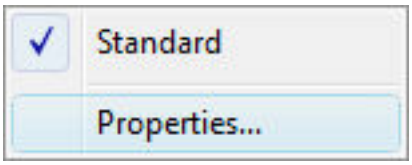


Figure 6.805



Procedures

- Exploring dimension styles (1.21.4)
- Comparing dimension styles (1.21.4)
- To list the current status of the dimension variables (1.21.5)
- To create a new dimension style (1.21.6)
- To edit a dimension style (1.21.6)
- To define dimension style overrides (1.21.6)
- To rename a dimension style (1.21.6)
- To set a dimension style current (1.21.6)
- Using the status bar to set a dimension style current (1.21.6)
- To copy a dimension style between drawings (1.21.6)
- To move a dimension style between drawings (1.21.6)
- To delete a dimension style (1.21.6)

Related Commands

AiDimStyle - creates Dimension Styles based on the properties of selected Dimension, Tolerance or Leader entities. -Dimstyle (6.11.21) - creates and modifies dimension styles at the command line. Explorer (6.13.14) - opens the Drawing Explorer dialog box.

Figure 6.806

Lines and Arrows	
Tick size	0 mm
Arrow size	180 mm
Arrowheads	Yes
Arrow	Closed filled
Arrow 1	Closed filled
Arrow 2	Closed filled
Leader arrow	Closed filled
Dim line color	ByBlock
Dim line type	ByLayer
Dim line LW	ByBlock
Dim line ext	100 mm
Dim baseline spacing	380 mm
Dim line 1	Yes
Dim line 2	Yes
Ext line color	ByBlock
Ext line LW	ByBlock
Ext line ext	180 mm
Ext line offset	180 mm
Ext line 1	Yes
Ext line 2	Yes
Ext line type 1	ByLayer
Ext line type 2	ByLayer
Ext line fixed	No
Ext line fixed length	1 mm
Center mark	Mark
Center mark size	90
Arc length symbol	Preceding dimension tex
Jog angle	45

Figure 6.807

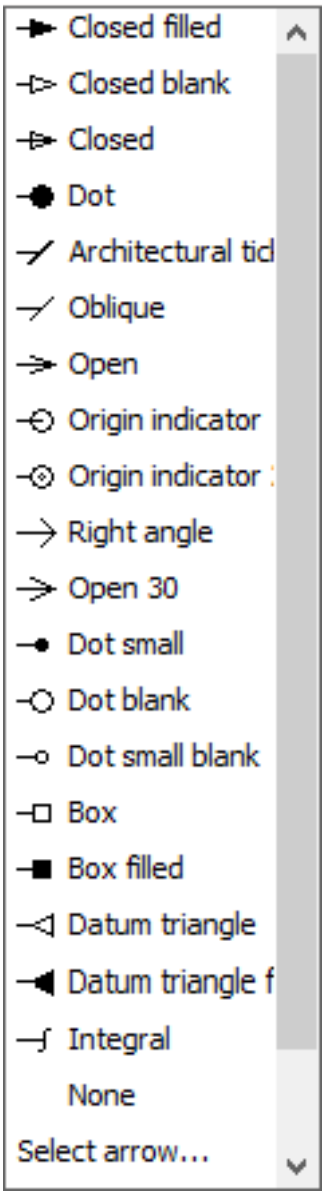

Closed filled
Closed blank
Closed
Dot
Architectural tick
Oblique
Open
Origin indicator
Origin indicator 2
Right angle
Open 30
Dot small
Dot blank
Dot small blank
Box
Box filled
Datum triangle
Datum triangle filled
Integral
None
Select arrow...

Figure 6.808

Text	
Text style	Standard
Text color	ByBlock
Text fill	No fill
Text fill color	ByBlock
Text height	180 mm
Draw frame around text	No
Text position vertical	Above
Text position horizontal	Centered
Text offset	90
Text vertical offset	180
Text inside align	No
Text outside align	Yes
Text view direction	Left to right

Figure 6.809

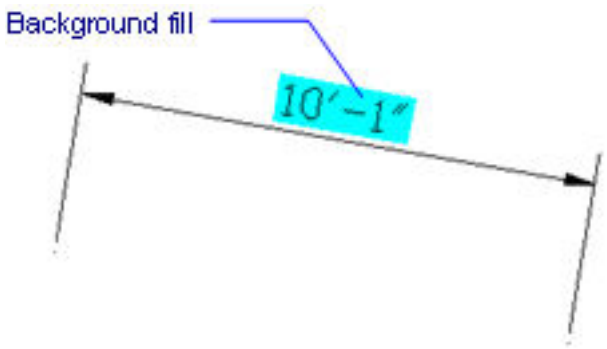


Figure 6.810

Fit	
Arrow and text fit	Best fit
Text inside	No
Dim line inside	No
Text movement	Keep dim line with text
Dim scale overall	0.1
Place text manually	No
Dim line forced	No

Figure 6.811

Primary units	
Dim units	Decimal
Dim precision	0
Fractional type	Horizontal
Decimal separator	.
Dim round	0
Dim prefix	
Dim suffix	
Dim sub-units suffix	
Dim scale linear	1
Dim sub-units scale	100
Suppress leading zeros	No
Suppress trailing zeros	No
Suppress zero feet	Yes
Suppress zero inches	Yes
Dim angle units	Decimal degrees
Dim angle precision	0.0000
Suppress angle leading zeros	No
Suppress angle trailing zeros	No

Figure 6.812

Alternate units	
Alt enabled	No
Alt units	Architectural Stacked
Alt precision	0'-0 1/4"
Alt scale factor	0.03937
Alt sub-units scale	100
Alt round	0
Alt prefix	
Alt suffix	
Alt sub-units suffix	
Alt suppress leading zeros	No
Alt suppress trailing zeros	No
Alt suppress zero feet	Yes
Alt suppress zero inches	Yes

Figure 6.813

Tolerances	
Tolerance display	No
Limits display	No
Tolerance precision	0.0000
Tolerance limit lower	0
Tolerance limit upper	0
Tolerance text height	1 mm
Tolerance position vertical	Middle
Tolerance suppress leading zeros	No
Tolerance suppress trailing zeros	No
Tolerance suppress zero feet	Yes
Tolerance suppress zero inches	Yes
Alt tolerance precision	0'-0 1/4"
Alt tolerance suppress leading zeros	No
Alt tolerance suppress trailing zeros	0
Alt tolerance suppress zero feet	Yes
Alt tolerance suppress zero inches	Yes

Figure 6.814



6.11.21 -DimStyle

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and modifies dimensions styles at the command line.

Accessing the Command

 command bar: **-dimstyle** transparent: **'-dimstyle** alias: **-dst** menu bar: **Dimension | Apply Dimension Style Dimension | Save Dimension Style Dimension | Restore Dimension Style Dimension | Dimension Variable Status** ribbon:**Draw | Dimensions | Apply Dimension Style** : **-DimStyle Prompts you in the command bar:** Current dimension style: Standard Annotative: No Dimension styles or [?/ANnotative/Apply/Save/STatus/Variables/Restore] <Restore>: **(Enter an option)**

Command Options

Option	Description
?	Lists the names of all dimension styles loaded into the current drawing, other than Standard. Prompts you: **Dimension style(s) to list <*>** - **(enter a name or press Enter for all.)**
Annotative	Sets the annotative property of the dimension style.
Apply	Changes or restores the style of existing dimensions by applying the current style. Prompts you: **Select dimensions to apply the current style** - **(choose one or more dimensions.)**
Save	Saves the current dimension settings as a style; useful for saving overrides as a style. Prompts you: **Save style: ? to list style names/Name for new dimension style** - **(specifies the name of the style.)**
STatus	Displays the current value of every dimension variable.
Variables	Lists the values of dimension variables belonging to the selected dimension. Prompts you: **? to list styles/ENTER to select dimension/<Dimension style>** - **(enter an option)**: • **? to list styles** - lists the names of styles currently loaded in the drawing. • **ENTER to select dimension** - prompts you to chose a dimension to use as the current style. • **Dimension style** - lists the values of the style whose name you supply, as well as override values.

Restore	Resets the values of dimension variables to those of the selected dimension. Prompts you **? to list styles/ENTER to select dimension/<Dimension style>** - **(enter an option)**: • **? to list styles** - lists the names of styles currently loaded in the drawing. • **ENTER to select dimension** - prompts you to choose a dimension whose values you want to use as the current style. • **Dimension style** - lists the values of the style whose name you supply.

Procedures

- To apply the current dimension style (1.21.7)
- To save a dimension style (1.21.7)
- To restore a dimension style (1.21.7)
- To list the settings in the current dimension style (1.21.7)

Related Commands

AiDimStyle - creates Dimension Styles based on the properties of selected Dimension, Tolerance or Leader entities.
DimStyle (6.11.20) - creates and modifies dimension styles, through the Drawing Explorer.

Figure 6.815



6.11.22 DimStyleSet

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Reports the current dimension style in the command bar.

Accessing the Command

command bar: **dimstyleset : DimStyleSet** Dimension style set to <name of current dimension style>. The name of the current dimension style also displays in the Current Dimension Style (1.12.2) field in the Status bar.

Command Options

No options

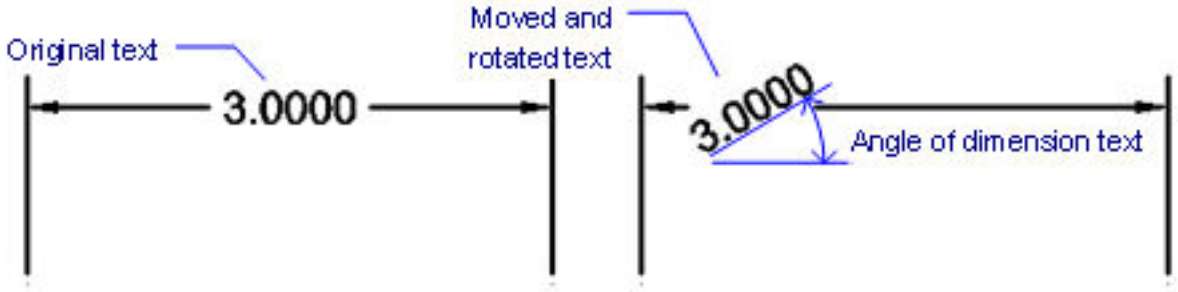
Related Commands

DimStyle (6.11.20) - creates and modifies dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - creates and modifies dimensions styles at the command line.

Figure 6.816



Figure 6.817



6.11.23 DimTEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Changes the position of dimension text. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscon

Accessing the Command

command bar: **dimtedit** alias: **dimted** menubar: **Dimension | Reposition Dimension Text** toolbar: **Dimensions** | ribbon:**Draw | Dimensions | Edit | Reposition Dimension Text : DimTEdit Prompts you in the command bar:** Select dimension to reposition text: **(Choose a dimension)** Specify or enter text location [Angle/Left/Center/Right/REstore] **(Drag the text to its new location, or else enter an option)**

Command Options

Option	Description
Restore ( )	Resets the rotation angle to 0 degrees; does not change repositioned text.
Angle ( )	Rotates the dimension text. Prompts you: **Angle of dimension text - (specify the angle of the text; measured counter-clockwise from the positive x axis.)**
Left	Moves the text next to the left extension line.
Center	Centers the text between the two extension lines.
Right	Moves the text next to the left extension line.

Procedures

- Editing the dimension text (1.21.15)
- To rotate the dimension text (1.21.15)
- To reposition the dimension text (1.21.15)
- To restore the text position (1.21.15)

Related Commands

DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimArc (6.11.5) - Places arc length dimensions. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17)- places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimLeader (6.11.13) and Leader (6.20.20) - places leaders to blocks, entities, or tolerances. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) -

6.12 Dm (Direct Modeling)

6.12.1 DmAngle3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Creates an angle constraint between faces and/or edges of 3D solids, WCS subentity, subentity of the coordinate system of the block. Creates a Planar Angle constraint by default: coordinate planes of the WCS are used as third reference object whenever possible. Allows to control the top angle of a cone.

Figure 6.818



Accessing the Command

 **Angle**  **Cone angle** command bar: **dmangle3d** menu bar: **Parametric | 3D Constraints | Angle** toolbar: **3D Constraints | Angle** ribbon:

1. **Solid |3D Constraints | Angle** and **Cone angle**
2. **Surface |3D Constraints |Angle** and **Cone angle**
3. **Assembly |3D Constraints |Angle**

Quad cursor menu: **Constraints |Angle** and **Cone angle** : **dmAngle3d Prompts you in the command bar:** Select first entity or specify [cone Angle constraint/World/Block]: **(Select a face or an edge.)** Entities/subentities in set: 1 Select second entity [WOrld/BLock]: **(Select a face or an edge or select an option.)** Specify angle value (reference: Global XY plane) or [Set reference entity] <90.00>:**(Either press Enter to accept the current value, type a new value, or select reference entity.)** Press the TAB key to select obscured geometry.

Command Options

Option	Description
cone Angle constraint	Allows to control the top angle of a cone (6.9.63) by specifying angle between the axis and the cone face (= cone half angle); prompts you: **Select a cone face:** - **(Select the face of a cone.) Specify angle value <xx.xx>:** - **(Either press Enter to accept the current value or type a new value. Values must be greater than 0° and smaller than 90°.)**
World	Allows you to select the WCS subentity as an argument; prompts you: **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]:** - (**Select a reference coordinate system subentity as an argument.)**
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; prompts you: **Select block reference** - **(Select a block reference of which the reference coordinate system will be used as an argument.) Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**
Set reference entity	Allows to manually specify the reference entity; prompts you: **Select reference entity** [Keep as is/WOrld/BLock/None]: Keep as is: allows you to keep the proposed reference object; WOrld: allows you to select the WCS subentity as an argument; BLock: allows you to choose as an argument the subentity of the coordinate system of the block; None: creates Angle constraint instead Planar Angle constraint, without third argument.

Related Commands

MechanicalBrowserOpen (6.8.29)- displays the Mechanical Browser. DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmUpdate (6.12.33) - Forces 3d constraints to update. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids. DmRadius3d (6.12.21) - applies a radius constraint to cylindrical surfaces or circular edges.

Figure 6.819



6.12.2 DmAudit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces). DmAudit is a powerful tool to analyze and automatically fix problems in 3D geometry, which are caused by incorrect operations or if geometry was created in another CAD package and imported using the Import (6.17.10) command. It is recommended to run DmAudit whenever 3D geometry is imported if theImport (6.17.10) command is not set up to do this automatically. After fixing 3D geometry using **DmAudit,** operations such as Boolean operations, direct modeling and generating drawing views are possible. The DMAUDITLEVEL (1.13.11) system variable controls whether certain types of errors are reported by DmAudit or not. If set to 1 then modeling kernel dynamic range errors are ignored. If DMREPORTPANEL (1.13.11)= ON, BricsCAD reports the results in the Mechanical Browser / Report (6.8.32) panel instead of the command bar.

Accessing the Command

 command bar: **dmaudit** menu bar:**Model |Direct Modeling | Audit** toolbar: **Direct Modeling | Audit** ribbon:**Sheet Metal | Heal | Audit** Quad cursormenu: **Model |Audit** : **dmaudit Prompts you in the command bar:** Select entities to audit [Entire model] <Entire model>: **(Select an entity to include in the operation or press Enter to analyze the entire model.)** Entities in set: 1 Select entities to audit [Entire model] <Entire model>: **(Select more entities or press Enter.)** Entities in set: xx Choose action [Check/Fix/Settings] <Fix>::**(Choose an option or press Enter.)**

Command Options

Option	Description
Check	Checks the 3D geometry in the drawing for errors without attempting to fix them. Can be used to check whether a model has problems which prevent processing.
Fix	Checks the 3D geometry in the drawing for errors and attempts to fix them.

Settings	Prompts you: **Select setting [Back/report Panel/copy to File/audit Level/Coincident faces/Nested blocks] <Back>:** • Back: Returns to the main prompt of the command. • Report Panel: Redirects the output of the command to the Report (6.8.32) panel. • Copy to File: Reports the output of the command in a file. • audit Level: Allows to set the strictness of the check. Prompts you: **Set audit level [Low/Medium/High] <Medium>:** – The ”Low” setting runs basic analysis (e.g., catching blatantly fatal topology errors). – The ”High” setting goes deep (e.g., catching self-intersections in solid bodies). – The default audit level is ”Medium”. • Coincident faces: Checks and fixes coincident faces (requires audit level set to High). • Nested blocks: Checks and fixes entities in nested blocks.

Related Commands

DmAuditAll (6.12.3) -checks and heals flaws in drawings inserted as external references. DmSimplify (6.12.27) - simplifies the geometry and topology of 3d solid entities. DmSimplifyAll (6.12.29) - simplifies the geometry and topology of solids, within drawings inserted as external references. DmStitch (6.12.28) - converts a set of region and surface entities that bound a watertight area to a 3d solid. Import (6.17.10) - imports the geometry from external files into the current drawing.

Figure 6.820



6.12.3 DmAuditAll

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces) in drawings inserted as external references. DmAuditAll is a powerful tool to analyze and automatically fix problems in 3D geometry in the file and in drawings inserted as external references. It is recommended to run DmAuditAll whenever 3D geometry is imported if theImport (6.17.10) command is not set up to do this automatically. After fixing 3D geometry using **DmAuditAll,** operations such as Boolean operations, direct modeling and generating drawing views are possible. The DMAUDITLEVEL (1.13.11) system variable controls whether certain types of errors are reported by DmAudit or not. If set to 1 then modeling kernel dynamic range errors are ignored. If DMREPORTPANEL (1.13.11)= ON, BricsCAD reports the results in the Mechanical Browser / Report (6.8.32) panel instead of the command bar.

Accessing the Command

command bar: **dmauditall** : **dmauditall Prompts you in the command bar:** Select entities to audit [Entire model] <Entire model>: **(Select an entity to include in the operation or press Enter to analyze the entire model.)** Entities in set: 1 Select entities to audit [Entire model] <Entire model>: **(Select more entities or press Enter.)** Entities in set: xx Choose action [Check/Fix/Settings] <Fix>:: **(Choose an option or press Enter.)**

Command Options

Option	Description
Check	Checks the 3D geometry in the drawing for errors without attempting to fix them. Can be used to check whether a model has problems which prevent processing.
Fix	Checks the 3D geometry in the drawing for errors and attempts to fix them.
Settings	Prompts you: **Select setting [Back/report Panel/copy to File/audit Level/Coincident faces] <Back>:** • Back: Returns to the main prompt of the command. • Report Panel: Redirects the output of the command to the Report (6.8.32) panel. • Copy to File: Reports the output of the command in a file. • audit Level: Allows to set the strictness of the check. Prompts you: **Set audit level [Low/Medium/High] <Medium>:** – The ”Low” setting runs basic analysis (e.g., catching blatantly fatal topology errors). – The ”High” setting goes deep (e.g., catching self-intersections in solid bodies). – The default audit level is ”Medium”. • Coincident faces: Checks and fixes coincident faces (requires audit level set to High).

Related Commands

DmAudit (6.12.2)- fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces). DmSimplify (6.12.27) - simplifies the geometry and topology of 3d solid entities. DmSimplifyAll (6.12.29) - simplifies the geometry and topology of solids, within drawings inserted as external references. DmStitch (6.12.28) - converts a set of region and surface entities that bound a watertight area to a 3d solid. Import (6.17.10) - imports the geometry

Figure 6.821



6.12.4 DmChamfer

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Creates an equal distance chamfer between adjacent faces sharing a sharp edge.

Accessing the Command


command bar: **dmchamfer** menu bar: **Model | Direct Modeling | Chamfer** toolbar: **Direct Modeling | Chamfer** ribbon: **Solid | Edit | Chamfer** Quad cursor menu: **Model | Chamfer** : **dmchamfer Prompts you in the command bar:** Select edges to create chamfer: **(Select the edges you want to chamfer. Press and hold the TAB key to select hidden geometry. Press Enter to stop selecting.)** The chamfer is applied dynamically. To define the chamfer distance, do one of the following:

- Click the left mouse button
- Type a value in the dynamic input field.

Related Commands

DmDelete (**??**) - deletes faces and solids. DmExtrude (6.12.13) - creates a 3D solids by extruding closed 2D entities or regions. DmFillet (6.12.14) - creates a smooth fillet between adjacent faces sharing a sharp edge. DmMove (6.12.17) - moves the selected faces or edges of a solid using a vector. DmPushPull (6.12.20) - adds or removes volume from a solid by moving a face or surface. DmRevolve (6.12.22) - Creates 3D solids by revolution of closed 2D entities or regions about an axis. DmRotate (6.12.24) - rotates faces of a solid around an axis. Chamfer (6.9.9) - creates chamfers; adds an angle to intersections. Fillet (6.14.7) - fillets intersections; joins intersecting lines with an arc of any suitable radius.

Figure 6.822



6.12.5 DmCoincident3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Applies a coincident constraint between two edges, two faces or an edge and a face of two different solids; as well as between a point and a spline, helix, ellipse, polyline, cylinder, and sphere.

Accessing the Command

 and command bar: **dmcoincident3d** menu bar: **Parametric | 3D Constraints | Coincident** toolbar: **3D Constraints | Coincident** ribbon:

- **Solid |3D Constraints | Coincident**
- **Surface |3D Constraints | Coincident**
- **Assembley |3D Constraints | Coincident**

Quad: **Constraints | Coincident** : **dmCoincident3d Prompts you in the command bar:** Select a pair of subentities or [Autoconstrain/WOrld/BLock] <Autoconstrain>: **(Select the first edge or face or press Enter.)** Select second subentity [WOrld/BLock]: **(Select the second edge or face.)** Press the TAB key to select obscured geometry.

Command Options

Option	Description
Autoconstrain	Automatically creates a coincident constraint between each pair of connected faces in selection of solids. Prompts you: **Select a set of entities:(Select at least two solids.)** Entities in set: 2 **Select a set of entities: (Select more solids or press Enter.)** You can check which constraints are created in the Mechanical Browser (3.2.2) panel.
World	Allows you to select the WCS subentity as an argument; prompts you: **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; prompts you: **Select block reference** - **(Select a block reference of which the reference coordinate system will be used as an argument.) Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**

Related Commands

MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser. DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmUpdate (6.12.33) - Forces 3d constraints to update. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids. DmRadius3d (6.12.21) applies a radius constraint to cylindrical surfaces or circular edges.

Figure 6.823



6.12.6 DmConcentric3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Applies a concentric constraint between two cylindrical, spherical or conical surfaces.

Accessing the Command

 command bar: **dmconcentric3d** menu bar: **Parametric | 3D Constraints | Concentric** toolbar: **3D Constraints | Concentric** ribbon:

- **Solid |3D Constraints | Concentric**
- **Surface |3D Constraints | Concentric**
- **Assembley |3D Constraints | Concentric**

Quad cursor menu: **Constraints | Concentric** : **dmConcentric3d Prompts you in the command bar:** Select first subentity [WOrld/BLock]: **(Select the first cylindrical (circular or elliptical), spherical or conical surface or choose an option.)** Select second subentity [WOrld/BLock]: **(Select the second cylindrical (circular or elliptical), spherical or conical surface or choose an option.)** Press the TAB key to select obscured geometry.

Command Options

Option	Description
World	Allows you to select the WCS subentity as an argument; prompts you: **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; prompts you: **Select block reference** - **(Select a block reference of which the reference coordinate system will be used as an argument.) Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**

Related Commands

MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser. DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmUpdate (6.12.33) - Forces 3d constraints to update. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids. DmRadius3d (6.12.21) - applies a radius constraint to cylindrical surfaces or circular edges.

Figure 6.824



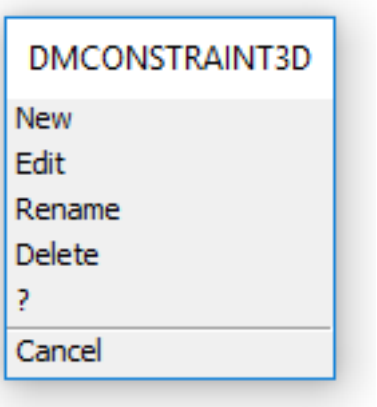
6.12.7 DmConstraint3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities.

Accessing the Command

command bar: **DmConstraint3d** : **dmconstraint3d Prompts you in the command bar:** Enter option [New/Edit/Rename/Delete/? <?>: **(select an option or press Enter to list the constraints in the Prompt History window.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/D/DmConstraint3d.png})

Figure 6.825



Command Options

Option	Description
New	Allows to create a new parameter; prompts you: **Specify 3D constraint type [Fix/ Coincident/ CONcentric/ Parallel/ PErpendicular/ Tangent/ RIgidset/ Distance/ Radius/ Angle]: -(Type a 3D constraint type.)** • Fix: Applies a fixed constraint to a solid or to an edge or a face of a solid. The option is equivalent to the DmFix3d (6.12.15)command ( ). • Coincident: Applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. The option is equivalent to the DmCoincident3d (6.12.5)command ( ). • CONcentric: Applies a concentric constraint between two cylindrical, spherical or conical surfaces. The option is equivalent to the DmConcentric3d (6.12.6)command ( ). • Parallel: Applies a parallel constraint between two faces of a solid or of different solids. The option is equivalent to the DmParallel3d (6.12.18)command ( ). • PErpendicular: Applies a perpendicular constraint between two faces of a solid or of different solids. The option is equivalent to the DmPerpendicular3d (6.12.19)command ( ). • Tangent: Applies a tangent constraint between a face and a curved surface of different solids. The option is equivalent to the DmTangent3d (6.12.30)command ( ). • RIgidset: Allows to define a set of entities or subentities as a rigid body. The option is equivalent to the DmRigidSet3d (6.12.23)command ( ). • Distance: Applies a distance constraint between two subentities of a solid or of different solids. The option is equivalent to the DmDistance3d (6.12.12)command ( ). • Radius: Applies a radius constraint to cylindrical surfaces or circular edges. The option is equivalent to the DmRadius3d (6.12.21)command ( ). • Angle: Applies an angle constraint between to faces of a solid or of different solids. The option is equivalent to the DmAngle3d (6.12.1)command ( ).

Edit	Allows to edit the selected constraint; prompts you: **Enter constraint name:**- **(Type the name of a constraint.) Enter option [Enable/Value/eXpose/Properties]:** - **(choose an option)** • Enable: toggles the selected constraint ON/OFF. • Value: type a value in the command bar. • eXpose: choose whether the parameter is visible and can be modified when the component is inserted in an assembly; Prompts you: **Set expose mode [ON/OFF/Auto] <Auto>:** - **(choose an option): Auto**= the parameter is exposed only if it does not depend on other parameters. **ON**= the parameter is always exposed. **OFF**= the parameter is neverexposed. • Properties: allows to edit properties of the selected constraint; prompts you: **Select property [Use as/Placements]:** or **Select property [Placements/Directions]: - (choose an option)**: **Use as** - **(chose which geometry is used in a 3D constraint to measure; prompts you)**: **Specify how to use the first argument [Boundary/Center] <Center>:** – Boundarymeans that the constraint is applied to the boundary of the argument; – Axisis applied to an argument that has axis, namely : circle, cylinder, cone, or torus; – Centeris applied to an argument that has central point, namely : circle, sphere, or torus; – Vertexcan be applied to the cone vertex only. **Placements**- **(specify how to place an argument with respect to another argument of the same constraint)**; prompts you: **Specify placement of the first argument [Any/Outside/Inside/Keep] <Keep>:** - **(choose an option orpress Enter to keep the current placement of the argument.)** – Outside -argument placement outside ofanotherargument; – Inside - argument placement inside ofanotherargument; – Any -bothOutsideandInsideplacement is suitable for user's intentions; – Keep- keep the current placement of the argument.By defaultKeepoption is used. **Directions**- **(specify whether the directions of the constraint arguments are Same or Opposite)**; prompts you: **Specify directions of the constraint arguments [Any/ Same/ Opposite/ Keep] <Any>:**- **(choose an option orpress Enter to keep the Any option.)** – Same -same arguments direction; – Opposite -opposite arguments direction; – Any -both Same and Opposite directions are suitable for user's intentions; – Keep -keep the current directions of the arguments.

Rename	Allows to rename a constraint; prompts you: **Enter constraint name:** - **(Type the name of a constraint.)** **Enter new constraint name:** - **(Type the new name of a constraint.)**
Delete	Allows to delete a constraint; prompts you: **Enter constraint name:** - **(Type the name of a constraint.)**

The user can use Mechanical Browser to the editing of the 3D Constraints Properties. (1.19.5)

Related Commands

MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser. DmUpdate (6.12.33) - Forces 3d constraints to update. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids. DmRadius3d (6.12.21) - applies a radius constraint to cylindrical surfaces or circular edges.

Figure 6.826



6.12.8 DmCopyFaces

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Copies a set of 3D solid faces forming a feature (e.g. hole, rib or protrusion) to a new position on the same or another 3D solid.

Accessing the Command

 command bar: **dmcopyfaces** menu bar: **Model | Direct Modeling | Copy Faces** toolbar: **Direct Modeling | Copy Faces** ribbon: **Solid | Edit | Copy Faces** Quad cursor menu: **Model| Copy Faces** : **dmcopyfaces Prompts you in the command bar:** Select several entities/subentites: **(Select one or more entities.)** Entities in set: 1 Select several entities/subentites: **(Select more entities or, press Enter stop selecting entities.)** Enter base point: **(Pick a point.)** Select insertion point or [Rotate/Multiple/Mode]: **(Pick another point, or enter an option.)**

Command Options

Option	Description
Rotate	Allows to rotate the selected entities; prompts you: **Specify rotation angle - (type a value) Select insertion point or [Rotate/mUltiple/MOde] - (pick a point or choose an option.)**
Multiple	Allows to create multiple copies: prompts you: **Select insertion point or [Rotate/mUltiple/MOde] - (pick a point or choose an option for the first copy.) Select insertion point or [Rotate/mUltiple/MOde] - (pick a point or choose an option for the next copy.) Select insertion point or [Rotate/mUltiple/MOde] - (pick a point or choose an option for the next copy or press Enter to stop.)**
Mode	Toggles between various cutting options; prompts you: **Cutting mode [Placement face/Select faces/No cut] <Placement face> - (choose an option.)** • Placement Face : (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • Select Faces : (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • No Cut : (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so

Procedures

Copying features (1.18.2)

Related Commands

BmInsert (6.8.12) - inserts an existing mechanical component or sheet metal form feature into the current drawing.

Figure 6.827

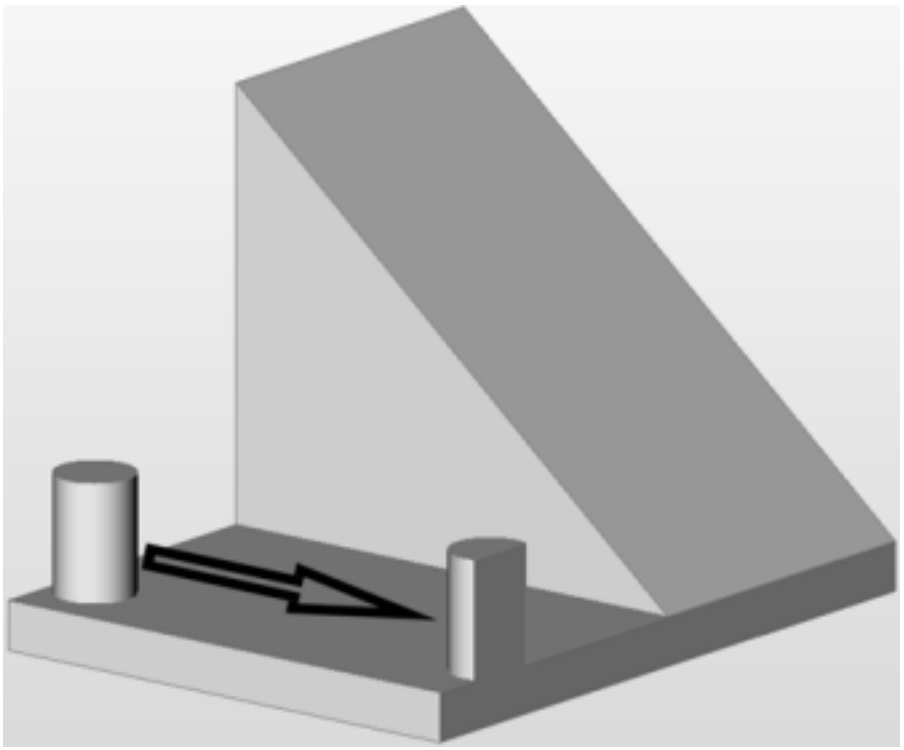


Figure 6.828

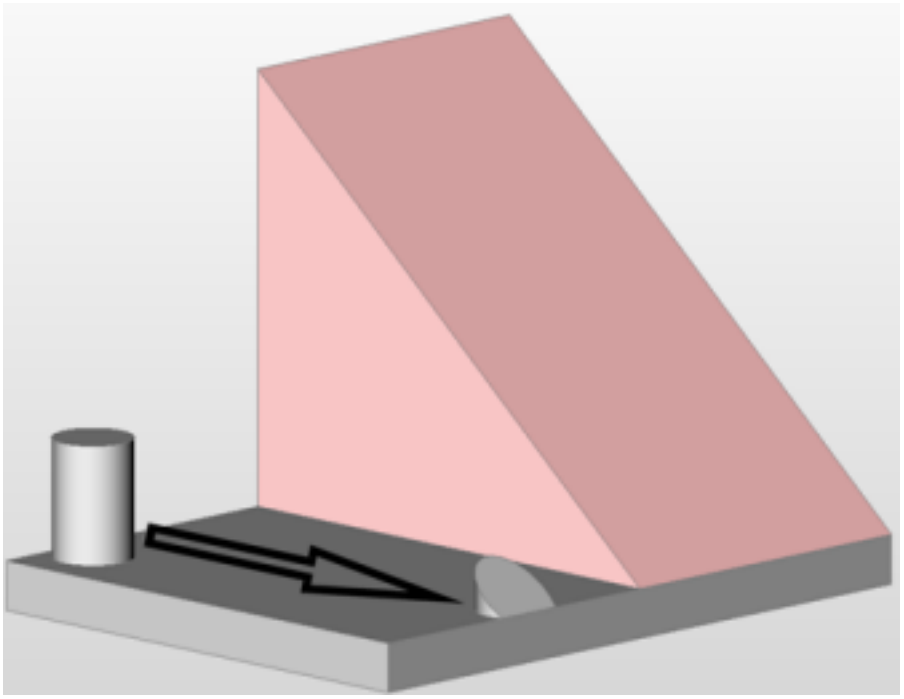


Figure 6.829

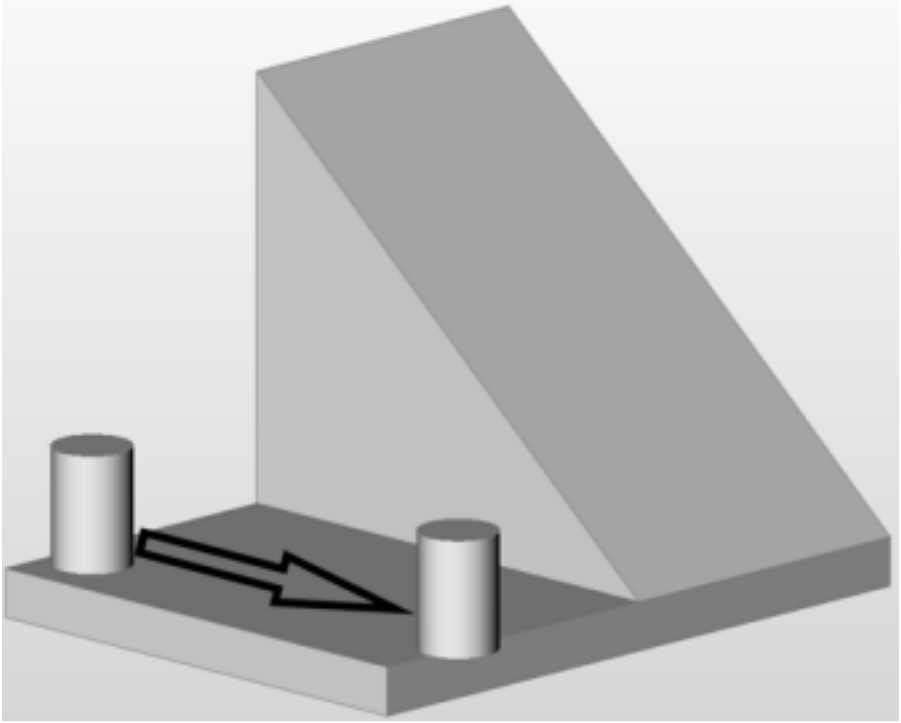


Figure 6.830



6.12.9 DmDeformCurve

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves.

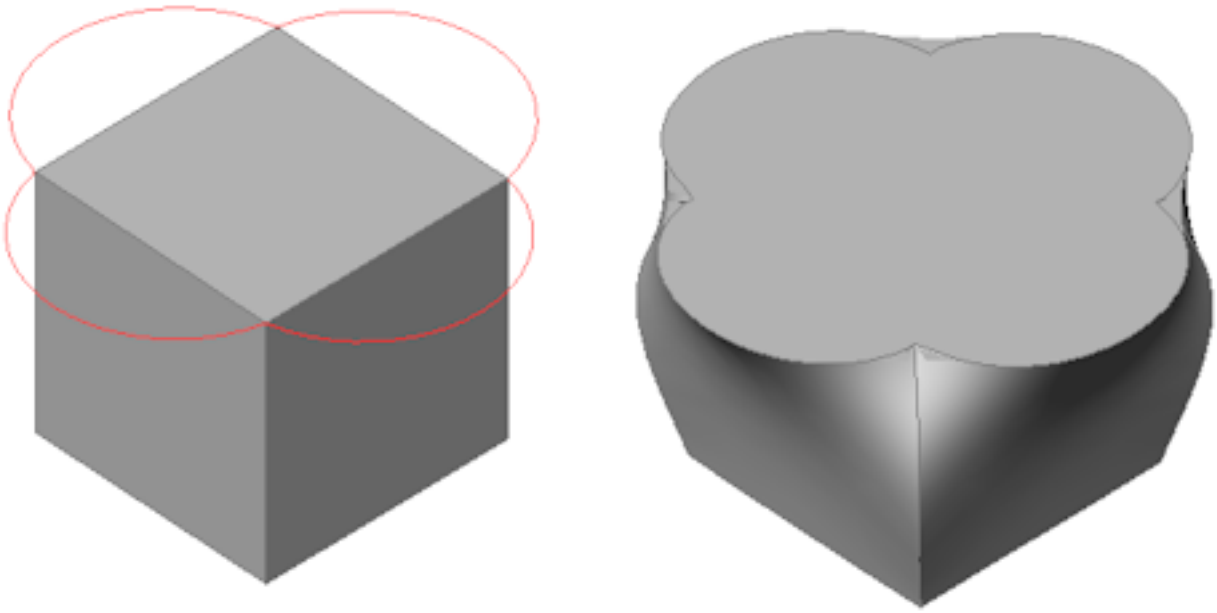
Accessing the Command

 command bar: **dmdeformcurve** menu bar: **Model | Direct Modeling | Deform | Transform Curve** toolbar: **Direct Modeling | Transform Curve** ribbon:

- **Solid | Deform | Transform Curve**
- **Surface |Deform | Transform Curve**

Quad cursor menu:**Model | Transform Curve** : **dmdeformcurve Prompts you in the command bar:** Select edge to deform or [add Faces] <done>: **(Select an edge.)** Select target curve: **(Select a curved to replace the selected edge with.)** Select edge to deform or [add Faces] <done>: **(Select an edge or press Enter to conclude the command.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DeformCurve.png})

Figure 6.831



Command Options

Option	Description
Add Faces	Allows to select additional faces to deform (in addition to the faces adjacent to selected edges). Prompts you: **Select additional deformable faces: (Select a face.)** Entities in set: 1 **Select additional deformable faces: (Select a face or press Enter to stop selecting additional faces.)**

Related Commands

DmDeformMove (6.12.10) - deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges. DmDeformPoint (6.12.11) - deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction. DmTwist (6.12.32) - modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

Figure 6.832



6.12.10 DmDeformMove

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges.

Accessing the Command

 command bar: **dmdeformmove** menu bar: **Model | Direct Modeling | Deform | Move Edge** toolbar: **Direct Modeling | Move Edge** ribbon:

- **Solid | Deform | Move Edge**
- **Surface |Deform |Move Edge**

Quad cursor menu:**Model |Move Edge** : **dmdeformmove Prompts you in the command bar:** Select edges of a 3D solid, surface or region: **(Select an edge.)** Entities in set: 1 Select edges of a 3D solid, surface or region: **(Select more edges or press Enter.)** The selected edges are moved dynamically forcing the deformation of the adjacent faces. Specify deformation value or set [Base point/Direction/additional Faces]: **(Specify the deformation or choose an option.)**

Command Options

Option	Description
Specify Deformation Value	To specify the deformation do one of the following: • Specify a point in the drawing. • Type a value in the dynamic entry fields. Hit the TAB key to switch between the distance (1) and angle (2) fields. The edges are rotated around an axis passing through the base point (3) in the deformation direction. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Direc (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Direc Dynamics (left) and final result (right)
Base Pöint	Allows to specify the base point. The deformation axis passes through the base point. Prompts you: **Specify new deformation point:** - **(Specify a point in the drawing.)**
Direction	Allows to specify the direction of the deformation axis. Prompts you: **Select axial entity or define direction by [2Points/Xaxis/Yaxis/Zaxis] <2Points>: (Choose an option or press Enter to define the direction by specifying two points in the drawing.)**
Additional Faces	Allows to select additional faces to be deformed, in addition to ones adjacent to the selected edge(s). Prompts you: **Select additional deformable faces:-(Select a face.)** Entities in set: 1 **Select additional deformable faces:-(Select more faces or press Enter.)**

Related Commands

DmDeformCurve (6.12.9) - deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves. DmDeformPoint (6.12.11) - deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction. DmTwist (6.12.32) - modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

Figure 6.833

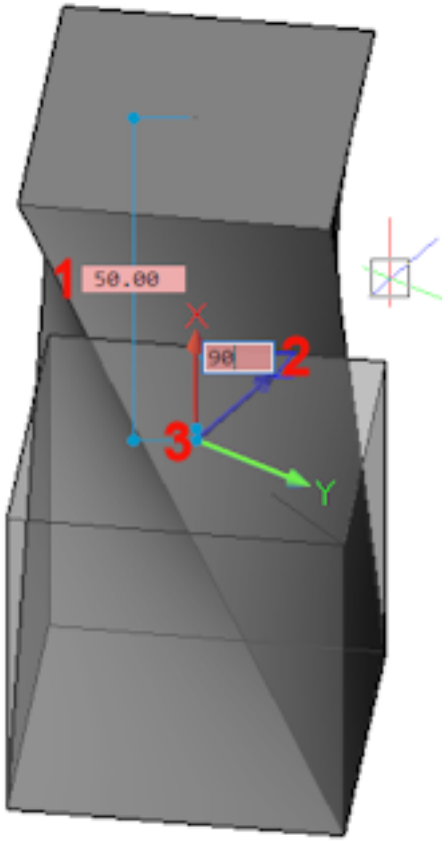


Figure 6.834



Figure 6.835



6.12.11 DmDeformPoint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction. The selected faces are deformed as smooth as possible. If there is G1 or G2 continuity between faces, it is preserved after deformation.

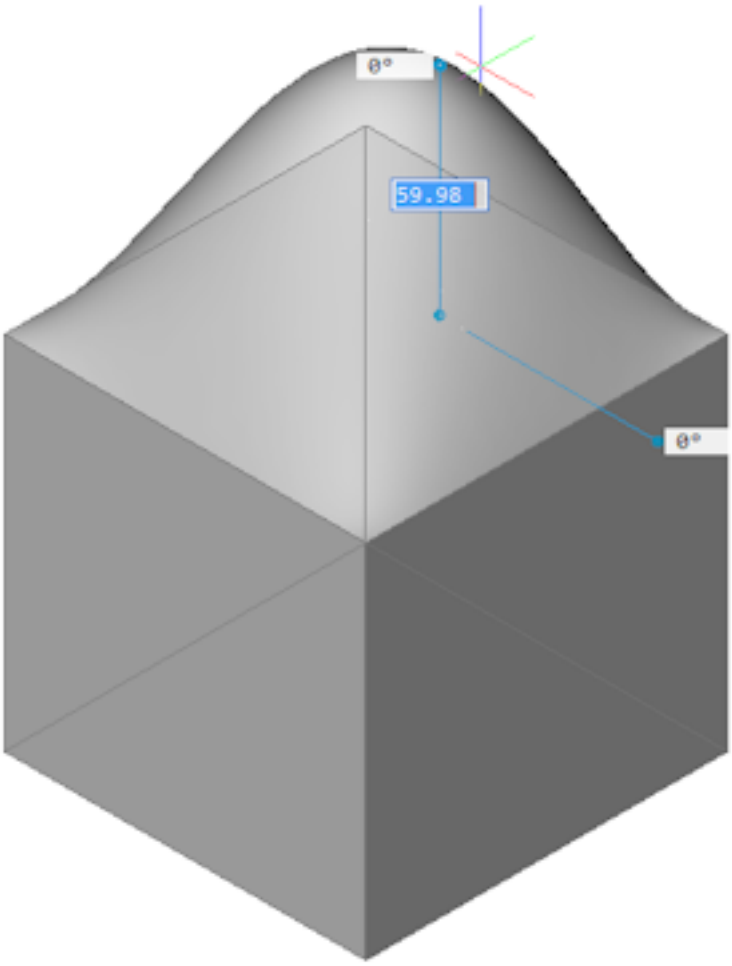
Accessing the Command

 command bar: **dmdeformpoint** menu bar: **Model | Direct Modeling | Deform | Move Edge** toolbar: **Direct Modeling | Move Edge** ribbon:

- **Solid | Deform | Move Edge**
- **Surface |Deform |Move Edge**

Quad cursor menu:**Model |Move Edge** : **dmdeformpoint** Prompts you in the command bar: Select faces of a 3D solid or surface: **(Select a face of a 3D solid or a surface or a region.)** Entities in set: 1 Select faces of a 3D solid or surface: **(Select a connected face of the same 3D solid or press Enter.)** Specify deformation point or [change Parameters] <use default point>: **(Specify a point or press Enter to accept the default point.)** The selected entities are deformed dynamically towards the surface normal direction at the deformation point. Specify deformation value or [set Target point/set Direction/change Base point]: **(Specify a point or type a value.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DeformPoint.png})

Figure 6.836



Command Options

Option	Description
Change Parameters	Prompts you: **Change parameter [Alpha/Beta/Gamma/Delta] <done>:** • Alpha: Specifies the resistance to stretching. It is a second order tensor which can be described with three numbers. Each value must be either 0 or positive number: • alpha U: U-direction resistance • alpha V: V-direction resistance, • alpha theta: the angle between U and V principal directions of the surface and the material property directions. • Beta: Specifies the resistance to bending. Similarly to Alpha, Beta is defined as a triple: beta U, beta V, beta theta. Each value must be either 0 or a positive number. • Gamma: Specifies the resistance to the rate of change in bending. The value must be either 0 or a positive number. • Delta: Specifies the resistance to deviations from the default shape. The value must be either 0 or a positive number.
Set Target Point	Prompts you: **Specify deformation target point:**- **(Specify a point.)** The entities are deformed towards the specified point.
Set Direction	Prompts you: **Select axial entity or define direction by [2Points/Xaxis/Yaxis/Zaxis] <2Points>: (Choose an option or press Enter to define the extrusion direction by specifying two points.)** The selected entities are deformed dynamically in the specified direction. Prompts you: **Specify deformation value or [set Target point/set Direction/change Base point]:**- **(Specify a point or type a value or choose an option.)**
Change Base Point	Prompts you: **Specify new deformation point:** -**(Specify a point.)** Prompts you: **Specify deformation value or [set Target point/set Direction/change Base point]:**- (**Specify a point or type a value or choose an option.)**

Related Commands

DmDeformCurve (6.12.9) - deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves. DmDeformMove (6.12.10) - deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges. DmTwist (6.12.32) - modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

Figure 6.837



6.12.12 DmDistance3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Applies a distance constraint between two subentities of a solid or of different solids, as well as between a point and cylinder or sphere.

Accessing the Command

 command bar: **dmdistance3d** menu bar: **Parametric | 3D Constraints | Distance** toolbar: **3D Constraints | Distance** ribbon:

- **Solid |3D Constraints | Distance**
- **Surface |3D Constraints | Distance**
- **Assembley |3D Constraints | Distance**

Quad cursor menu: **Constraints | Distance** : **dmDistance3d** Select a first entity or specify [Use as/WOrld/BLock]: **(Select the first entity or choose an option.)** Press the TAB key to select obscured geometry. Enter distance value <xxx.xx>: **(Type the desired distance.)**

Command Options

Option	Description
Use as	Allows to specify how the distance must be measured between cylindrical and spherical faces.The default measurement mode is **Axis**. Prompts you: Specify how to use the argument **[Boundary/Axis/Center/Vertex] <Axis>:** - **(Choose an option)** • Boundarymeans that the constraint is applied to the boundary of the argument; • Axisis applied to an argument that has axis, namely: circle, cylinder, cone, or torus; • Centeris applied to an argument that has central point, namely: circle, sphere, or torus; • Vertexcan be applied to the cone vertex only.
World	Allows you to select the WCS subentity as an argument; prompts you: **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; prompts you: **Select block reference** - **(Select a block reference of which the reference coordinate system will be used as an argument.) Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**

Related Commands

MechanicalBrowserOpen (6.8.29)- displays the Mechanical Browser. DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmUpdate (6.12.33) - Forces 3d constraints to update. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmRadius3d (6.12.21) - applies a radius constraint to cylindrical surfaces or circular edges.

Figure 6.838



6.12.13 DmExtrude

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Creates 3D solids or surfaces by extruding open or closed 2D entities, faces of 3D solids, regions or closed boundaries. Optionally adds volume to solids or removes volume from solids.

Accessing the Command

 **Solid mode**  **Surface mode** command bar: **dmextrude** menu bar: **Model | Direct Modeling | Extrude solid and Extrudesurface** toolbar: **Direct Modeling |Extrude solid and Extrudesurface** ribbon:

- **Solid | Create | Extrude solid**
- **Surface | Create | Extrudesurface**

Quad cursor menu: **Mode | Extrude solid and Extrudesurface** : **dmExtrude** When launched from the command bar, menu bar, toolbar or ribbon: **Prompts you in the command bar:** Select entities/subentities to extrude or set [MOde]: **(Select a region or a closed 2D entity (closed polylines, circles and ellipses are accepted) or a face of solid.)** Entities in set: 1 Select entities/subentities to extrude or set [MOde]: **(Select another entity or subentity or press Enter to stop selecting entities.)** When launched from the Quad cursor menu: First select the entity, entities or boundary to be extruded. Closed 2D entities (1)(2) and regions (3) are accepted. Closed boundaries in the XY-plane of the current coordinate system (4) or on the face of a solid (5) are recognized automatically if the **Select detected boundaries** option () on the **Selection Modes** toolbar is active.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrude1	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrude2	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrude3	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrude4	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrude5
1	2	3	4	5

Do one of the following:

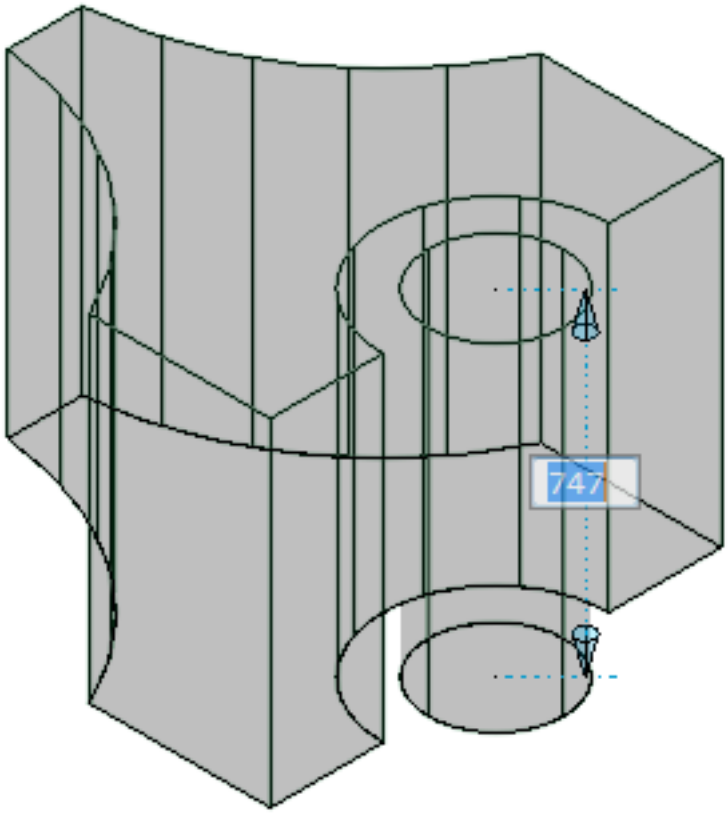
- Launch the command when the desired entity or boundary highlights.
- To extrude multiple entities: compose a selection set, then launch the command. Click to add the highlighted entity to the selection.

Solid mode: Specify height of extrusion or [Auto(subtract or create)/Create/SUbtract/Unite/Both sides/Taper angle/Direction/Limit] <Auto>: The selected entities are extruded dynamically. **(Move the mouse to specify the extrusion direction.)** The extrusion direction is perpendicular to the selected entity or entities. Do one of the following to define the height of the extrusion:

- Specify a point in the drawing.
- Type a value in the dynamic entry field.
- Choose the Limit option.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrude.png})

Figure 6.839



Surface mode: Specify height of extrusion or set [Auto/Create/Slice/Both sides/Taper angle/Direction] <Auto>: The selected entities are extruded dynamically. Move the mouse to specify the extrusion direction. The extrusion direction is perpendicular to the selected entity or entities unless the **Direction** option is selected. Do one of the following to define the height of the extrusion:

- Specify a point in the drawing.
- Type a value in the dynamic entry field.

Command Options

Option Description Mode Allows to create either solids or surfaces. Prompts you: **Choose type of created entity [SOlid/SUrface] <Solid>:**

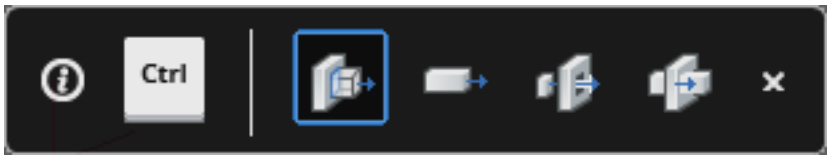
- Solid: creates solids.
- Surface: creates surfaces.

The **Extrude** tool ( ) starts the command in **Solid** mode. The **Extrude Surface** tool ( ) starts the command in **Surface** mode. Select Entities Click an entity or choose an option in the dmExtrude prompt menu. Depending on the value of the DELOBJ (1.13.11) (Delete Object) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities. Taper Angle Specifies the taper angle for the extrusion. Prompts you: **Specify taper angle value in degrees** - **(type a value in the command bar.)** The angle is measured from the extrusion direction. A negative value tapers outwards. This option allows to create (truncated) pyramids and cones. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtr
If **Dynamic Dimensions** are ON, press the TAB key to jump to the **Taper Angle** field, then type the desired value and press the TAB key again to apply the new value. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comm
Direction Allows to specify the extrusion direction. Prompts you: **Select axial entity or define direction by [2Points/Xaxis/Yaxis <2Points>:** Select an option:

- 2 Points: Define the extrusion direction by specifying two points.
- X axis: Sets the extrusion direction parallel to the X axis of the current UCS.
- Y axis: Sets the extrusion direction parallel to the Y axis of the current UCS.
- Z axis: Sets the extrusion direction parallel to the Z axis of the current UCS.

Limit Allows to limit the extrusion by the face of a solid. Prompts you: **Select face or 3d solid to limit extrude: (Move the cursor over a face, then click when the face highlights.)**

Figure 6.840



(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrude1.png}) 1. Select entities. 2. Launch DMEXTRUDE.	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrude2.png}) 3. Choose Limit option. 4. Select cylindrical face.	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrude3.png}) Result

The SELECTIONPREVIEW (1.13.11) system variable must be set to 2 or 3 to highlight faces. Press the TAB key to subsequently highlight overlapping faces. Auto* The result depends on the extrusion direction of the 3D solid and the value of the DMEXTRUDEMODE (1.13.11) system variable.

- when extruded inside the existing solid, the new solid is subtracted from the existing solid (right). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrudeAuto.png})
- when extruded outside the existing solid (left) the result depends on the value of the DMEXTRUDEMODE (1.13.11) system variable, which is stored as the sum of the value of the selected options.
- 0 : the new solid is unified with the existing solid. There is no interaction with intersecting solids. This is the default option for 3D Modelling.
- 1 : a separate solid is created which is unified with intersecting solids.
- 2 : the new solid is unified with the existing solid and subtracted from intersecting solids.
- 3 : a separate solid is created and subtracted from intersecting solids. This is the default option for BIM.
- 4 : the new solid is unified with the existing solid, intersecting solids left intact.
- 5 : a separate solid is created, intersecting solids are left intact.

Create* Regardless of the extrusion direction, a new 3D solid is created. Subtract* The 3D solid is subtracted from each interfering existing solid. Unite* The new 3D solid is unified with each interfering existing solid. Slice* The extruded surface cuts through solids, much like the Slice (6.27.36) command. Both Sides Extrudes the selected boundary (solid mode) or linear entities (surface mode) symmetrically at both sides. * Repeatedly hit the CTRL key during the dynamic display of the extrusion to cycle the various options:

- Auto / Create / Subtract / Unite (solid mode)
- Auto / Create / Slice (surface mode)

The behavior of the **Auto** option is controlled through the DMEXTRUDEMODE (1.13.11) system variable. The value of the variable is the sum of the selected bitcodes.

- **1**: Set **OFF** to unite with, or **ON** to create a new solid when extruding a contour lying on the face of a solid, away from that solid.
- **2**: Set **OFF**to unite with, or **ON** to subtract from solids which intersect with the created volume.
- **4**: Set **ON** to modify solids which intersect with the created volume.

If the Hotkey Assistant (HKA) is ON the Hotkey Assistantwidget displays: Solid Mode: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrudeTips.png})

-  : Auto
-  : Create
-  : Subtract
-  : Unite

Surface Mode: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtru

-  : Auto

Figure 6.841



-  : Create
-  : Slice

Related Commands

Extrude (6.13.21) - Extrudes 2D entities to 3D solids or surfaces. Loft (6.20.37) - creates 3D solids passing through two or more cross sections. Sweep (6.27.60) - creates 3D solids or 3D surfaces by sweeping profiles (open or closed 2D entities or 3D subentities) along an open or closed path. DmChamfer (6.12.4) - creates an equal distance chamfer between adjacent faces sharing a sharp edge. DmDelete (**??**) - deletes faces and solids. DmFillet (6.12.14) - creates a smooth fillet between adjacent faces sharing a sharp edge. DmMove (6.12.17) - moves the selected faces or edges of a solid using a vector. DmPushPull (6.12.20) - adds or removes volume from a solid by moving a face or surface. DmRevolve (6.12.22) - Creates 3D solids or surfaces by revolution of open or closed 2D entities or regions about an axis. DmRotate (6.12.24) - rotates faces of a solid around an axis. DmStitch (6.12.28) - converts a set of region and surface entities that bound a watertight area to a 3d solid. DmThicken (6.12.31) - creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, and faces of 3D solids.

Figure 6.842



Figure 6.843

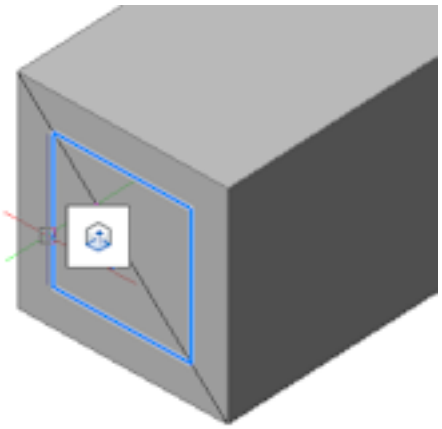


Figure 6.844



Figure 6.845



Figure 6.846

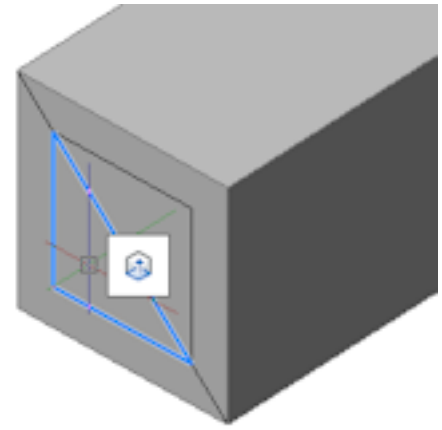


Figure 6.847

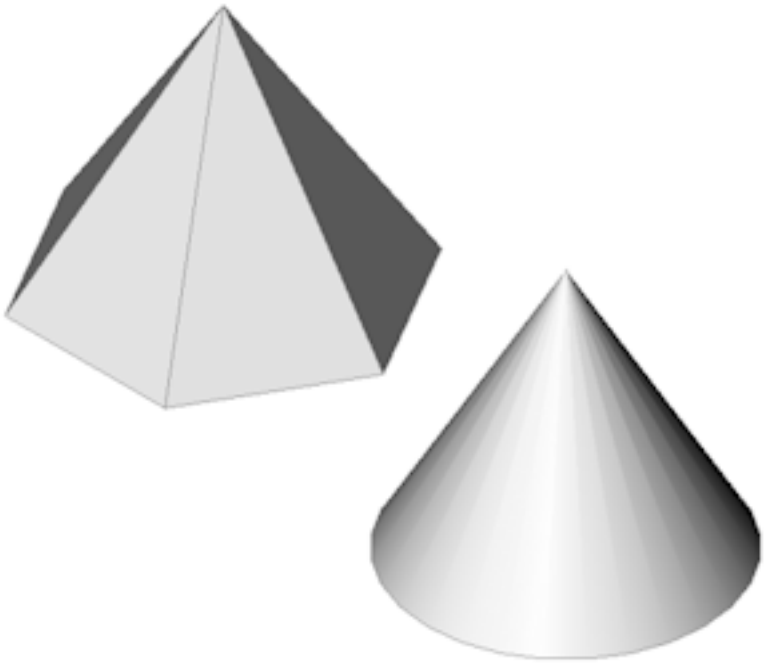


Figure 6.848

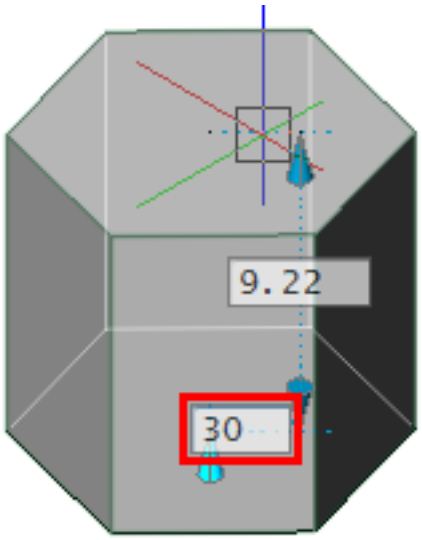


Figure 6.849

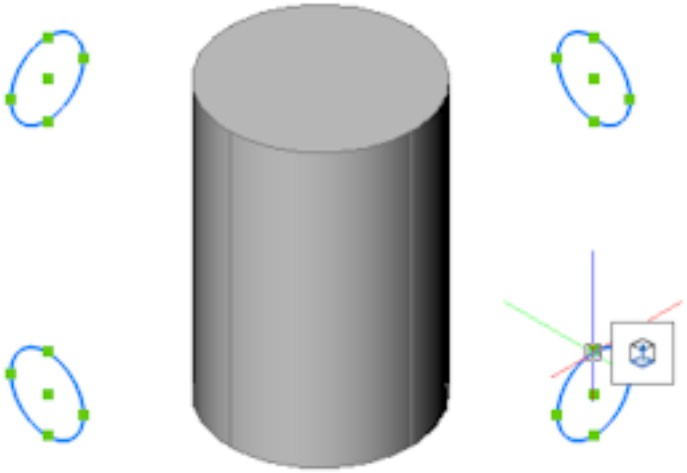


Figure 6.850

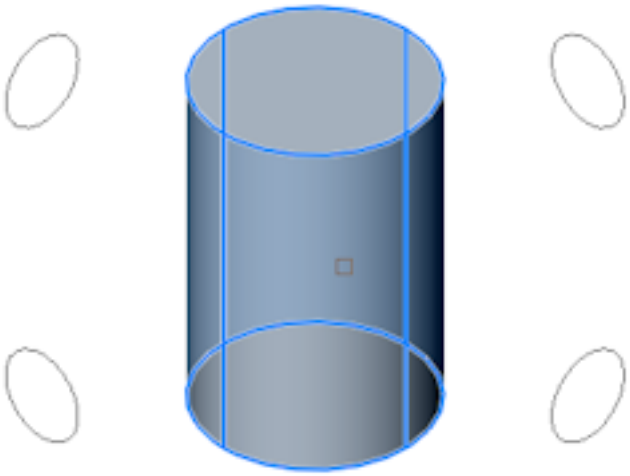


Figure 6.851

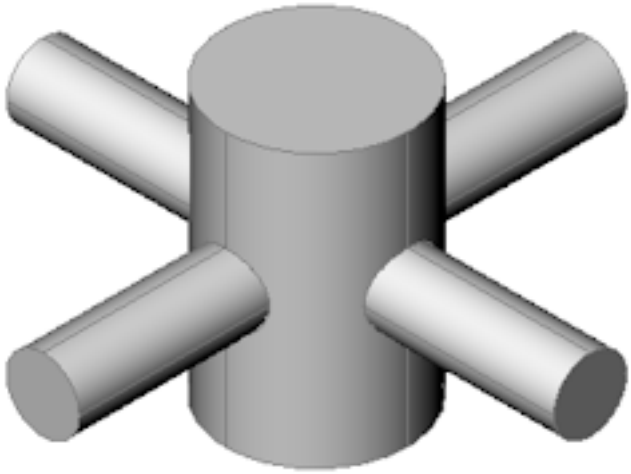


Figure 6.852

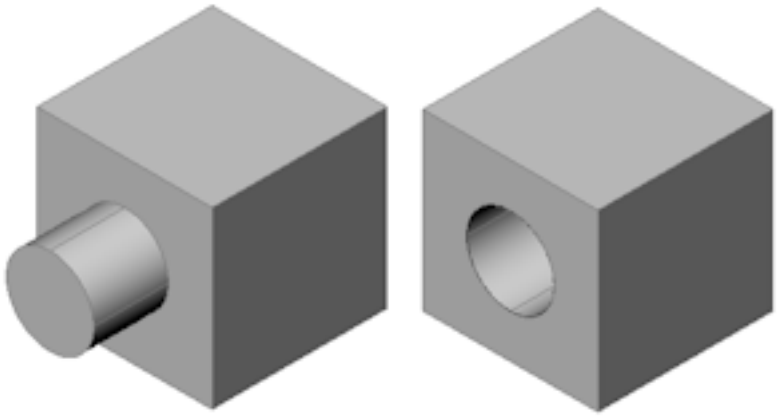


Figure 6.853



6.12.14 DmFillet

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Creates a smooth fillet between adjacent faces sharing a sharp edge.

Accessing the Command


command bar: **dmfillet** menu bar: **Model | Direct Modeling | Fillet** toolbar: **Direct Modeling | Fillet** ribbon: **Solid | Edit | Fillet** Quad cursor menu: **Model | Fillet** : **dmfillet Prompts you in the command bar:** Select edges to create fillet: **(Select the edges you want to fillet. Press and hold the TAB key to select hidden geometry. Press Enter to stop selecting.)** The fillet is applied dynamically. To define the fillet radius, do one of the following:

- Click the left mouse button.
- Type a value in the dynamic input field.

Command Options

No options.

Related Commands

DmChamfer (6.12.4) - creates an equal distance chamfer between adjacent faces sharing a sharp edge. DmDelete - deletes faces and solids. DmExtrude (6.12.13) - creates a 3D solids by extruding closed 2D entities or regions. DmMove (6.12.17) - moves the selected faces or edges of a solid using a vector. DmPushPull (6.12.20) - adds or removes volume from a solid by moving a face or surface. DmRevolve (6.12.22) - Creates 3D solids by revolution of closed 2D entities or regions about an axis. DmRotate (6.12.24) - rotates faces of a solid around an axis. Chamfer (6.9.9) - creates chamfers; adds an angle to intersections. Fillet (6.14.7) - fillets intersections; joins intersecting lines with an arc of any suitable radius.

Figure 6.854



6.12.15 DmFix3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Applies a fixed constraint to a solid or to an edge or a face of a solid.

Accessing the Command

 command bar: **dmfix3d** menu bar: **Parametric | 3D Constraints | Fix** toolbar: **3D Constraints | Fix** ribbon:

- **Solid |3D Constraints | Fix**
- **Surface |3D Constraints | Fix**
- **Assembley |3D Constraints | Fix**

Quad cursor menu: **Constraints | Fix** : **dmFix3d Prompts you in the command bar:** Select an edge, face or 3D solid or [Multiple/BLock]: **(Select a solid, or an edge or a face of a solid.)** Press the TAB key to select obscured geometry.

Command Options

Option	Description
Solid	All faces and edges of the solid are fixed.
Face	The selected face is fixed and cannot be moved or rotated.
Edge	The selected edge is fixed. The adjacent faces can be rotated, but not moved.
Multiple	Allows to select multiple entities or subentities; prompts you: **Select edges, faces or 3D entities** - **(Select a solid, or an edge or a face of a solid)** Entities in set: 1 **Select edges, faces or 3D entities** - **(Select more solids, edges or faces of solids or press Enter.)**
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; prompts you: **Select block reference** - **(Select a block reference of which the reference coordinate system will be used as an argument.) Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**

Related Commands

MechanicalBrowserOpen (6.8.29)- displays the Mechanical Browser. DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmUpdate (6.12.33) - Forces 3d constraints to update. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids. DmRadius3d (6.12.21) - applies a radius constraint to cylindrical surfaces or circular edges.

6.12.16 DmGroup

 Platinum  Platinum  Platinum Creates group features, which are named persistent collections of faces and edges of 3D solids and surfaces. A group feature can be coupled with its description, which can be used to store manufacturing instructions or any other informationwith your model. Group features usually survive after modification of 3D solids/surfaces and can be accessed or created in Lisp scripts and BRX applications.

Accessing the Command

command bar: **dmgroup** : **dmgroup Prompts you in the command bar:** Enter parameter option [New/Edit/Dissolve] <New>: **(Press Enter to create a new group or choose an option.)** Enter name for new group <Group_1>: **(Enter name* of the new group feature or press Enter to accept the Group_xx default name.)** Select entities: **(Select faces or edges of solids.)** Entities in set: 1 Select entities: **(Select more faces or edges of 3D solids or press Enter to stop selecting)** Enter description for new group: **(Type a description. which appears in the Description field of the feature in https://help.bricsys.com/hc/en-us/articles/360006693154-MechanicalBrowserOpenor press Enter to keep the description field empty.)**

Command Options

Option	Description
Edit	Allows to edit an existing group feature. Prompts you: **Enter name of group *** - **(type the name of the group to be edited.) Select entities** - **(Select faces or edges of solids.)** Entities in set: 1 **Select entities** - **(Select more faces or edges of 3D solids or press Enter to stop selecting.)** The new selection replaces the previous collection.
Dissolve	Allows to delete an existing group feature. Prompts you: **Enter name of group *** - (**type the name of the group to be removed.)** The group feature is deleted.

* Group feature names are case sensitive.

Related Commands

Group (6.15.22) - creates and modifies named groups of entities. DmSelect (6.12.26) - selects edges and faces of 3D solids or surfaces based on their geometric properties. DmSelectEdges (6.12.25) - selects the edges of faces and solids. SmSelect (6.28.27) - selects hard edges or form features.

Figure 6.855



6.12.17 DmMove

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Moves solids, faces or edges of a solid, or inserts using a vector.

Accessing the Command

 command bar: **dmmove** menu bar: **Model | Direct Modeling | Move** toolbar: **Direct Modeling | Move** ribbon:

- **Solid | Edit | Move**
- **Surface | Edit | Move**
- **Assembly | Transform| Move**

Quad cursor menu: **Model | Move** : **dmmove Prompts you in the command bar:** Select several entities/subentities: **(Select the geometry you want to move. Press and hold the TAB key to select hidden geometry. Press Enter to stop selecting)** Enter base point <0,0,0>: **(Specify the base point for the move operation.)** The selected geometry moves dynamically. To specify the endpoint, do one of the following:

- Click the left mouse button.
- Type a value in the dynamic input field.

Command Options

No options.

Related Commands

Align (6.4.6) - aligns one entity with another through moving, rotating, and scaling. DmChamfer (6.12.4) - creates an equal distance chamfer between adjacent faces sharing a sharp edge. DmDelete - deletes faces and solids. DmExtrude (6.12.13) - creates a 3D solids by extruding closed 2D entities or regions. DmFillet (6.12.14) - creates a smooth fillet between adjacent faces sharing a sharp edge. DmPushPull (6.12.20) - adds or removes volume from a solid by moving a face or surface. DmRevolve (6.12.22) - Creates 3D solids by revolution of closed 2D entities or regions about an axis. DmRotate (6.12.24) - rotates faces of a solid around an axis.

Figure 6.856



6.12.18 DmParallel3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Applies a parallel constraint between two faces of a solid or of different solids.

Accessing the Command

 command bar: **dmparallel3d** menu bar: **Parametric | 3D Constraints | Parallel** toolbar: **3D Constraints | Parallel** ribbon:

- **Solid |3D Constraints | Parallel**
- **Surface |3D Constraints | Parallel**
- **Assembley |3D Constraints | Parallel**

Quad cursor menu: **Constraints | Parallel** : **dmParallel3d Prompts you in the command bar:** Select first subentity [WOrld/BLock]: **(Select the first face or choose an option.)** Select first subentity [WOrld/BLock]: **(Select the second face or choose an option.)** Press the TAB key to select obscured geometry.

Command Options

Option	Description
World	Allows you to select the WCS subentity as an argument; prompts you: **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; prompts you: **Select block reference** - **(Select a block reference of which the reference coordinate system will be used as an argument.) Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**

Related Commands

MechanicalBrowserOpen (6.8.29)- displays the Mechanical Browser. DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmUpdate (6.12.33) - Forces 3d constraints to update. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids. DmRadius3d (6.12.21) - applies a radius constraint to cylindrical surfaces or circular edges.

Figure 6.857



6.12.19 DmPerpendicular3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Applies a perpendicular constraint between two faces of a solid or of different solids.

Accessing the Command

 command bar: **dmperpendicular3d** menu bar: **Parametric | 3D Constraints | Perpendicular** toolbar: **3D Constraints | Perpendicular** ribbon:

- **Solid |3D Constraints | Perpendicular**
- **Surface |3D Constraints | Perpendicular**
- **Assembley |3D Constraints | Perpendicular**

Quad cursor menu: **Constraints | Perpendicular : dmPerpendicular3d Prompts you in the command bar:** Select first subentity [WOrld/BLock]: **(Select the first face or choose an option.)** Select second subentity [WOrld/BLock]: **(Select the second face or choose an option.)** Press the TAB key to select obscured geometry.

Command Options

Option	Description
World	Allows you to select the WCS subentity as an argument; prompts you: **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; prompts you: **Select block reference** - **(Select a block reference of which the reference coordinate system will be used as an argument.) Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**

Related Commands

MechanicalBrowserOpen (6.8.29)- displays the Mechanical Browser. DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmUpdate (6.12.33) - Forces 3d constraints to update. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids. DmRadius3d (6.12.21) - applies a radius constraint to cylindrical surfaces or circular edges.

Figure 6.858



6.12.20 DmPushPull

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Adds or removes volume from a solid by moving/offsetting a face. The command can also be used to edit faces of a surface.

Accessing the Command



Command bar: **dmpushpull** Menu bar:**Model | Direct Modeling | Push/Pull** Toolbar:**Direct Modeling | Push/Pull** Ribbon: **tab | panel**

- **Solid | Edit | Push/Pull**
- **Surface | Edit | Push/Pull**

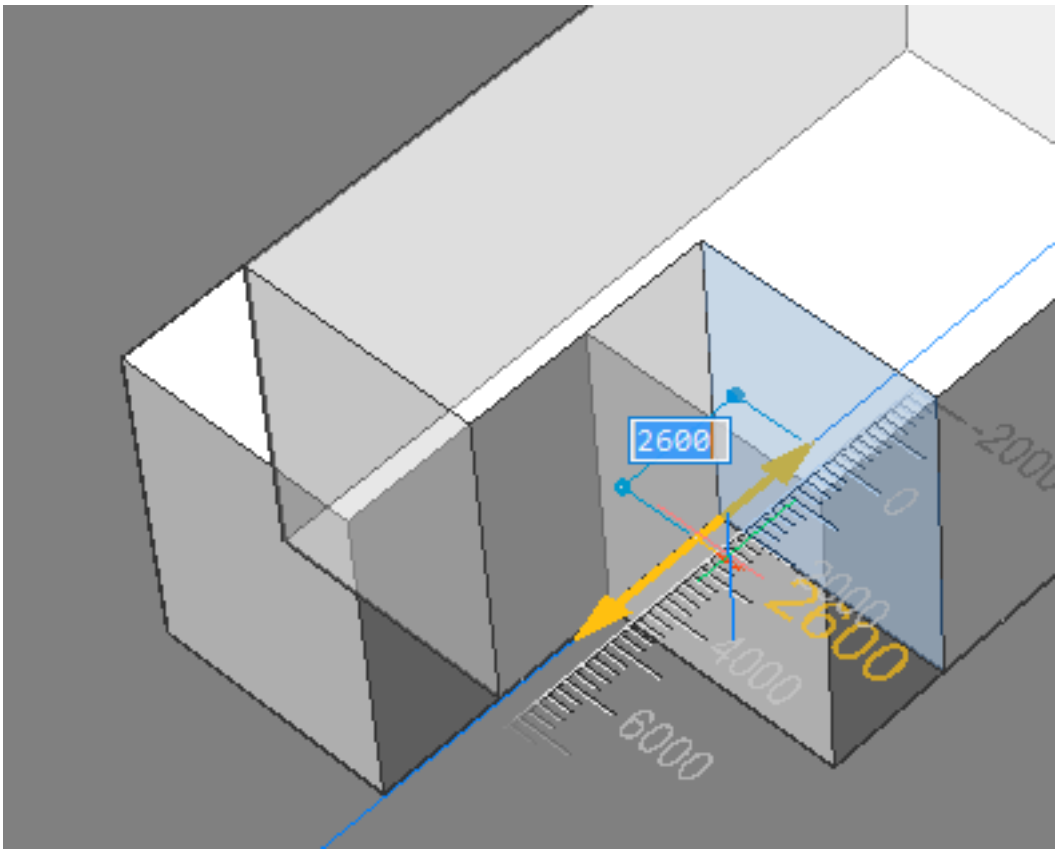
Quad: **Model | Push/Pull** : **dmpushpull Prompts you in the command bar:** Select faces* to push/pull: **(Select a face or detected boundary on the same or different solids.)** Entities in set: 1 Select faces to push/pull: **(Select more faces or detected boundaries;hit the TAB key to select hidden geometry. Press Enter to stop selecting geometry.)** Specify distance: **(Specify a point or type a distance.)**

Note	BIM: Plies of multiply compositions are selectable if the **Display Composition**property is YES, and the **Select Faces** option of the SELECTIONMODES (1.13.11) system variable is set.

The selected geometry moves dynamically; the Manipulator displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco
Do one of the following:

- (option)The current reference face highlights.Hit the TAB key to select a different reference face. All faces which are parallel to the face being pushpulled are recognized, the nearest first. Press and hold the SHIFT key when hitting the TAB key to cycle reference faces in reverse order. To select a reference face on a different solid: move the cursor over that face, then hit the TAB key. The selected face highlights.
- Click a point in the model.
- Type a value in the dynamic distance field.
- Move the cursor to adjust the value in the dynamic distance field, then click. Optionally zoom in/out to set the snap increment size of the ruler.

Figure 6.859



Command Arguments

Argument	Description
Planar faces	The faces are moved, thus adding or removing volume from the solid(s). The value of the movement vector displays in the dynamic entry field. If DMPUSHPULLSUBTRACT (1.13.11) = 1,solids being pushpulled are subtracted from other 3D solids they intersect.
Cylindrical and spherical faces	Edits the radius. The current radius displays in the dynamic entry field.
Toroidal faces	Edits the tube radius of the torus. The current radius displays in the dynamic entry field.
Spline faces	Creates offset faces and replaces selected faces with these offset faces, thus adding or removing volume from the solid(s). The current offset distance displays in the dynamic entry field.

Procedures

To control dimensions (1.18.2) Moving an insert vertically (5.3.9) Moving an insert horizontally (5.3.9) Creating T-connections (2.4.4) Creating T-connections ply-by-ply (2.4.4) Modifying the thickness of a solid (2.4.4) Modifying the height of a wall (2.4.4) Modifying the height of a wall ply-by-ply (2.4.4) Setting the distance between parallel walls (2.4.4) Creating a parallel copy of a wall (2.4.4) Creating a partial copy of a wall (2.4.4)

Related Commands

DmChamfer (6.12.4) - creates an equal distance chamfer between adjacent faces sharing a sharp edge. DmDelete - deletes faces and solids. DmExtrude (6.12.13) - creates a 3D solids by extruding closed 2D entities or regions. DmFillet (6.12.14) - creates a smooth fillet between adjacent faces sharing a sharp edge. DmMove (6.12.17) - moves the selected faces or edges of a solid using a vector. DmRevolve (6.12.22) - Creates 3D solids by revolution of closed 2D entities or regions about an axis. DmRotate (6.12.24) - rotates faces of a solid around an axis. BimDrag (6.7.8) - Moves 3D solids perpendicular to a selected face by dragging that face. Optionally preserves connections to other solids. LConnect (6.20.19) -Creates an L-connection between solids. TConnect (6.29.10) -Connects solid faces to other solids.

Figure 6.860



6.12.21 DmRadius3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Applies a radius constraint to cylindrical surfaces or circular edges.

Accessing the Command

 command bar: **dmradius3d** menu bar: **Parametric | 3D Constraints | Radius** toolbar: **3D Constraints | Radius** ribbon:

- **Solid |3D Constraints | Radius**
- **Surface |3D Constraints | Radius**
- **Assembley |3D Constraints | Radius**

Quad cursor menu: **Constraints | Radius** : **dmRadius3d** Select a face or circular edge: **(Select a cylindrical surface or a circular edge)** Enter radius value <current value>: **(Either press Enter to accept the current value, or type a new value.)** Press the TAB key to select obscured geometry. In case the surface of an elliptical cylinder is selected, the surface is converted to a circular cylinder.

Command Options

No options.

Related Commands

MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser. DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmUpdate (6.12.33) - Forces 3d constraints to update. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids.

Figure 6.861



6.12.22 DmRevolve

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Creates 3D solids or surfaces by revolution of open or closed 2D entities or regions about an axis.

Accessing the Command

 **Revolve solid**  **Revolve surface** command bar: **dmrevolve** menu bar: **Model | Direct Modeling | Revolve** toolbar: **Direct Modeling | Revolve solid** and **Revolve surface** ribbon:

- **Solid | Create | Revolve solid**
- **Surface | Create | Revolve surface**

Quad cursor menu: **Model | Revolve solid** and **Revolve surface** : **dmRevolve** When launched from the command bar, menu bar, toolbar or ribbon: **Prompts you in the command bar:** Select entities/subentities to revolve: Select planar entities/subentities to revolve or set [MOde]: **(Select a region or a closed 2D entity (closed polylines, circles, and ellipses are accepted) or a face of solid.)** Entities/subentities in set: 1 Select planar entities/subentities to extrude or set [MOde]: (**Select another entity or subentity or press Enter to stop selecting entities.)** When launched from the Quad cursor menu: First select the entity, entities or boundary to be extruded. Closed 2D entities (1)(2) and regions (2) are accepted. Closed boundaries in the XY-plane of the current coordinate system (4) or on the face of a solid (5) are recognized automatically if the **Select detected boundaries** option () on the **Selection Modes** toolbar is active.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-dmrevolve-1	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-dmrevolve-2	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-dmrevolve-3	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-dmrevolve-4	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-dmrevolve-5
1	2	3	4	5

Do one of the following:

- Launch the command when the desired entity or boundary highlights.
- To revolve multiple entities: compose a selection set, then launch the command.

Solid mode: Select axial entity or define axis by [2Points/Xaxis/Yaxis/Zaxis] <2Points>: **(Select a linear entity, or select an option, or right click to define the axis by two points.)** The selected entities are revolved dynamically. Move the mouse to specify the angle direction. Specify angle of revolving or [Auto/Create/SUbtract/Unite] <Auto>: **(Choose an option, then specify the angle.)**

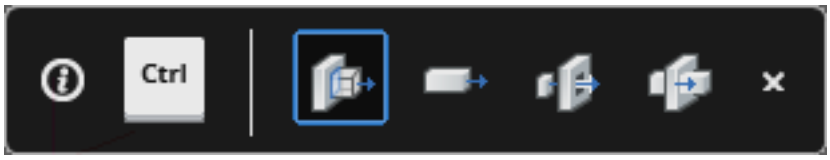
Surface mode: Select axial entity or define axis by [2Points/Xaxis/Yaxis/Zaxis] <2Points>: **(Select a linear entity, or select an option, or right click to define the axis by two points.)** The selected entities are revolved dynamically. Move the mouse to specify the angle direction. Specify angle of revolving or [Auto/Create] <Create>:

Command Options

Option Description Mode Allows to create either solids or surfaces. Prompts you: **Choose type of created entity [SOlid/SUrface] <Solid>:**

- Solid: creates solids.
- Surface: creates surfaces.

Figure 6.862



The **Revolve** tool ( ) starts the command in **Solid** mode. The **Revolve Surface** tool ( ) starts the command in **Surface** mode. Select Entities/Subentities to extrude Click an entity or choose an option in the dmRevolve prompt menu. Depending on the value of the DELOBJ (1.18.1) (Delete Object) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities. Select a linear (sub)entity as axis or define axis by <2Points>/Xaxis/Yaxis/Zaxis: Select a line or an edge of a solid. Edges of a solid can only be selected if the **Edges** option of the SELECTIONMODES (1.13.11) system variable is set. Click the **Edges** tool button ( ) on the **Selection Modes** toolbar to toggle this setting. A pressed button indicates the option is set. 2Points Specify start point of axis <0,0,0>: Specify the start point of the axis or press Enter to accept the origin point of the current UCS. Specify endpoint of the axis or <use axis parallel to view direction>: Specify the end point of the axis or press Enter to set the axis parallel to the view direction (= perpendicular to the screen). Zaxis Uses the x axis of the current UCS as the axis of revolution Yaxis Uses the y axis of the current UCS as the axis of revolution. Xaxis Uses the z axis of the current UCS as the axis of revolution. Press the CTRL key to cycle the various options. Auto* The result depends on the extrusion direction of the 3D solid and the value of the DMEXTRUDEMODE (1.13.11) system variable.

- when revolved inside the existing solid, the new solid is subtracted from the existing solid.
- when revolved outside the existing solid:
- If DMEXTRUDEMODE = 0: the new solid is unified with the existing solid (Auto (Subtract or Unite))
- If DMEXTRUDEMODE = 1: a separate solid is created (Auto (Subtract or Create))

Create* Regardless of the revolving direction, a new 3D solid is created. Subtract* The 3D solid is subtracted from each interfering existing solid. Unite* The new 3D solid is unified with each interfering existing solid. * Repeatedly hit the CTRL key during the dynamic display of the extrusion to cycle the various options: Auto / Create / Subtract / Unite showing the result of the currently selected option. If Hotkey Assistant (HKA) is ON the Hotkey Assistantwidget displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmExtrude

-  : Auto
-  : Create
-  : Subtract
-  : Unite

Related Commands

Loft (6.20.37) - creates 3D solids passing through two or more cross sections. Revolve (6.26.30) - revolves open or closed 2D entities and solid edges and faces to turn them into 3D solids or surfaces. Sweep (6.27.60) - creates 3D solids or surfaces by sweeping profiles (open or closed 2D entities or 3D subentities) along an open or closed path. DmChamfer (6.12.4) - creates an equal distance chamfer between adjacent faces sharing a sharp edge. DmDelete - deletes faces and solids. DmExtrude (6.12.13) - creates a 3D solids by extruding closed 2D entities or regions. DmFillet (6.12.14) - creates a smooth fillet between adjacent faces sharing a sharp edge. DmMove (6.12.17) - moves the selected faces or edges of a solid using a vector. DmPushPull (6.12.20) - adds or removes volume from a solid by moving a face or surface. DmRotate (6.12.24) - rotates faces of a solid around an axis.

Figure 6.863



Figure 6.864

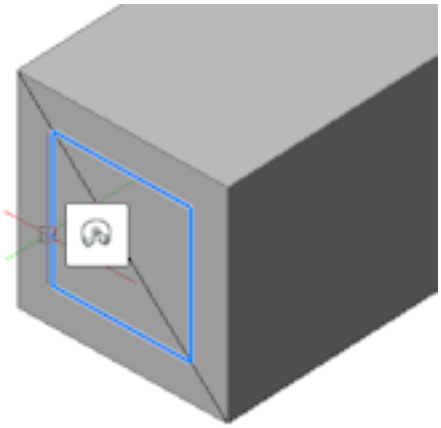


Figure 6.865



Figure 6.866



Figure 6.867

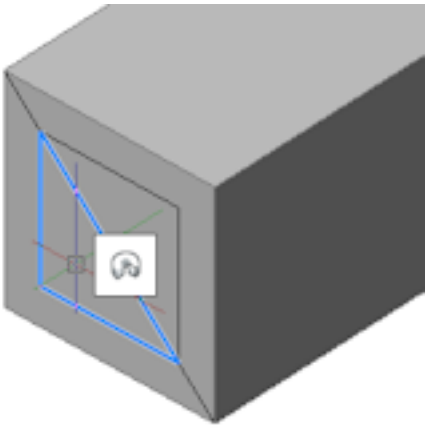


Figure 6.868



6.12.23 DmRigidSet3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Allows to define a set of entities or subentities as a rigid body.

Accessing the Command


command bar: **dmrigidset3d** menu bar: **Parametric | 3D Constraints | Rigid Set** toolbar:**3D Constraints | Rigid Set** ribbon:

- **Solid |3D Constraints | Rigid Set**
- **Surface |3D Constraints | Rigid Set**
- **Assembley |3D Constraints | Rigid Set**

Quad cursor menu: **Constraints | Rigid Set** : **dmrigidset3d Prompts you in the command bar:** Select edges, faces, 3D entities or [WOrld/BLock]: **(Select entities or subentities or choose an option.)** Entities in set: 1 Select edges, faces, 3D entities or [WOrld/BLock]: **(Select more entities or subentities or choose an option, or press Enter.)** Press the TAB key to select obscured geometry. All members of a rigid set are moved (DmMove (6.12.17) command) and rotated (DmRotate (6.12.24) command) together, while their relative positions within the rigid set are preserved.

Command Options

Option	Description
World	Allows you to select the WCS subentity as an argument; prompts you: **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; prompts you: **Select block reference** - **(Select a block reference of which the reference coordinate system will be used as an argument.) Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**

Related Commands

MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmUpdate (6.12.33) - Forces 3d constraints to update. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids. DmRadius3d (6.12.21) - applies a radius constraint to cylindrical surfaces or circular edges.

Figure 6.869



6.12.24 DmRotate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Rotates faces of a solid around an axis.

Accessing the Command

 command bar: **dmrotate** menu bar: **Model | Direct Modeling | Rotate** toolbar: **Direct Modeling | Rotate** ribbon:

- **Solid | Edit | Rotate**
- **Surface | Edit | Rotate**
- **Assembly | Transform | Rotate**

Quad cursor menu: **Model| Rotate** : **dmrotate Prompts you in the command bar:** Select several entities/subentities: **(Select the faces or detected boundaries you want to rotate. Press and hold the TAB key to select hidden geometry. Press Enter to stop selecting.)** Select axial entity or define axis by [2Points/Xaxis/Yaxis/Zaxis] <2Points>: **(Choose an option or right click to define the axis by 2 points.)**

Command Options

Option	Description
2Points	Prompts you in the command bar. **Specify start point of axis <0,0,0>** - **(Specify a point or right click to us the origin of the current coordinate system.) Specify end point of axis or <use axis parallel to view direction>** - **(Specify a point or right click to define the axis parallel to the current view direction (= perpendicular to the screen).)**
Xaxis	Rotates the selection around the X-axis of the current coordinate system.
Yaxis	Rotates the selection around the Y-axis of the current coordinate system.
Zaxis	Rotates the selection around the Z-axis of the current coordinate system.

Related Commands

DmChamfer (6.12.4) - creates an equal distance chamfer between adjacent faces sharing a sharp edge. DmDelete - deletes faces and solids. DmExtrude (6.12.13) - creates a 3D solids by extruding closed 2D entities or regions. DmFillet (6.12.14) - creates a smooth fillet between adjacent faces sharing a sharp edge. DmMove (6.12.17) - moves the selected faces or edges of a solid using a vector. DmPushPull (6.12.20) - adds or removes volume from a solid by moving a face or surface. DmRevolve (6.12.22) - Creates 3D solids by the revolution of closed 2D entities or regions about an axis.

Figure 6.870



6.12.25 DmSelectEdges

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Selects the edges of faces and solids.

Accessing the Command

 command bar: **dmselectedges** menu bar:**Model | Smart Selection | Edges** toolbar: **Direct Modeling| Edges** ribbon:**Solid | Select | Edges** Quad cursor menu:**Select| Edges** : **dmselectedges Prompts you in the command bar:** Select faces or solids: **(Select faces or solids)** Entities/subentities in set: 2 Select faces or solids: **(Select more faces and or solids or press Enter to stop)** The edges of all selected faces and solids are selected. Launch a command to be applied to the selection, e.g. DmFillet or DmChamfer.

Command Options

Press and hold the SHIFT key, then click the edges to be removed from the selection set.

Procedures

Selecting edges of faces or solids (1.12.5)

Related Commands

DmSelect (6.12.26) - selects edges and faces of 3D solids or surfaces based on their geometric properties. SmSelect (6.28.27) - selects hard edges or form features.

Figure 6.871



6.12.26 DmSelect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Selects edges and faces of 3D solids or surfaces based on their geometric properties. The command allows to select a group of faces, which form a geometric feature, such as a protrusion, a depression, a blend, or a blend network. You can combine different options of this command, e.g. select all blends, which radius is less than given value.

Accessing the Command

command bar: **dmselect** menu bar: **Model | Smart Selection** ribbon:**Solid | Selectcommand group** Quad cursor menu:**Selectcommand group** : **dmselect Prompts you in the command bar:** Select [Selection/Attribute/Relation/Primitive/fea <Finish>: **(Select an option.)** If a selection set of faces or edges is present, the initial prompt is: Use preselection as [sUbset/Sample/sEed] <Sample>: **(Select an option.)** Selecting Cancel in a sub-prompt or pressing the Esc button returns to main prompt.

Command Options

Option	Description
Selection	Allows to select a set of subentities. Prompts you: **Select [sUbset/Sample/sEed] <Sample>:** • Subset: only faces/edges in the subset can be selected. • Sample: the preselected set of faces/edges should be considered as samples, of which the geometric characteristics (length, area, radius) will be used as parameters for the selection. • Seed: the preselected set of faces /edges be should be considered as seed elements for the selection of features such as blends, blend networks, protrusion or depressions.

Attribute	Prompts you: **Select [Radius/Area/Length/Type/Exclude short fillets] <Radius>:** • Radius: selects faces/edges by radius. • Area: selects faces by area. • Length: selects edges by length. • Type: selects faces/edges by their geometric type. If you choose any of the above options, you are promted: **Select [Current sample/Value] <Current sample> :** • Current Sample: Press <ENTER> to use the value of the sample face/edge. • Value:type a value in the command bar. In case of a non-zero value, you are prompted: **Is [Less/Less or eQual(LQ)/Equal/Greater or eQual(GQ)/Greater] <Equal>:(Select an option.)** • Exclude short fillets: excludes short fillets from the selection, which might be useful when working with imported sheet metal parts.
Relation	Prompts you: **Select [Coincident/Parallel/coAxial/Smooth] <Parallel>:** • Coincident: selects coincident faces/edges. • Parallel: selects parallel faces/edges. • Coaxial: selects coaxial faces/edges. • Smooth: selects the smooth side of a part.
Primitive	Prompts you: **Select [Face/Edge/Loop/edge Network] <Face>:** • Face: selects faces. • Edge: selects edges. • Loop: selects edge loops. Please, not that the preselection will determine the result. It is necessary to select a set of connected faces. The command will then select all edges, which describe the border of the selection set; internal edges will be skipped. If the result contains multiple loops and you want only one, the preselection should contain a set of connected faces one edge of the desired output loop. If none of the loops of the selection set contains the selected edge, the output will be empty. • Edge Network: finds an adjacent edge which has similar convexity for a given input edge.

feaTure	Prompts you: **Select [Protrusion/Depression/Blend/blend Network] <blend Network>:** • Protrusion: selects protrusions. • Depression: selects depressions. • Blend: selects blends. • Blend Network: selects blend networks.
Finish	Concludes the command and selects faces/edges according to the selected options.

Related Commands

QSelect (6.25.5) - composes a selection set using filters. Select (6.27.17) - selects one or more entities; can be accessed later with the Previous option. SelectAlignedFaces (6.27.16) - selects all faces in a model which are coplanar with a selected face. SelectAlignedSolids (6.27.18) - selects all solids in a model of which a face is coplanar with a selected face.. SelectConnectedSolids (6.27.20) - selects all faces in a model which are connected to a selected face. SelectSimilar (6.27.21) - selects all entities of the same type as the current selection, based on a set of matching properties, stored in the SELECTSIMILARMODE (1.13.11) system variable.

Figure 6.872



6.12.27 DmSimplify

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Simplifies the geometry and topology of 3d solid entities. DmSimplify removes unnecessary edges and vertices, merges seam edges, and replaces the geometry of faces and edges by analytic surfaces and curves, if possible within the user-specified tolerance. It is recommended to always run this command on imported 3d solid geometry.

Accessing the Command


command bar: **dmsimplify** menu bar:**Model | Direct Modeling | Simplify** toolbar:**Direct Modeling| Simplify** ribbon:

- **Solid | Edit | Simplify**
- **Sheet Metal | Heal | Simplify**

Quad cursor menu:**Model | Simplify** : **dmsimplify Prompts you in the command bar:** Select entities to simplify or [Entire model/SEttings] <Entire model>: **(Select entities or press Enter to select the entire model)**

- Geometry simplified
- Topology simplified

Command Options

Option	Description
Settings	Allows to access the command settings Prompts you in the command bar: **Simplify [Geometry/Topology]/<Back>:** • Geometry: define the Geometry settings • Topology: define the Topology settings • Back: return to the default command prompt
Geometry	Prompts you in the command bar: **Select geometric simplification tolerance or [ON/OFF/Topology] <1e-7>:** Choose an option: • Type a value to set the simplification tolerance. • ON : switches geometrical simplification on. • OFF : switches geometrical simplification off. • Topology : define the Topology settings.
Topology	Prompts you in the command bar: **Select topology simplification mode [ON/OFF/Geometry] <ON>:** Choose an option: • ON : switches topological simplification on. • OFF : switches topological simplification off. • Geometry : define the geometrical settings.

Related Commands

Import (6.17.10) - imports the geometry from external files into the current drawing. DmStitch (6.12.28) - converts a set of region and surface entities that bound a watertight area to a 3d solid. DmAudit (6.12.2) - fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces). DmAuditAll (6.12.3) -checks and heals flaws in drawings inserted as external references. DmSimplifyAll (6.12.29) - simplifies the geometry and topology of solids,

Figure 6.873



6.12.28 DmStitch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Converts a set of region and surface entities that bound a watertight area to a 3d solid. If DMREPORTPANEL (1.13.11)= ON, BricsCAD reports the results in the Report (6.8.32) panel instead of the command bar.

Accessing the Command

 command bar: **dmstitch** menu bar: **Model | Direct Modeling | Stitch** toolbar: **Direct Modeling| Stitch** ribbon:

- **Solid | Edit | Stitch**
- **Surface |Edit | Stitch**

Quad cursor menu:**Model | Stitch** : **dmstitch Prompts you in the command bar:** Select surfaces to stitch or choose [validation MOde] <Auto>: **(Select regions or surfaces.)** Entities in set: XX Select surfaces to stitch or choose [validation MOde] <Auto>: **(Select more regions or surfaces or right click.)** Created valid 3D solid.

Command Options

Option	Description
Validation Mode	Allows to set the validation mode either manual or automatic. Prompts you: **Choose validation mode [Auto/Manual] <Auto>:** - **(Select a validation mode.)** The command reports success or failure. In automatic validation mode, the command creates only valid 3d solid entities without free or non-manifold edges. If you want to create an invalid 3d solid or understand the reason of a failure, run the command in manual validation mode. The command then reports problems (free or non-manifold edges) and highlights them. You can either accept the invalid 3d solid by pressing Enter or refuse it by pressing Cancel. The selected validation mode is stored and re-used for future calls of the command.

Related Commands

DmExtrude (6.12.13) - creates 3D solids by extruding closed 2D entities, regions or closed boundaries. DmSimplify (6.12.27) - simplifies the geometry and topology of 3d solid entities. DmAudit (6.12.2) - fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces).

Figure 6.874



6.12.29 DmSimplifyAll

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Simplifies thegeometry in solids, within drawings inserted as external references. DmSimplifyAll removes unnecessary edges and vertices, merges seam edges, and replaces the geometry of faces and edges by analytic surfaces and curves, if possible within the user-specified tolerance. It is recommended to always run this command on imported drawings with inserted external references.

Accessing the Command

command bar: **dmsimplifyall** : **dmsimplifyall Prompts you in the command bar:** Select entities to simplify or [Entire model/SEttings] <Entire model>: **(Select entities or press Enter to select the entire model)** Geometry simplified Topology simplified

Command Options

Option	Description
Settings	Allows to access the command settings Prompts you in the command bar: **Simplify [Geometry/Topology]/<Back>:** • Geometry: define the Geometry settings • Topology: define the Topology settings • Back: return to the default command prompt
Geometry	Prompts you in the command bar: **Select geometric simplification tolerance or [ON/OFF/Topology] <1e-7>:** Choose an option: • Type a value to set the simplification tolerance. • ON : switches geometrical simplification on. • OFF : switches geometrical simplification off. • Topology : define the Topology settings.
Topology	Prompts you in the command bar: **Select topology simplification mode [ON/OFF/Geometry] <ON>:** Choose an option: • ON : switches topological simplification on. • OFF : switches topological simplification off. • Geometry : define the geometrical settings.

Related Commands

Import (6.17.10) - imports the geometry from external files into the current drawing. DmStitch (6.12.28) - converts a set of region and surface entities that bound a watertight area to a 3d solid. DmAudit (6.12.2) - fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces). DmAuditAll (6.12.3) -checks and heals flaws in drawings inserted as external references. DmSimplify (6.12.27) - simplifies the geometry and topology of 3d solid entities. Erase (6.13.8) - erases entities from the drawing. Deletes 3D solid faces and face edges.

Figure 6.875



6.12.30 DmTangent3d

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Applies a tangent constraint between a face and a curved surface of different solids.

Accessing the Command


command bar: **dmtangent3d** menu bar: **Parametric | 3D Constraints | Tangent** toolbar: **3D Constraints | Tangent** ribbon:

- **Solid |3D Constraints |Tangent**
- **Surface |3D Constraints | Tangent**
- **Assembley |3D Constraints | Tangent**

Quad cursor menu: **Constraints | Tangent** : **dmtangent3d Prompts you in the command bar:** Select first subentity [WOrld/BLock]: **(Select the first cylindrical (circular or elliptical), spherical or conical surface or choose an option.)** Select second subentity [WOrld/BLock]: **(Select the second cylindrical (circular or elliptical), spherical or conical surface or choose an option.)** Press the TAB key to select obscured geometry.

Command Options

Option	Description
World	Allows you to select the WCS subentity as an argument; prompts you: **Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**
Block	Allows you to choose as an argument the subentity of the coordinate system of the block; prompts you: **Select block reference** - **(Select a block reference of which the reference coordinate system will be used as an argument.) Select reference coordinate system subentity [Origin/X axis/Y axis/Z axis/XY plane/YZ plane/ZX plane]** - **(Select a reference coordinate system subentity as an argument.)**

Related Commands

MechanicalBrowserOpen (6.8.29) - displays the Mechanical Browser. DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmUpdate (6.12.33) - Forces 3d constraints to update. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids. DmRadius3d (6.12.21) - applies a radius constraint to cylindrical surfaces or circular edges.

Figure 6.876



6.12.31 DmThicken

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, faces of 3D solids, regions and wire entities. The following wire entities are accepted: lines, polylines, circles, ellipses, arcs, helices, splines

Accessing the Command


command bar: **dmthicken** menu bar: **Model | Direct Modeling | Thicken** toolbar: **Direct Modeling | Thicken** ribbon:**Solid | Deform | Thicken** Quad cursor menu:**Model | Thicken** : **dmthicken Prompts you in the command bar:** Select entities/subentities to thicken: **(Select a face of a solid, a surface or a wire entity.)** Entities in set: 1 Select entities/subentities to thicken: **(Select more solid faces or surfaces or press Enter.)** S pecify thickness value: **(Specify a point or type a value.)** The thickening direction is perpendicular at every point to the selected entity or entities. If you thicken a multi-face surface of several adjacent faces of one 3D solid/surface, then the corresponding faces remain adjacent on the opposite side of the new 3D solid(s). This differs the DmThicken command from DmExtrude (6.12.13). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/Dr
From left to right: extruded faces of a prism - thickened faces of a prism - thickened face of a cylinder - thickened fillets. When a wire entity is selected, a circle with the radius of the specified thickness value is swept along the wire: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/DirectModeling/DmThickenWires.png})

Figure 6.877

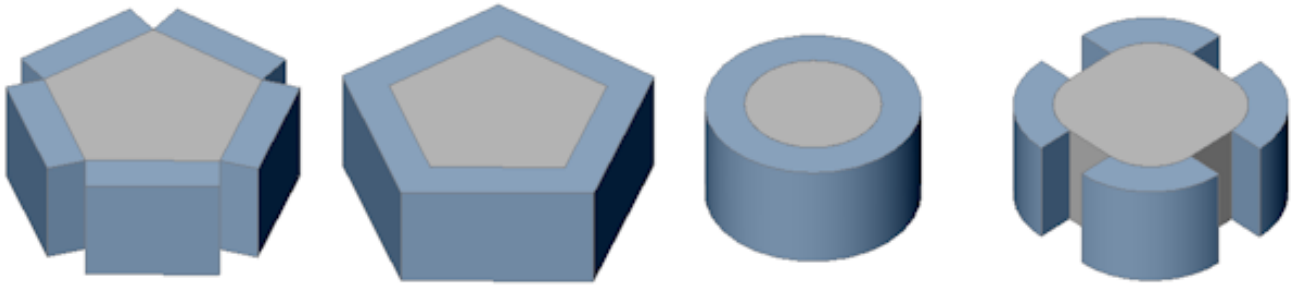
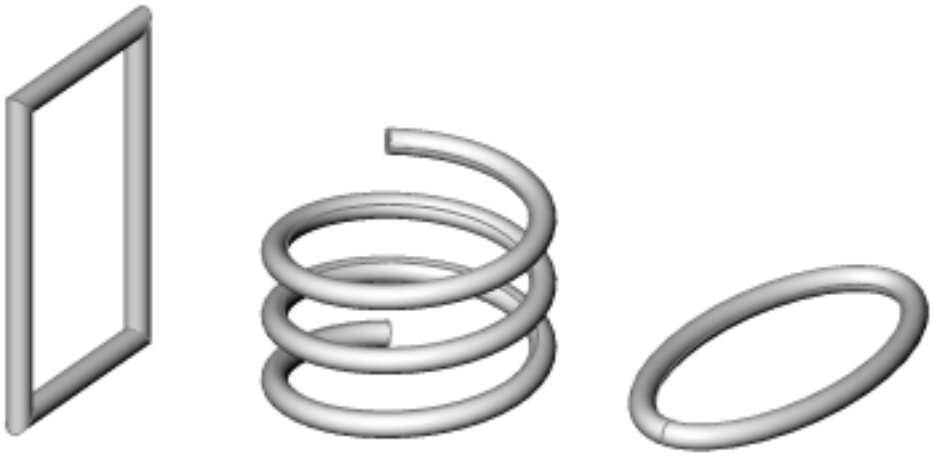


Figure 6.878



Command Options

No command options.

Related Commands

DmExtrude (6.12.13) - creates 3D solids or 3D surfaces by extruding closed 2D entities, faces of 3D solids, regions or closed boundaries. Optionally adds volume to solids or removes volume from solids. Sweep (6.27.60) - creates 3D solids or 3D surfaces by sweeping profiles (open or closed 2D entities or 3D subentities) along an open or closed path.

Figure 6.879



6.12.32 DmTwist

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

Accessing the Command

 command bar: **dmtwist** menu bar: **Model | Direct Modeling | Deform | Twist** toolbar: **Deform | Twist** ribbon:

- **Solid | Deform | Twist**
- **Surface | Deform | Twist**

Quad cursor menu: **Model | Twist** : **dmtwist Prompts you in the command bar:** Select object to twist: **(Select a solid, a surface or a region.)** Entities in set: 1 Select object to twist: **(Select more objects or right click.)** Enter start point of twisting axis: **(Specify a point where the twisting deformation will start.)** Enter end point of twisting axis: **(Specify a point where the twisting deformation will end.)** Pick start point of twisting: **(Specify a point.)** Specify twist angle or set [Continuity]: (**Specify an angle dynamically or type a value or type C to set the Continuity.)**

Command Options

Option	Description
Continuity	Allows to control whether the transition between the deformed and non-deformed part is sharp or smooth. Prompts you: **Specify Continuity [G0/G1/G2]** - **(Choose an option.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Direc G0: No continuity (left), G1: Tangent Faces (middle) and G2: Curvature continuity (right)

Related Commands

DmDeformCurve (6.12.9) - deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves. DmDeformMove (6.12.10) - deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges. DmDeformPoint (6.12.11) - deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction. Loft (6.20.37) - creates 3D solids passing through two or more cross sections.

Figure 6.880

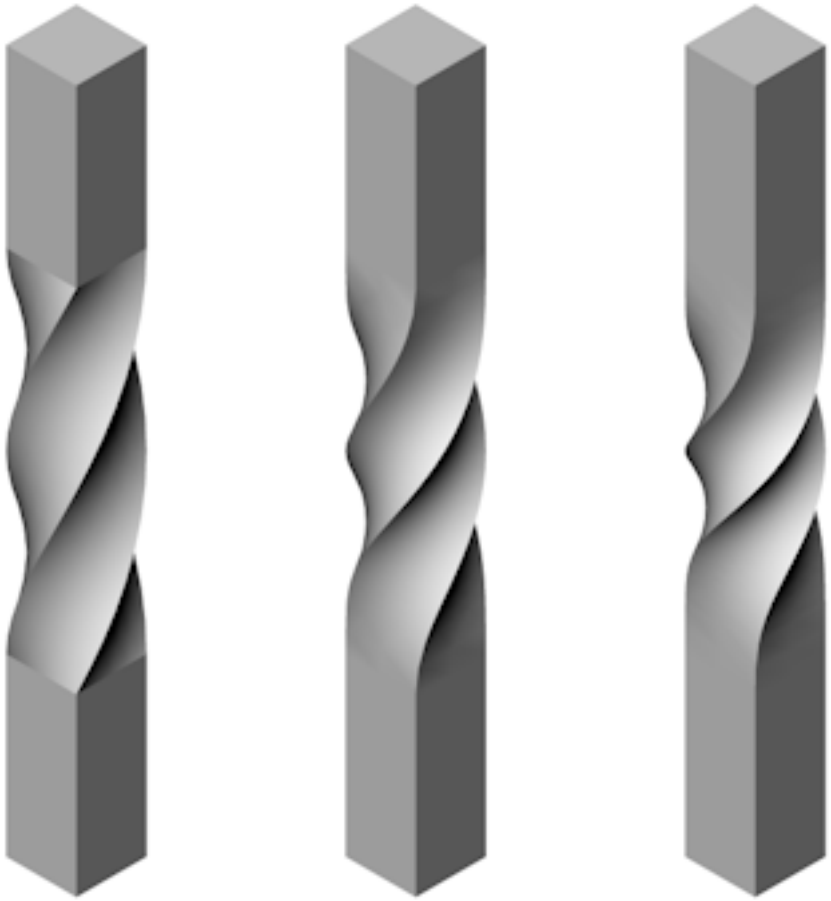


Figure 6.881



6.12.33 DmUpdate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Forces 3d constraints to update.

Accessing the Command


command bar: **dmupdate** menu bar: none toolbar: **3D Constraints | Update** : **dmUpdate**

Command Options

No options

Related Commands

MechanicalBrowserOpen (6.8.29)- displays the Mechanical Browser. DmConstraint3D (6.12.7) - applies geometric relationships and dimensional constraints between subentities (faces, surfaces, edges, ...) of 3D entities. DmCoincident3d (6.12.5) - applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. DmConcentric3d (6.12.6) - applies a concentric constraint between two cylindrical, spherical or conical surfaces. DmFix3d (6.12.15) - Applies a fixed constraint to a solid or to an edge or a face of a solid. DmParallel3d (6.12.18) - applies a parallel constraint between two faces of a solid or of different solids. DmPerpendicular3d (6.12.19) - applies a perpendicular constraint between two faces of a solid or of different solids. DmRigidSet3d (6.12.23) - Allows to define a set of entities or subentities as a rigid body. DmTangent3d (6.12.30) - applies a tangent constraint between a face and a curved surface of different solids. DmAngle3d (6.12.1) - applies an angle constraint between two faces of a solid or of different solids. DmDistance3d (6.12.12) - applies a distance constraint between two subentities of a solid or of different solids. DmRadius3d (6.12.21) - applies a radius constraint to cylindrical surfaces or circular edges.

6.13 E

6.13.1 EAttEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits the values and most properties of all attributes contained in a single block, through a dialog box (short for ”enhanced attribute editor”). 

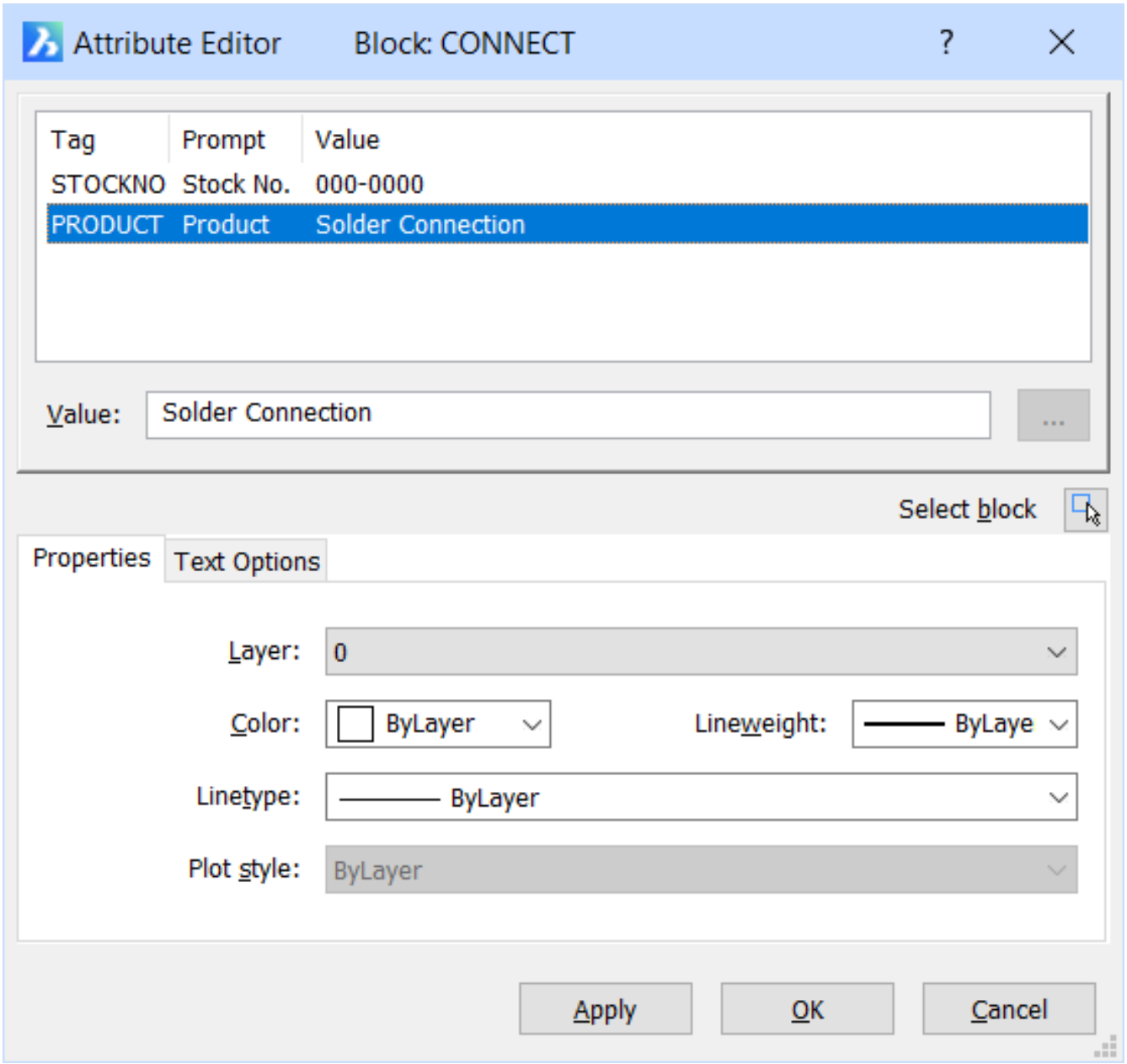
Figure 6.882

Shape Classic Pro Platinum Mechanical BIM

Accessing the Command

command bar: **eattedit** alias: **ate** menu bar: **Tools | Attributes | Edit Block Attributes** ribbon: **Insert | Blocks | Edit Attributes | Edit Block Attributes** toolbar: **Attributes |** : **EAttEdit Prompts you in the command bar:** Select a block: (**Choose a block that contains at least one attribute)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/eattedit-dlg-1.png}) Change the values and/or properties of the attributes, and then click **OK**. The changes you made apply only to the attributes in the block you selected; the changes are not applied globally. Click the **Apply** button to apply the changes to the attributes, and thenclick the **Select Block** button to choose another block, this time one with entities. When you select a block with no attributes, the command displays the dialog box, but with no entries for attributes and you cannot add attributes; click the **Select Block** button to choose another block, this time one with entities. When you select an entity that is not a block, the program reports, ”The selected entity is not a block,” and then returns to the command prompt.

Figure 6.883


Attribute Editor
Block: CONNECT
Tag
Prompt
Value
STOCKNO
Stock No.
000-0000
PRODUCT
Product
Solder Connection
Value:
Solder Connection
Select block
Properties
Text Options
Layer:
0
Color:
ByLayer
Lineweight:
ByLaye
Linetype:
ByLayer
Plot style:
ByLayer
Apply
OK
Cancel

Command Options

Option	Description
Value	Edits the value of the attribute.
Select Block	Chooses another block.
Properties Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/ea properties-tab-1.png})
Layer	Specifies the layer name; to change it, choose another one from the droplist.
Color	Specifies the color; to change it, choose another one from the droplist, or else click **Select Color** to access the Color dialog box.
Lineweight	Specifies the lineweight; to change it, choose another one from the droplist.
Linetype	Specifies the linetype; to change it, choose another one from the droplist, or else choose **Load** to access the Drawing Explorer dialog box to load additional linetypes
Plot Style	Specifies the plot style; to change it, choose another one from the droplist. (This option is available only when plot styles are enabled in the drawing; otherwise, the droplist is grayed-out.)
Text Options Tab	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/ea textoptions-tab-1.png})
Text Style	Specifies the text style name; to change it, choose another one from the droplist.
Justification	Specifies the justification; to change it, choose another one from the droplist.
Height	Specifies the height of the text.
Rotation	Specifies the rotation angle of the text.
Upside Down	Toggles whether the text is displayed upside down.
Backwards	Toggles whether the text is displayed backwards.
Width Factor	Specifies the width of the characters; use a value of less than 1.0 (such as 0.85) to squeeze more text into a small space.
Oblique Angle	Specifies the slant of the characters; can be used to mimic italic text.
Annotative	Toggles whether the text follows annotative scaling.
Apply	Applies the changes you made, without leaving the dialog box. Click **Select Block** to edit the attributes of another block.
OK	Applies the changes you made to the attribute, and then exists the dialog box.
Cancel	Exits the dialog box, discarding the changes you made to the attribute.

Procedures

- Using the Attribute Editor (1.22.13)
- Editing attributes in the Properties bar (1.22.13)

Figure 6.884

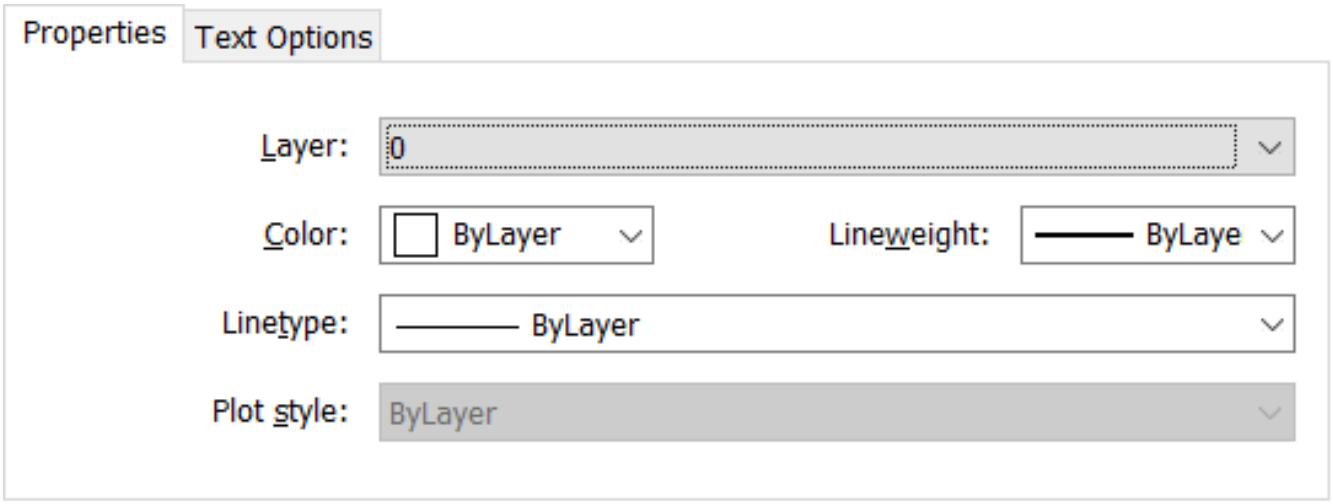
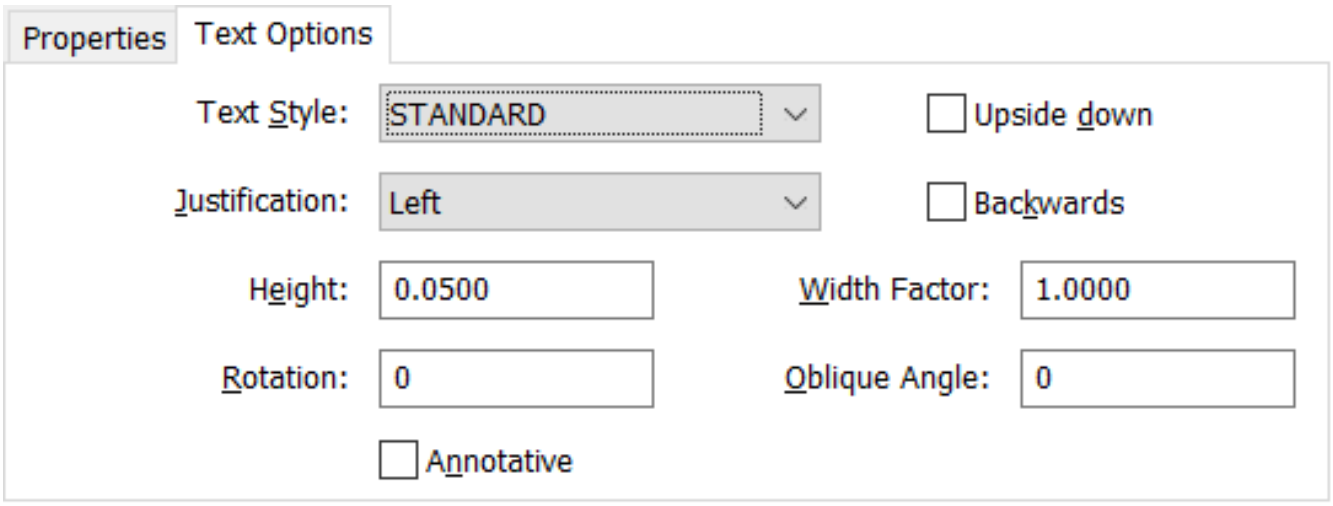


Figure 6.885



Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttDisp (6.4.35) - toggles the display of attribute text. AttEdit (6.4.36) - edits attribute values and properties. -AttExt (6.4.29) - extracts attributes from drawings to data files through the command bar. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. DataExtraction (6.10.1) - exports entity properties, block attributes and drawing information to a an external file. DdAtte (6.10.13) - edits the values of attributes through a dialog box. Explode (6.13.13) - reduces attributes to tags. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. Properties (6.24.49) - edits attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place.

Figure 6.886



6.13.2 EdgeSurf

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates 3D Coons mesh surface patches between four entities (short for "edge-defined surface").

Accessing the Command


command bar: **edgesurf** ribbon:**Model | Meshes | Coons Surface** toolbar: **Meshes | 3D Mesh Surfaces |** : **EdgeSurf Prompts you in the command bar:** Select the first of four connected linear entities for edge surface: **(Pick an entity)** Select second edge: **(Pick a second entity)** Select third edge: **(Pick a third entity)** Select last edge: **(Pick a fourth entity)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image18.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image14.jpg})

Figure 6.887

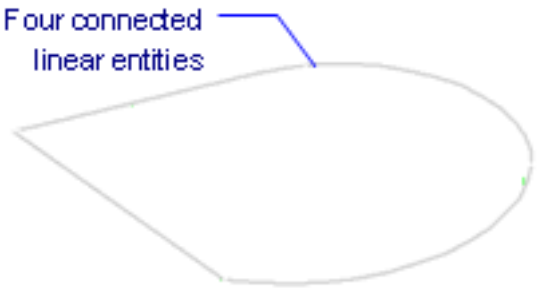
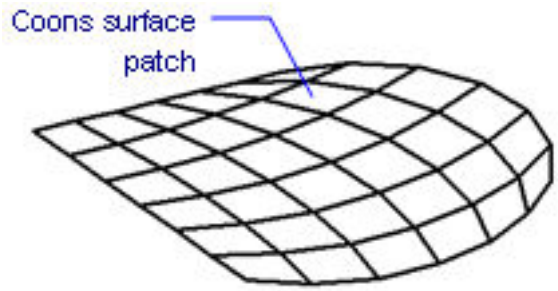


Figure 6.888



Command Options

Select the first of four connected linear entities for edge surface	Pick an entity. The entities must be: • Linear - open entities, such as lines, arcs, open splines, and open polylines • Connected - touching or overlapping, so that they form a closed area
Select second edge	Pick a second entity
Select third edge	Pick a third entity
Select last edge	Pick a fourth entity

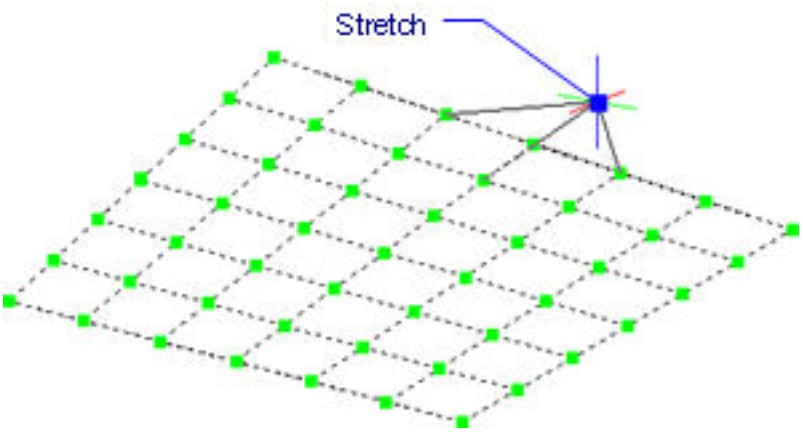
Grips Editing

Edge meshes can be edited directly through grips:

1. Select the mesh. Notice that it has several grips.
2. Drag a grip to stretch the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image488.jpg})

Figure 6.889



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.890



6.13.3 EditEData

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates, edits, and deletes extended entity data (short for "edit entity data").

Accessing the Command



command bar: **editedata** transparent: **'editedata** menu bar: **Tools | Entity Data | Create or Edit Entity** toolbar: **Entity Data |**  : **EditEData Prompts you in the command bar:** Select entity to create or edit Entity Data: **(Choose any entity in the drawing)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V dlg.png}) Click **Insert**, and then select entity data types from the menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/ insert-mnu.png}) Select **Application Name**, and then the other data fields become available.

Figure 6.891

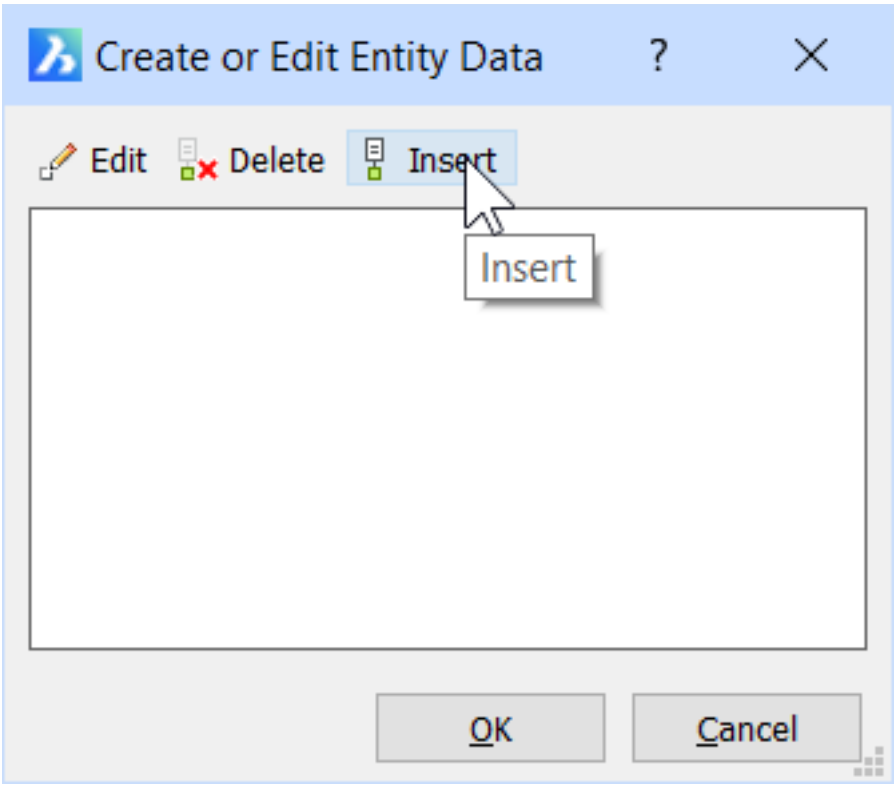
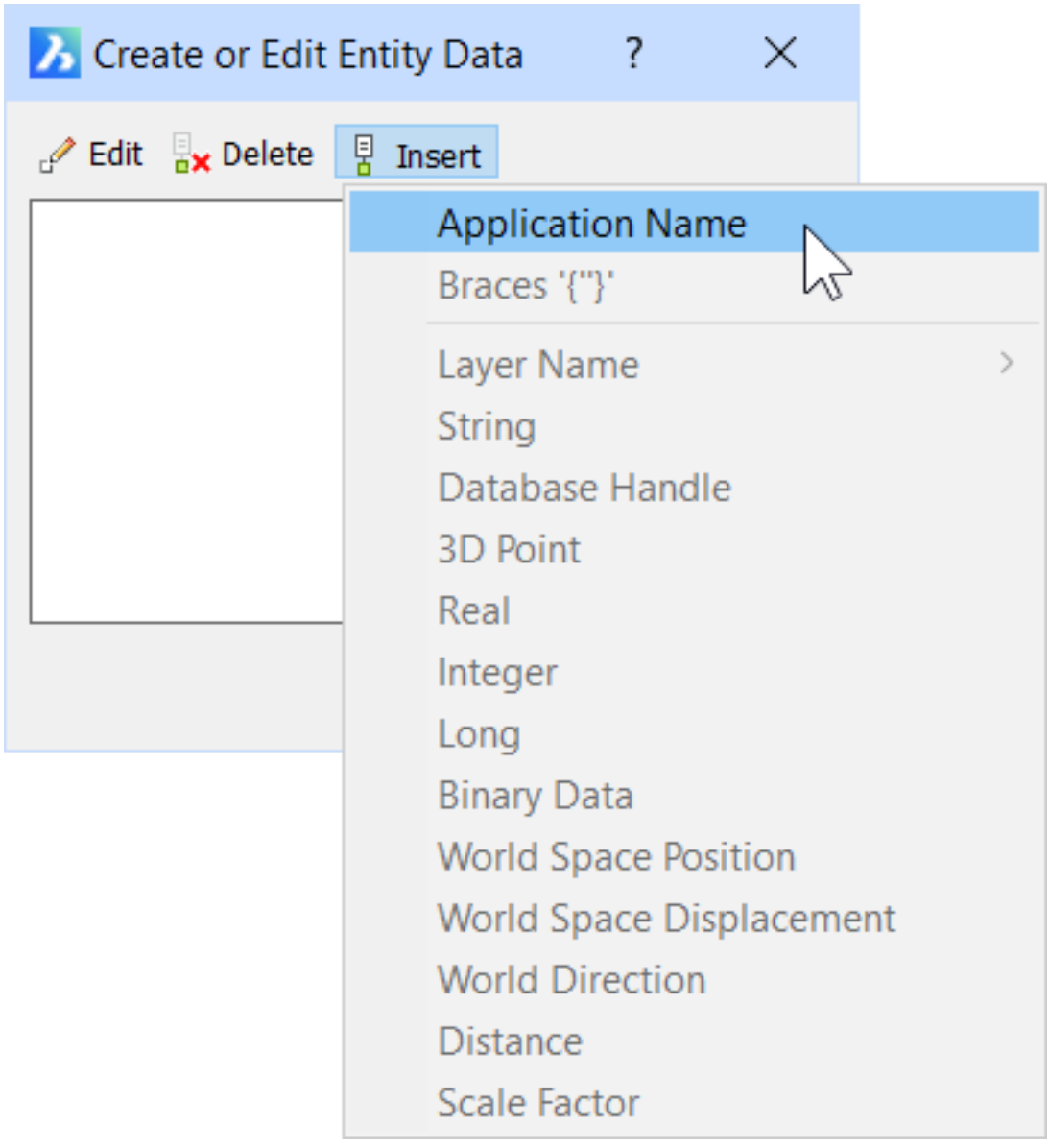

Create or Edit Entity Data
?
Edit
Delete
Insert
Insert
OK
Cancel

Figure 6.892


Create or Edit Entity Data
?
Edit
Delete
Insert
Application Name
Braces '{"}'
Layer Name
String
Database Handle
3D Point
Real
Integer
Long
Binary Data
World Space Position
World Space Displacement
World Direction
Distance
Scale Factor

Command Options

Option	Description
Select entity to create or edit Entity Data	Chooses any entity in the drawing.

Dialog Box Menu Bar	Description
Edit	Edits the selected parameter.
Delete	Erases the selected parameter.
Insert	Inserts extended entity data; displays a menu of data options.

Insert Menu Options	Description
Application Name	Specifies the name of the application; this name identifies the "owner" of the data, and is used by other edata commands, such as DelEData.
Braces	Inserts a pair of braces {} to create subsets of data.
Layer Name	Specifies the layer; choose a layer name from the sub menu.The layer must already exist in the drawing.
String	Inserts a text string.
Database Handle	Inserts the handle (DWG database identifier) of an entity; prompts you: **Select entity** - **(pick any entity in the drawing.)**
3D Point	Inserts a 3D coordinate.
Real	Inserts a real number.
Integer	Inserts an integer (8-bit).
Long	Inserts a long (16-bit) integer.
Binary Data	Inserts binary data; useful for storing images.
World Space Position	Inserts x,y,z coordinates picked in the drawing; prompts you: **Select 3D World Space Position Point** - **(pick a point in the drawing, or enter x,y,z coordinates."World" refers to WCS, the world coordinate space.)**
World Space Displacement	Inserts x,y,z coordinates picked in the drawing; prompts you: **Select 3D World Space Displacement Point** - **(pick a point in the drawing, or enter x,y,z coordinates.)**
World Direction	Inserts x,y,z coordinates picked in the drawing; prompts you: **Select 3D World Direction Point** - **(pick a point in the drawing, or enter x,y,z coordinates.)**
Distance	Inserts x,y,z distance between two points picked in the drawing; prompts you: 1. **Distance** - **(pick a point in the drawing, or enter x,y,z coordinates.)** 2. **Second point** - **(specify another point.)**
Scale Factor	Inserts a scale factor.

Related Commands

CopyEData (6.9.53) - copies extended entity data to other entities. DelEData (6.10.25) - erases extended entity data from entities. MoveEData (6.21.32) - moves extended entity data to other entities. ReAssocApp (6.26.2) - associates applications with entities.

Figure 6.893



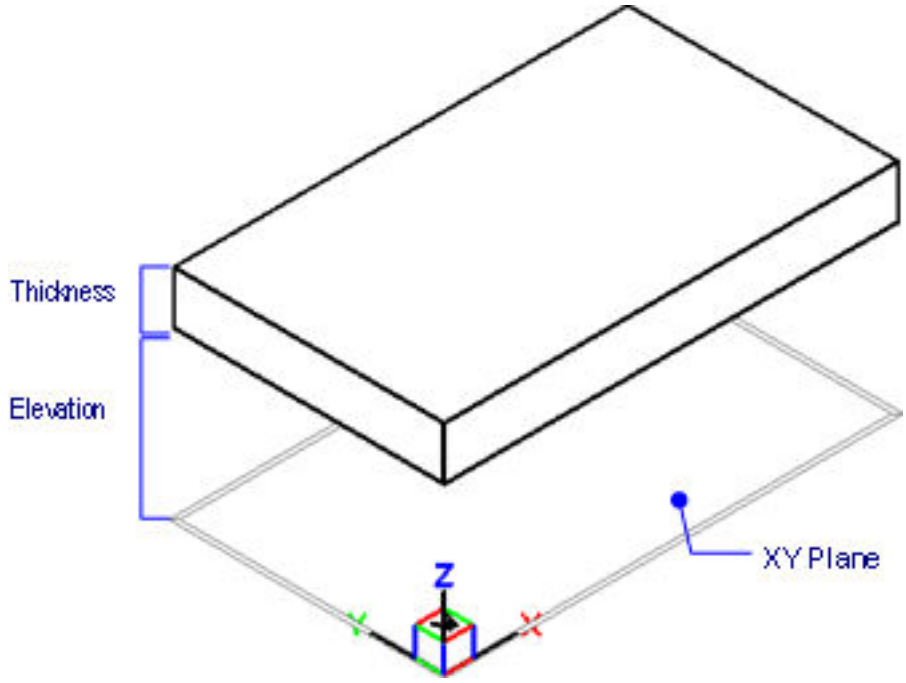
6.13.4 Elev

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Changes the elevation (as measuredfrom the x,y-plane in the z direction) and thickness (as measured from the elevation) of selected entities.

Accessing the Command


command bar: **elev** transparent: **'elev** alias: **elevation** menu bar: **Settings | Elevation** ribbon: **Settings | Elevation** toolbar: **Settings** |  : **Elev Prompts you in the command bar:** New current elevation <0.0000>: **(Enter a new value, or press Enter to keep the current value)** New current thickness <0.0000>: **(Enter a new value, or press Enter to keep the current value)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image145

Figure 6.894



Command Options

Option	Description
New current elevation	Specifies the elevation above or below the x,y-plane; enter a negative value for a negative elevation.The elevation is reported on the status bar as the z coordinate.
New current thickness	Specifies the thickness: the thickness begins at the elevation; enter negative values for thicknesses that are drawn downwards (negative z direction). The thickness is the extrusion distance (in the z direction) of non-3D entities, such as points, lines, polylines, arcs and circles.When thickness is not 0: • Points become vertical lines • Lines and open polylines become vertical planes • Arcs become curved surfaces • Circles become open-ended cylinders • Closed polylines become open-ended tubes

Related Commands

Flatten (6.14.10) - Flattens 2D and 3D entities: projects the entities onto the XY-plane of the current view. Change (6.9.10) - changes the elevation and thickness of entities through the command bar. Properties (6.24.49) - changes the elevation (z coordinate) and elevation through a palette. Settings (6.27.23) - sets the default elevation and thickness through the 3D section of the Settings dialog box.

Figure 6.895



6.13.5 Ellipse

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws ellipses and elliptical arcs.

Accessing the Command



command bar: **ellipse** alias: **el** menu bar: **Draw | Ellipse Draw | Elliptical Arc** ribbon:**Draw | Draw | Ellipses** toolbar: **Draw |**  **Draw |**  : **Ellipse Prompts you in the command bar:** First end of ellipse axis or [Arc/Center]: **(Pick a point, or else enter an option)** Second end of axis: **(Pick another point)** Set Other axis or [Rotation]: **(Pick a third point, or enter R)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image182.png
All methods to draw ellipses can be found in the **Draw | Ellipse** menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
All methods to draw elliptical arcs can be found in the Draw | Elliptical Arc menu: (\vref {fig:C:/Dev/Task-2448/images/staticbric

Figure 6.896

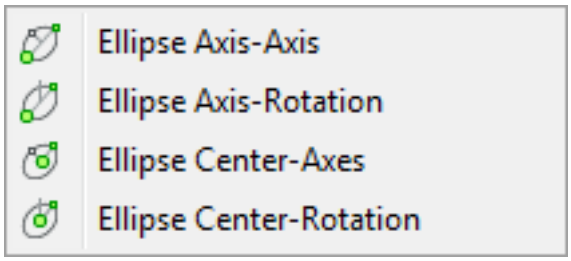
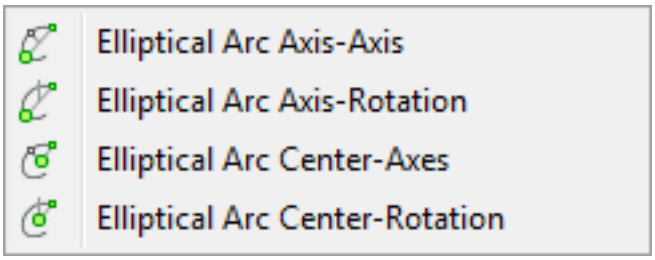


Figure 6.897



Command Options

Option	Description
First End of Ellipse Axis (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image180.png})	Specifies the first endpoint of the first axis.Ellipses are defined by the lengths of two axes, or one axis and an angle of rotation.
Second End of Axis	Specifies the other end of the first axis.
Other Axis	Specifies the endpoint of the second axis, measured from the center of the first axis. The longer of the two axes is called the ”major axis”; the shorter one is called the ”minor axis.”
Arc (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image183.png})	Draws elliptical arcs; prompts you: 1. **Center** - **(specify the center point of the elliptical arc.)** 2. **First end of ellipse axis** - **(specify the endpoint of the elliptical arc's first axis.)**
Parameter	Specifies parameters for the parametric formula that crafts the elliptical arc: p(b) = c + a * cos(u) = b * sin(u) where: a = major axis. b = minor axis. c = center of the ellipse. u = point along the arc. Cos and Sin are the trigonometric cosine and sine functions.
Start Angle of Arc	Specifies the start angle, creating the arc's starting point.The angles are measured counter-clockwise from the positive x axis.
End Angle	Specifies the end angle, creating the arc's endpoint.
Included	Specifies the included angle, measured from the starting point.
Center (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image186.png})	Draws the ellipse starting with a center point; prompts you: **Center of ellipse** - **(pick a point, or enter x, y coordinates.)**
Endpoint of Axis	Specifies the endpoint of one axis, measured from the center point.
Other Axis	Specifies the endpoint of the other axis, also measured from the center point.
Rotation (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image189.png})	Specifies the angle of the ellipse; prompts you: **Rotation around major axis** - **(pick a point, or enter an angle between 0 and 89.9 degrees.An angle of 0 degrees draws a circle; 89.9 draws a very thin ellipse.)**

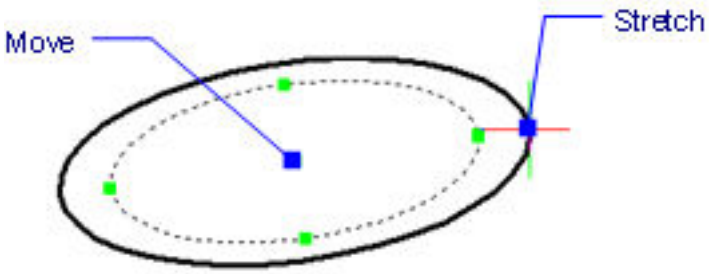
Procedures

- Setting the Polyline Ellipse (PELLIPSE) variable (1.17.17)
- General procedure to draw an elliptical arc (1.17.18)
- Editing an ellipse (1.17.17)
- Editing an elliptical arc (1.17.18)

Grips Editing

Ellipses can be edited directly through grips:

Figure 6.898



Related Commands

Arc (6.4.12) - draws circular arcs. Isoplane (6.17.22) - change the working isoplane for drawing isometric entities, including isocircles, which are made from ellipses Properties (6.24.49) - changes the properties of ellipses through a palette.

Figure 6.899

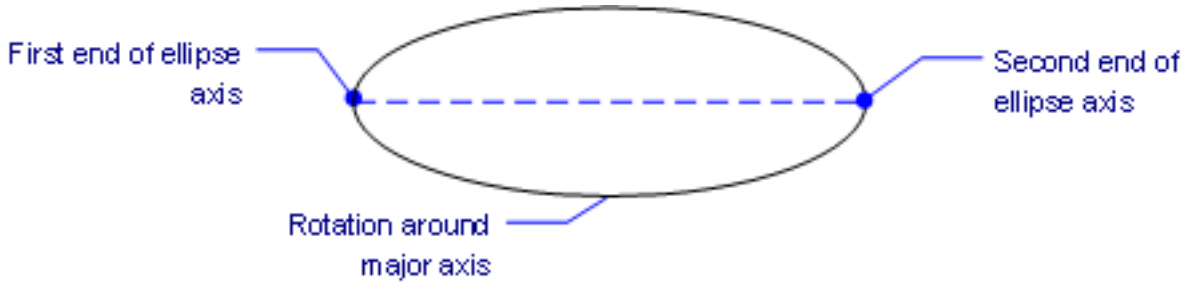


Figure 6.900

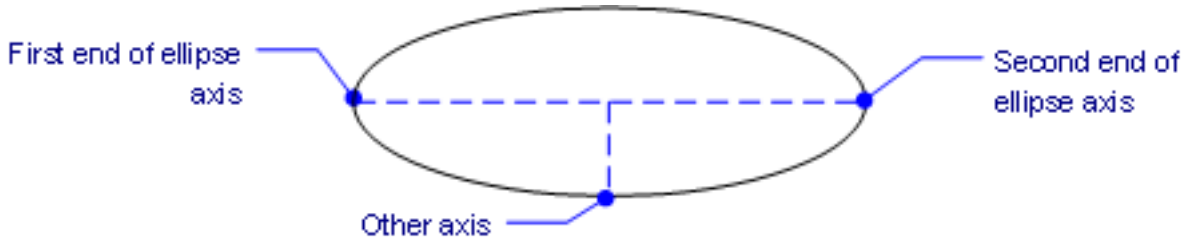


Figure 6.901

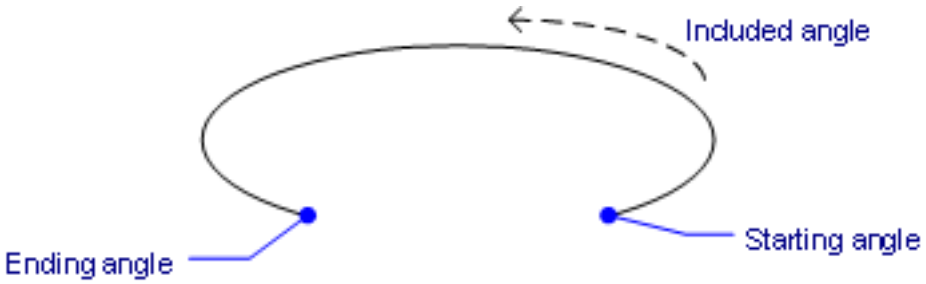


Figure 6.902

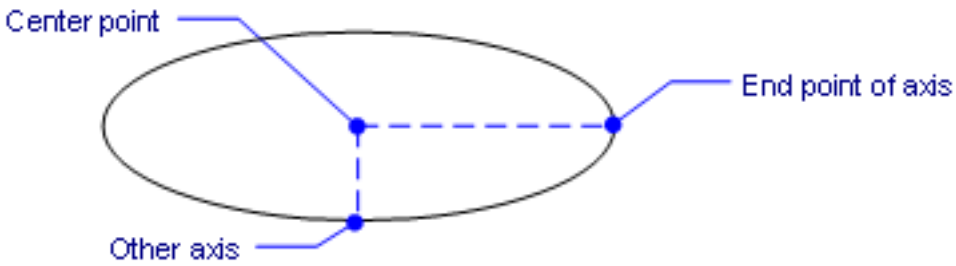


Figure 6.903

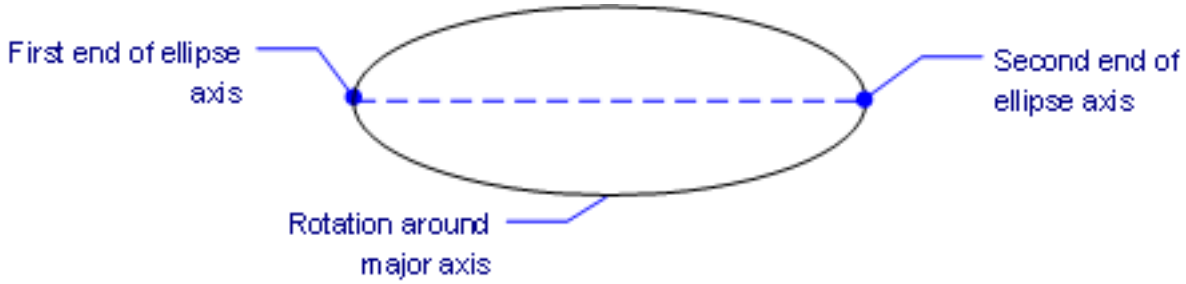


Figure 6.904



6.13.6 EndCompare

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Exits a drawing that was in comparison mode begun by the DwgCompare command.

Accessing the Command


command bar: **endcompare** toolbar: **Compare** |  : **EndCompare** Displays no prompts you in the command bar.

Command Options

No command options.

Related Commands

DwgCompare (6.10.41) - compares a selected drawing with the current drawing during a comparison session. KeepMe (6.19.1) - allows to merge the compared drawings visually.

Figure 6.905



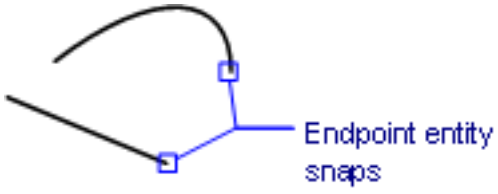
6.13.7 Endpoint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles endpoint entity snap; snaps to the ends of open entities, such as line, arcs, and open polylines and vertices of 3D solids.

Accessing the Command

 command bar: **endpoint** transparent: **end** ribbon:**Home | Entity Snaps | Snap to Endpoint** toolbar: **Entity Snaps |**  : **Endpoint** Displays no prompts in the command bar. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricsc When endpoint entity snap is on, the cursor snaps to the endpoints of open entities, such as lines and arcs. This entity snap also works with the endpoints of edges found on 3D solid models.

Figure 6.906



Command Options

Option	Description
Endpoint	Entering the command toggles Endpoint entity snap. It snaps to the ends of open entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode. Zvertex (6.35.9) - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Figure 6.907



6.13.8 Erase

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Erases entities from the drawing; erases the faces and edges of 3D solids.

Accessing the Command

 command bar: **erase** shortcut: **Del** key on keyboard alias: **delete, e** menu bar: **Edit | Erase** ribbon:**Home | Edit | Erase** toolbar: **Standard |**  Quad: **Model |**  : **Erase Prompts you in the command bar:** Select entities, edges or faces: **(Select one or more entities)** Entities in set: 1 Select entities, edges or faces: **(Select more entities, or else press Enter)**

Command Options

Option	Description
Entities	Selected entities are erased from the drawing.
Edges	Select the edges of an opening in a surface to delete the opening. Select the edge(s) between coplanar faces of a 3D solid to remove the edges. Alternatively use dm-Simplify to remove redundant edges.
Faces	Select the interior faces of an opening in a 3D solid to remove the opening.

Procedures

- Deleting holes (1.18.2)
- To remove edges between coplanar faces (1.18.2)

Related Commands

Break (6.6.18) - erases a portion of an entity. DmSimplify (6.12.27) - simplifies the geometry and topology of 3d solid entities. Oops (6.23.7) - returns erased entities. Overkill (6.23.14) - Deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones. Purge (6.24.57) - removes unused named entities from drawings, such as unused layers and linetypes. Trim (6.29.26) - removes portions of entities. U (6.30.1) - unerases erased entities. Undo (6.30.5) - unerases erased entities.

Figure 6.908



6.13.9 eTransmit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a package consisting of the drawing file and all its dependencies, such as external references, images, font files, plot configuration files, plot style tables, and font map files.

Accessing the Command

 command bar: **etransmit** menu bar:

- **File| eTransmit**
- **Tools | Drawing Explorer | Dependencies**

ribbon:

- **Home | Settings | Drawing Explorer | Dependencies**
- **Home | Transmit | eTransmit**

: **eTransmit** Displays the **Drawing Explorer** dialog box at the **Dependencies** node, listing the dependencies of the current drawing: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/etransmit-dlg.png}) Optionally click the **New File** () button in the toolbar to add another drawing to the eTransmit procedure.

A File dialog displays to let you choose a drawing. Click **eTransmit** () in the toolbar to start the eTransmit procedure. A dialog box lists the files to include: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands
1-dlg.png}) Select the files to be included in the eTransmit package (usually all of them), and then click the **Transmit** button. A dialog box displays options: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/
2-dlg.png}) Adjust the settings, and then click the **Transmit** button. Displays a report: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/eTransmitReport.png})

Figure 6.909

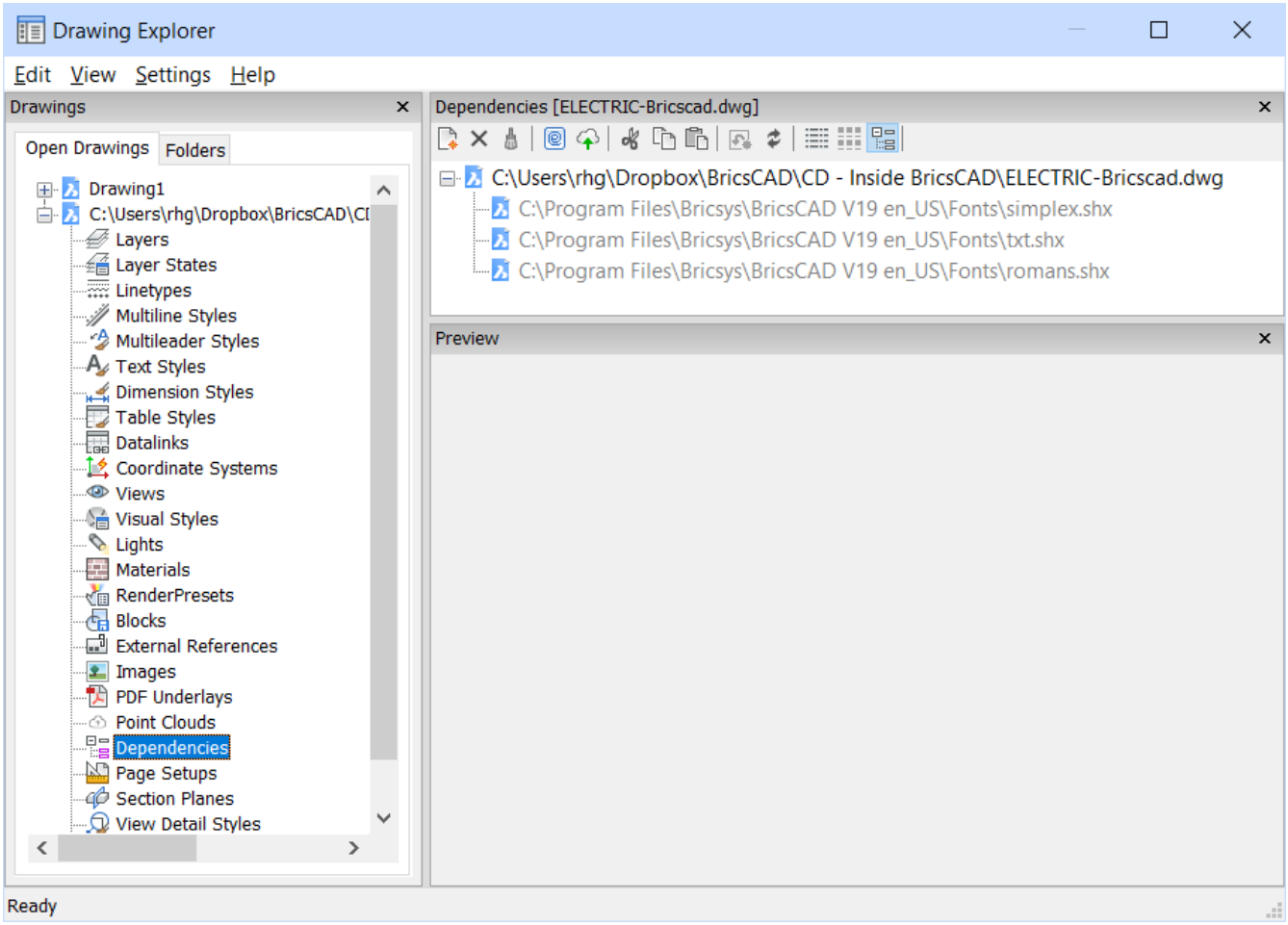


Figure 6.910

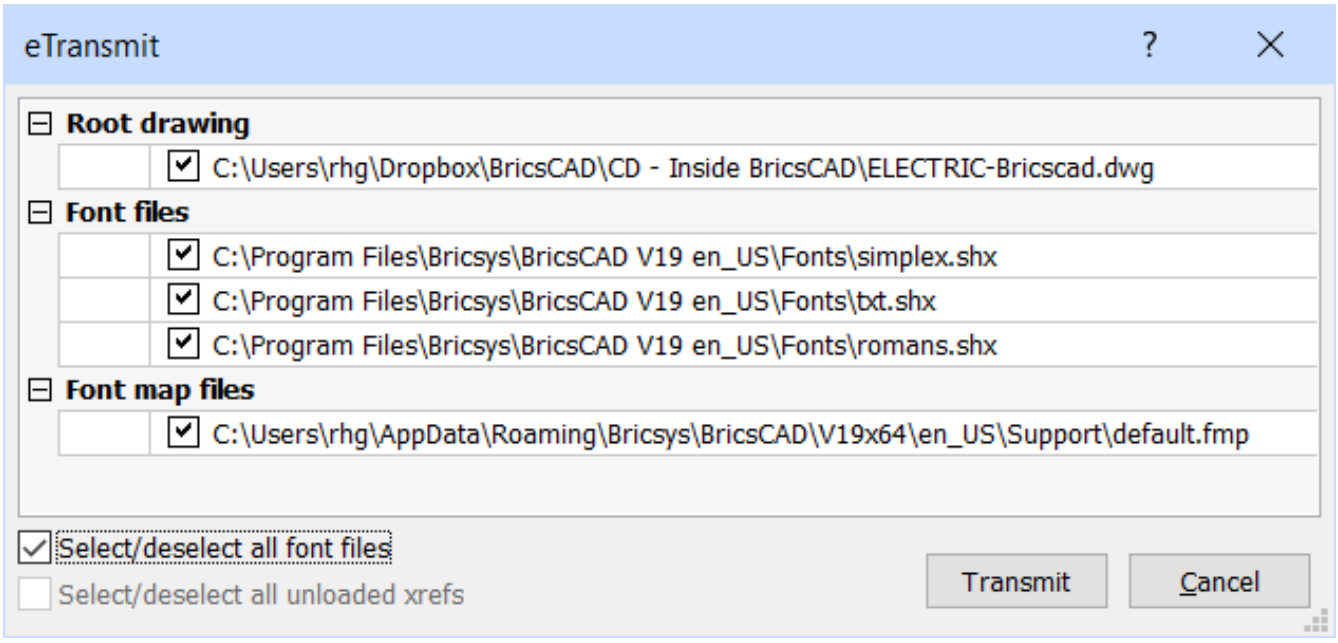


Figure 6.911

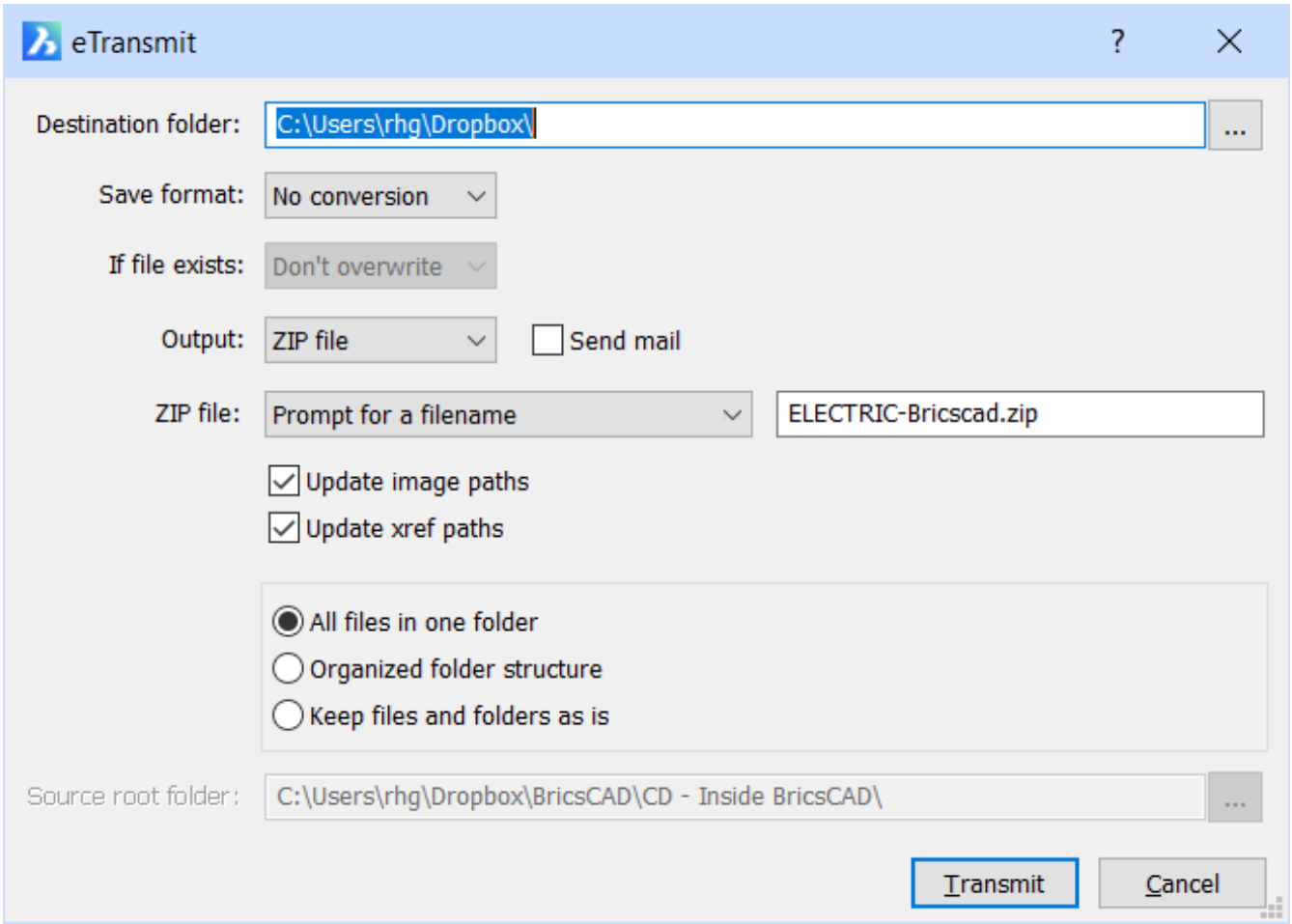
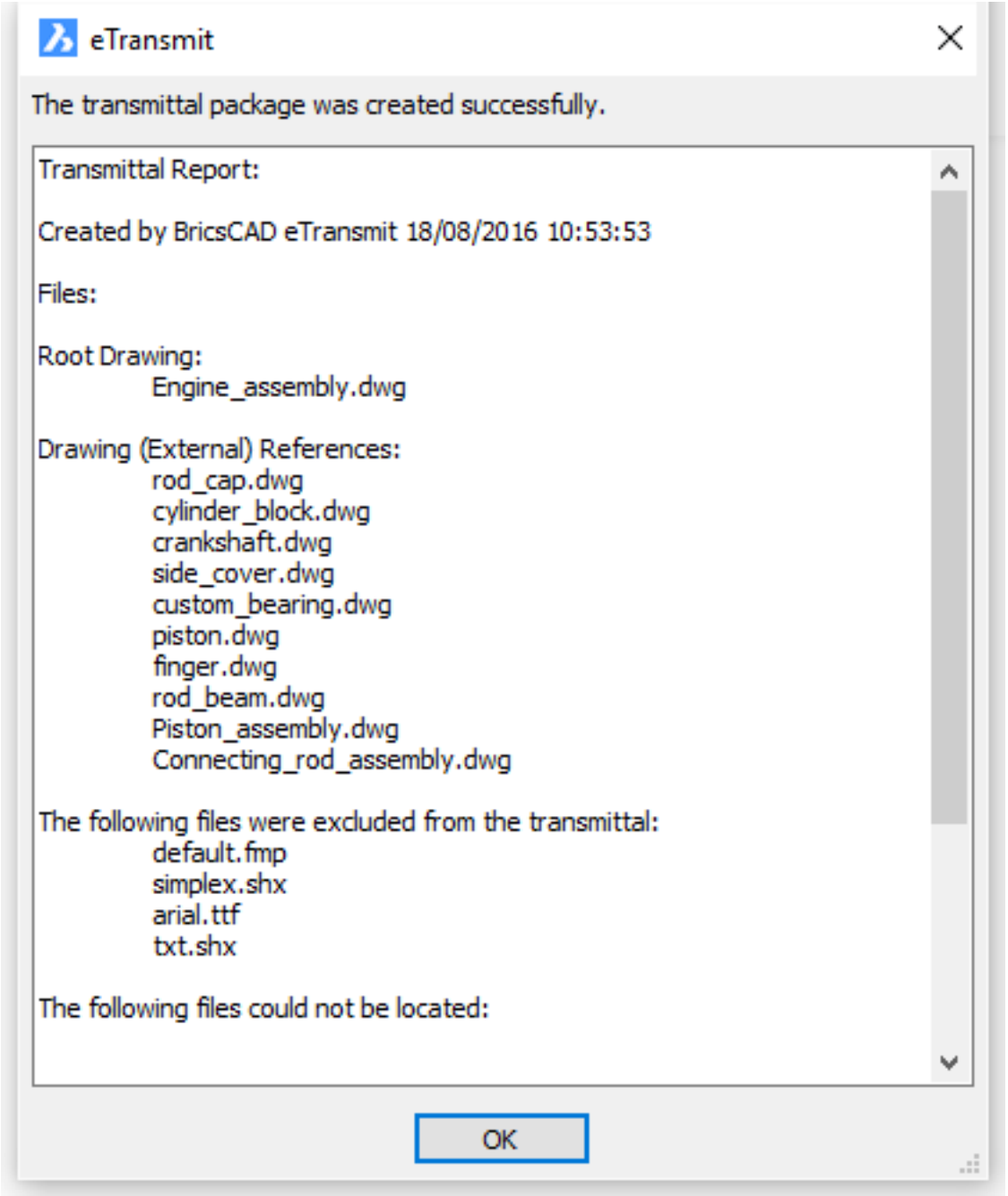

eTransmit
Destination folder:
C:\Users\rhg\Dropbox\
Save format:
No conversion
If file exists:
Don't overwrite
Output:
ZIP file
Send mail
ZIP file:
Prompt for a filename
ELECTRIC-Bricscad.zip
Update image paths
Update xref paths
All files in one folder
Organized folder structure
Keep files and folders as is
Source root folder:
C:\Users\rhg\Dropbox\BricsCAD\CD - Inside BricsCAD\
Transmit
Cancel

Figure 6.912


eTransmit
The transmittal package was created successfully.
Transmittal Report:
Created by BricsCAD eTransmit 18/08/2016 10:53:53
Files:
Root Drawing:
Engine_assembly.dwg
Drawing (External) References:
rod_cap.dwg
cylinder_block.dwg
crankshaft.dwg
side_cover.dwg
custom_bearing.dwg
piston.dwg
finger.dwg
rod_beam.dwg
Piston_assembly.dwg
Connecting_rod_assembly.dwg
The following files were excluded from the transmittal:
default.fmp
simplex.shx
arial.ttf
txt.shx
The following files could not be located:
OK

Tutorial: Package Your Drawings with eTransmit Click here (1.4.12) to watch

eTransmit Command Options

Option	Description
Destination folder	Specifies the output folder for the eTransmit package.If the destination folder does not exist, you are prompted to create the folder.
Save format	Specifies the format of the drawing files in the eTransmit package; choose an option from the droplist: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/eT
If file exists	Specifies what to do if a file already exists in the output folder; choose an option from the droplist: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/eT
Output	Specifies whether to create a ZIP file or copy the files to the output folder; choose an option from the droplist: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/eT
Send email:	When checked, sends an email with the zipped eTransmit package in attachment.This option is available only when the eTransmit output is a ZIP file only.
ZIP file	Specifies what to do when a ZIP file of the same name already exists in the output folder; choose an option from the droplist: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/eT
Update image paths	When checked, the saved paths for image files are updated, according to the new location.
Update xrefs paths	When checked, then the saved paths to externally-referenced files are updated, according to the new location.
All files in one folder	All files in the eTransmit package are copied to the destination folder.
Organized folder structure	A hierarchical folder tree is created in the destination folder
Keep files and folders as is	Keeps the folder structure of all files in the eTransmit package. This procedure allows to easily install the package on another computer using the folder structure of the source system.
Source root folder	Specifies the source root folder for relative paths of drawing-dependent files, such as xrefs or image files.

Procedures

To use Etransmit (1.14.10)

Related Commands

CloudUpload (6.9.32) - uploads the current drawing to Bricsys 24/7. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.913

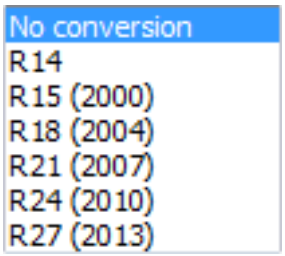


Figure 6.914

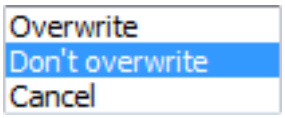


Figure 6.915

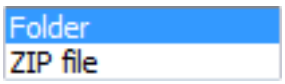


Figure 6.916

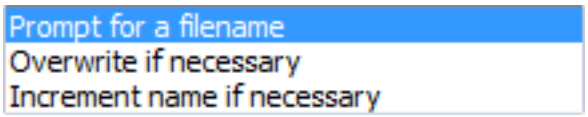


Figure 6.917



6.13.10 -eTransmit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a package ZIP file at the command line, consisting of the current drawing file and all its dependent files.

Accessing the Command

command bar: **-etransmit** : **-eTransmit Prompts you at the command line:** Transmittal package [Create/Settings] <Create>:**(Enter an option)** Enter .zip file name <C:\Program Files\Bricsys\BricsCAD V19 en_US\>: [Yes/No] <Yes>:**(Enter a file name)**

Command Options

Option	Description
Create	Packages the drawing and its support files into a ZIP file; prompts you for the file name: **Enter .zip file name <C:\Program Files\Bricsys\BricsCAD V19 en_US\>: [Yes/No] <Yes>** - **(enter an option)**: • **Yes** - saves the ZIP file in the folder displayed, using the drawing name as the file name • **No** -displays the Create Transmittal File dialog box; choose a folder and enter a file name, and then click **Save** When the drawing is not saved, prompts you: **There are unsaved document changes. Save them so to proceed with -ETRANSMIT? [Yes/No] <Yes>** - **(enter an option)**: • **Yes** - displays the Save As dialog box; choose a folder and enter a file name, and then click **Save** • **No** - command continues to the 'Transmittal package [Create/Settings]' option

Settings	Prompts you for changes to the settings in the command line: **Transmittal settings [Save format/Output format/FOlder structure/FIle list]** - **(enter an option)**: • **Save format** - specifies the format of the drawing files in the eTransmit package; choose an option: **File save version [keep Current/autocad 14 (R14)/autocad LT 95/autocad 2000/autocad 2004/autocad 2007/autocad 2010/autocad 2013/autocad 2018] <keep Current>** - **(enter a version.)** • **Output format** - specifies whether to create a ZIP file or copy the files to the output folder; choose an option: **Transmittal output format [Folder/Zip file] <Zip>** - **(choose to save files in a folder or in a ZIP file.)** • **FOlder structure** - specifies how files are organized into folders: **Transmittal folder structure [all files in ONe folder/Organized folder srtucture/Keep files and folders as is] <One folder>** - **(choose a storage method.)** • **FIle list** - determines which files are induced in the transmittal package: **Transmittal file types [eXternal references/Font files/Plotstyle files/Image files/Material texture files]** - **(choose the file types to include.)**

Procedures

- To use Etransmit (1.14.10)

Related Commands

eTransmit (1.14.10) - packages drawings and support files through a dialog box. CloudUpload (6.9.32) - uploads the current drawing to Bricsys 24/7.

Figure 6.918



6.13.11 ExpBlocks

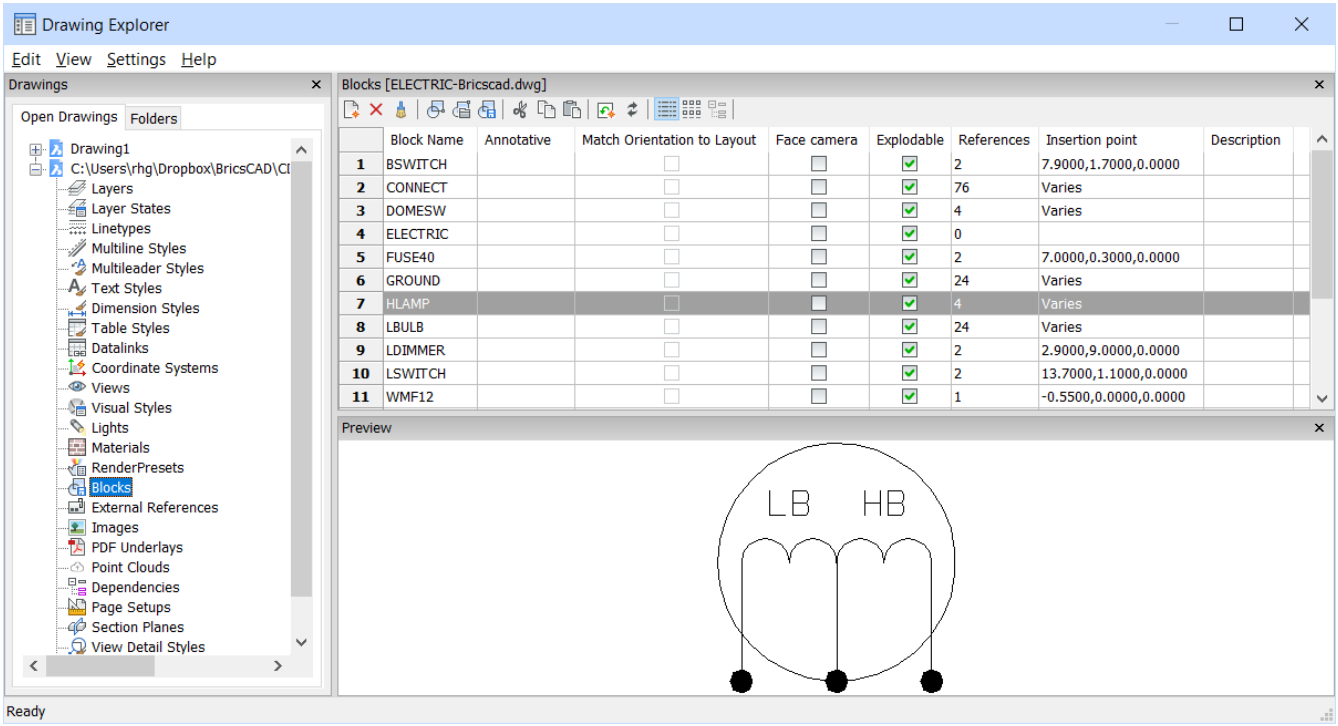
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Blocks section of the Drawing Explorer dialog box (short for "explorer blocks") for inserting, deleting, and creating blocks.

Accessing the Command



command bar: **expblocks** alias: **xb, explorer** menu bar: **Tools | Drawing Explorer | Blocks** ribbon:**Home | Settings | Drawing Explorer | Blocks** : **ExpBlocks** Displays the Blocks section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/expblock-dlg.png}) Insert, create, delete, or edit blocks, and then click **X**. Press F1 to access help.

Figure 6.919



Command Options

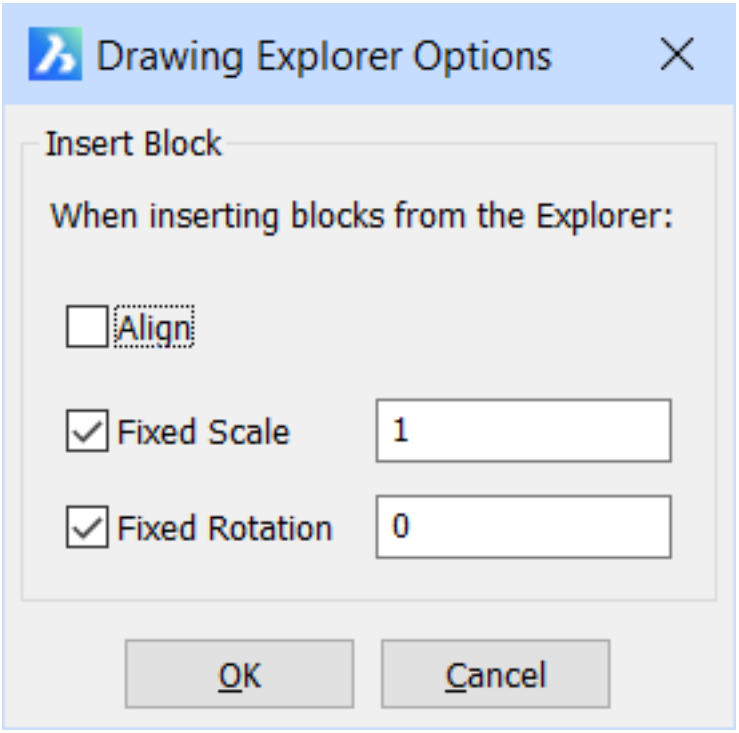
Option	Description
Block Name	Specifies the name of the block; click to rename.
Annotative	When checked, sets the Annotative property.
Match Orientation to Layout	When checked, sets the Match Orientation property of annotative blocks.
Face Camera	When checked, inserts the block to face the camera; currently not implemented. See the Camera (6.9.2) command.
Explodable	When checked, blocks of this definition can be exploded into its component parts by the Explode (6.13.13) command.
References	Reports the number of insertions of the block in the current drawing; number is set by the program, and so you cannot change it.
Insertion Point	Reports the insertion point of the block; reports ”Varies” when the block has been inserted in the drawing more than once.
Description	Optional description of the block.
Toolbar Buttons	
New	Creates new blocks; dismisses the Drawing Explorer dialog box, and the starts the -Block command. See Block (6.6.12) command.
Delete	Deletes the block definition; available only for those block definitions not inserted in the drawing.
Purge	Purges selected blocks from the drawing; available only for those block definitions that are not inserted in the drawing.
Insert	Inserts the selected block; dismisses the Drawing Explorer dialog box, and the starts the -Insert command. See -Insert (6.17.14) command.

Insert External Blocks	Inserts DWG files as blocks; starts the Insert (6.17.13) command, and then displays the Insert Block dialog box. See Insert command.
Save Block	Exports the selected block as a DWG file; displays the Save Block dialog box.

Right-click a block to display the shortcut menu, and then choose **Options**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc
options-dlg.png}) Choose options to preset for inserting blocks, and then click OK.

Options Dialog Box	**Description**
Align	When on, aligns annotative blocks with the current view.
Fixed Scale	When on, specifies the scale factor to be used for all block insertions
Fixed Rotation	When on, specifies the rotation angle to be used for all block insertions

Figure 6.920



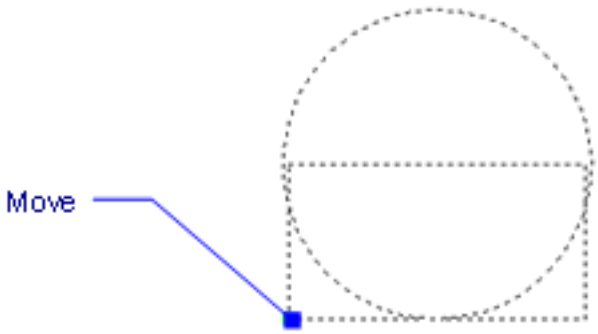
Grips Editing

Blocks can be edited directly through grips:

1. Select the block. Notice that it has one grip located at the insertion point.
2. Drag the grip to move the block.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/image760.png})

Figure 6.921



Procedures

- Open the Blocks Explorer (1.22.8)
- Open the Detail View of the Blocks Explorer (1.22.8)
- Open the Icon View of the Blocks Explorer (1.22.8)
- To rename a block (1.22.8)
- To insert a block from an unopened drawing (1.22.6)

Related Commands

-Block (6.6.12)- creates blocks at the command line. Block (6.6.11)- creates blocks through a dialog box. Blockify (6.6.13) -Automatically detects equally shaped solids in a model and replaces them by block references.Searches the drawing for an identical set of a 2D entities and replaces them by block references. Explorer (6.13.14) - Opens the Drawing Explorer dialog box. Insert (6.17.13) - inserts blocks through a dialog box. -Insert (6.17.14) - inserts blocks through the command bar.

Figure 6.922



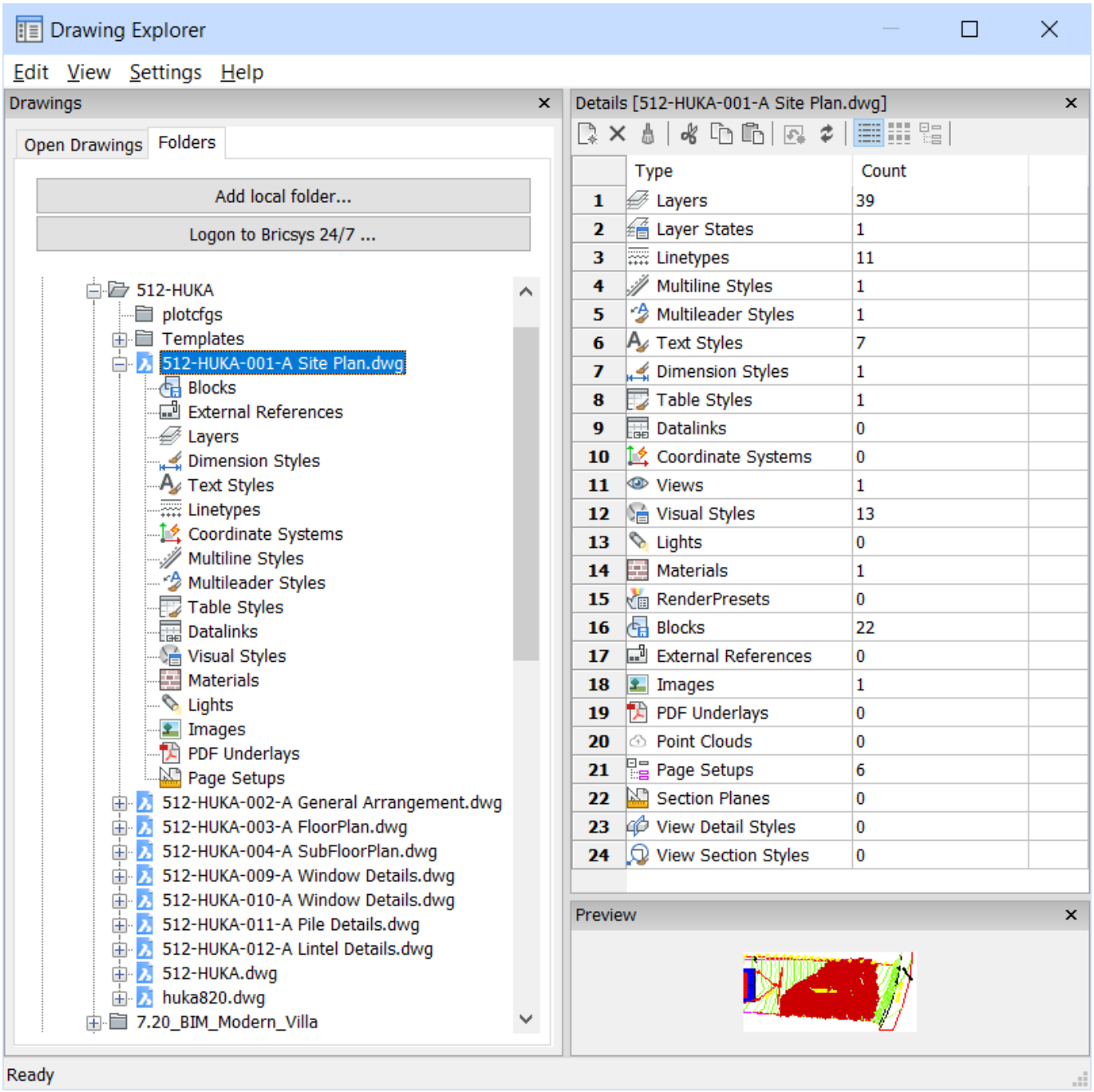
6.13.12 ExpFolders

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Accesses files by opening the Drawing Explorer at the Folders tab.

Accessing the Command

 command bar: **expfolders** menu bar:**Tools | Drawing Explorer | Folders** ribbon:**Home | Settings | Drawing Explorer | Folders** : **ExpFolders** Displays the Drawings Folders tab of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/expfolders.png})

Figure 6.923


Drawing Explorer
Edit View Settings Help
Drawings
Open Drawings
Folders
Add local folder...
Logon to Bricsys 24/7 ...
512-HUKA
plotcfgs
Templates
512-HUKA-001-A Site Plan.dwg
Blocks
External References
Layers
Dimension Styles
Text Styles
Linetypes
Coordinate Systems
Multiline Styles
Multileader Styles
Table Styles
Datalinks
Visual Styles
Materials
Lights
Images
PDF Underlays
Page Setups
512-HUKA-002-A General Arrangement.dwg
512-HUKA-003-A FloorPlan.dwg
512-HUKA-004-A SubFloorPlan.dwg
512-HUKA-009-A Window Details.dwg
512-HUKA-010-A Window Details.dwg
512-HUKA-011-A Pile Details.dwg
512-HUKA-012-A Lintel Details.dwg
512-HUKA.dwg
huka820.dwg
7.20_BIM_Modern_Villa
Details [512-HUKA-001-A Site Plan.dwg]
Type Count
1 Layers 39
2 Layer States 1
3 Linetypes 11
4 Multiline Styles 1
5 Multileader Styles 1
6 Text Styles 7
7 Dimension Styles 1
8 Table Styles 1
9 Datalinks 0
10 Coordinate Systems 0
11 Views 1
12 Visual Styles 13
13 Lights 0
14 Materials 1
15 RenderPresets 0
16 Blocks 22
17 External References 0
18 Images 1
19 PDF Underlays 0
20 Point Clouds 0
21 Page Setups 6
22 Section Planes 0
23 View Detail Styles 0
24 View Section Styles 0
Preview
Ready

Command Options

Option	Description
Add Local Folder	Opens the Choose a Folder dialog box: 1. Navigate to a folder 2. Click **Select Folder**
Logon to Bricsys 24/7	Displays the Bricsys 24/7 dialog box: 1. Click **Log in** 2. Enter the user name and password of a valid account See the CloudOpen (6.9.31) command.

Procedures

- Adding a drawing folder (1.14.2)
- To insert a block from an unopened drawing (1.14.2)

Related Commands

Explorer (6.13.14) - Opens the Drawing Explorer dialog box. CloudOpen (6.9.31) - Logs into and then opens a drawing from 24/7.

Figure 6.924



6.13.13 Explode

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Reduces complex entities to simpler ones, such as blocks into constituent parts, and polylines into lines and arcs.

Accessing the Command

 command bar: **explode** alias: **x** menu bar: **Modify | Explode** ribbon:**Change | Arrange | Explode** toolbar: **Modify |**  **: Explode Prompts you in the command bar:** Select entities to explode: **(Choose one or more entities)** Entities in set: 1 Select entities to explode: **(Choose more entities, or press else Enter to exit command)** The result depends on the entity(ies) you selected:

- Polylines become lines and arcs; width is lost.
- Dimensions becomes lines, text, and solids (arrowheads).
- Blocks become individual entities and nested blocks. (You may need to use Explode a second time.)
- ByBlock colors and linetypes may change.
- Attributes revert to attribute definitions.
- Nonuniformly-scaled blocks are not exploded, unless the EXPLMODE system variable is turned on.
- 3D surface entities become 3D faces.
- Simple entities, such as line and circles, cannot be exploded.

Command Options

Option	Description
Select entities to explode	Choose one or more entities.Entities on frozen and locked layers are not exploded.

Related Commands

Join (6.18.1) - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints. Xplode (6.34.7) - explodes complex entities, with control over the results. TxtExp (6.29.27) - explodes text into polylines. U (6.30.1) - undoes the effect of the explosion.

Figure 6.925



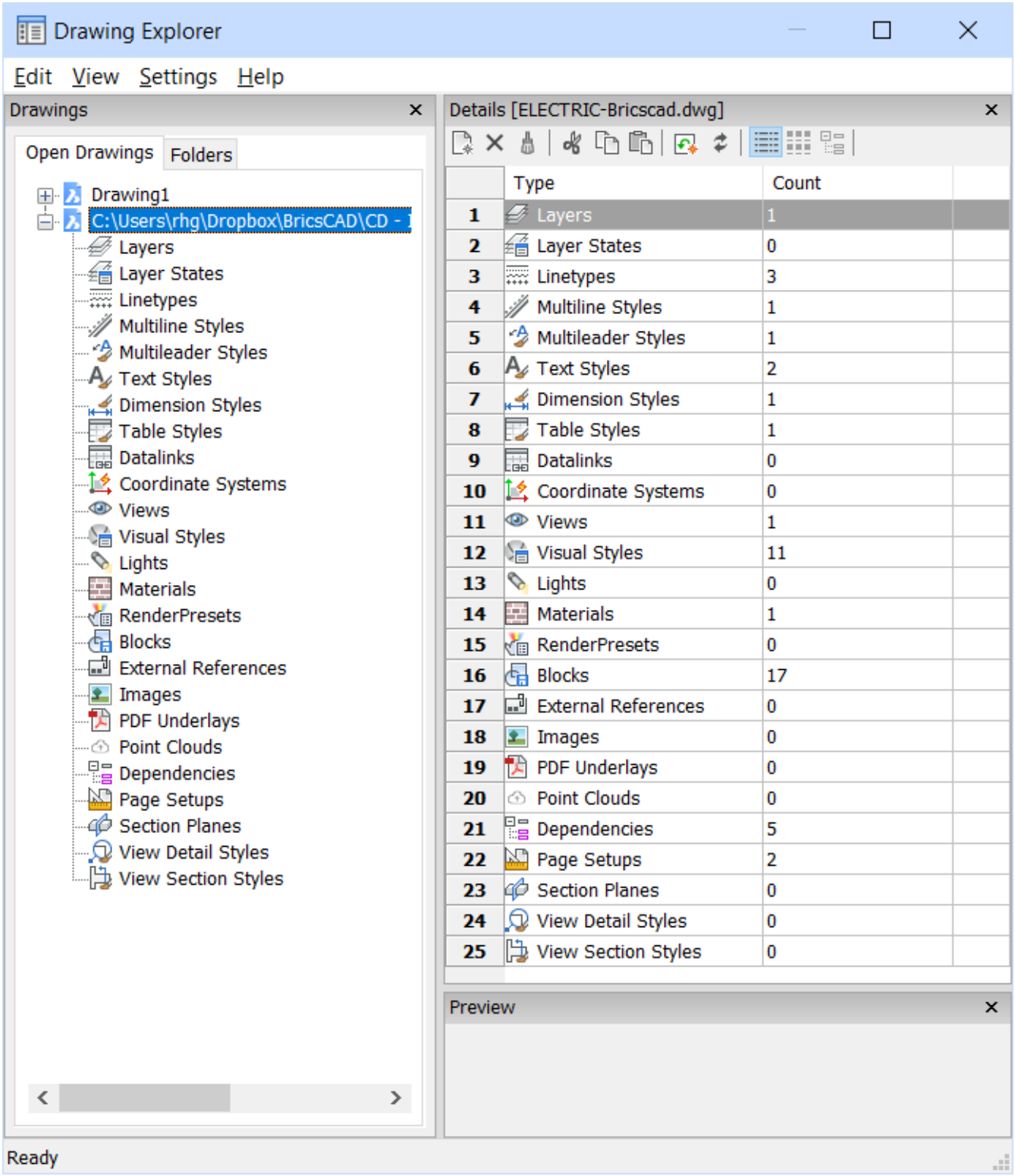
6.13.14 Explorer

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Drawing Explorer dialog box; controls Layers (6.20.2), Layer States (6.20.7), Linetypes (6.20.33), Multiline Styles (6.21.27), Multileader Styles (6.21.25),Text Styles (6.27.53), Dimension Styles (6.11.20), Table Styles (6.29.6), Datalinks (6.10.2),Coordinate Systems (6.13.18), Views (6.31.1), Visual Styles (6.31.13), Lights (6.20.29), Materials (6.21.11), Render Presets (6.26.24), Blocks (6.13.11), External References (6.34.8), Images (6.17.3), PDF Underlays (6.24.12), Point Clouds (6.24.32),Dependencies (6.13.9), Page Setups (6.24.1),Section Planes (6.27.13), View Detail Styles (6.31.5), and View Section Styles (6.31.11).

Accessing the Command


command bar: **explorer** keyboard shortcut: **Ctrl+2** menu bar: **Tools | Drawing Explorer |** (select a category) ribbon:**Home | Settings | Drawing Explorer** toolbar: **Standard** : **Explorer** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/explorer-dlg.png}) Press **F1** to access help.Click **X** to close.

Figure 6.926


Drawing Explorer
Edit View Settings Help
Drawings
Open Drawings
Folders
Drawing1
C:\Users\rhg\Dropbox\BricsCAD\CD - 1
Layers
Layer States
Linetypes
Multiline Styles
Multileader Styles
Text Styles
Dimension Styles
Table Styles
Datalinks
Coordinate Systems
Views
Visual Styles
Lights
Materials
RenderPresets
Blocks
External References
Images
PDF Underlays
Point Clouds
Dependencies
Page Setups
Section Planes
View Detail Styles
View Section Styles
Details [ELECTRIC-Bricscad.dwg]
Type
Count
1 Layers 1
2 Layer States 0
3 Linetypes 3
4 Multiline Styles 1
5 Multileader Styles 1
6 Text Styles 2
7 Dimension Styles 1
8 Table Styles 1
9 Datalinks 0
10 Coordinate Systems 0
11 Views 1
12 Visual Styles 11
13 Lights 0
14 Materials 1
15 RenderPresets 0
16 Blocks 17
17 External References 0
18 Images 0
19 PDF Underlays 0
20 Point Clouds 0
21 Dependencies 5
22 Page Setups 2
23 Section Planes 0
24 View Detail Styles 0
25 View Section Styles 0
Preview
Ready

Command Options

Columns can be switched on and off and repositioned by drag-and-drop:

- **To move a column** - drag it by its header to a new location
- **To change the width of a column** - drag the separator line back and forth
- **To restore a column's width** - double-click the separator line
- **To turn columns on and off** - select a column, then right click:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/DrawingExplorerGridColumnsContext.

- **Check box** - displays or hides the column
- **Show all columns**- displays all columns, including ones that were previously hidden
- **Restore column positions**- returns all columns to their default locutions

Notes: The shortcut keystrokes shown below operate inside the Drawing Explorer dialog box. The options in menus and the buttons displayed by the toolbar vary depending on the current item. Shown below are the actions for layers.

Menu Bar: Edit	Keyboard Shortcut	Description
New 	Ctrl+N	Creates a new item with a generic name, such as NewLayer 1 or New-Style1. Use the Rename button to rename it.
Delete 	Ctrl+D	Erases the selected item from the drawing. When the item is in use, such as a layer with entities, you have to option to retain the entities from the layer to be erased: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 deletelayer-dlg.png}) • **Delete** - removes the layer and the entities assigned to it • **Change** - move the entities to another layer, and then erases the layer; displays a list from which to choose one of all the other layers in the drawing • **Cancel** - cancels the operation so that no changes are made Some items cannot be erased, such as layer 0 and visual style 2D Wireframe.

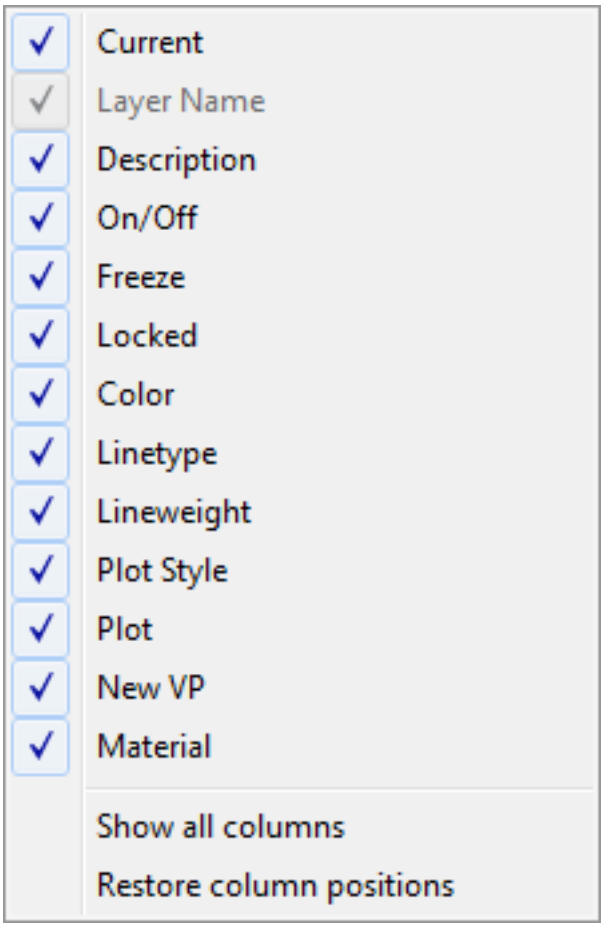
Merge To		(Layers only.) Merges the content of one layer with another ones: 1. Choose a source layer 2. Click **Edit** | **Merge to** 3. Choose a target (destination) from the dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 mergelayers-dlg.png}) • **Choose a target layer** - choose a layer name from the ones listed • **Merge** - move the entities from the source layer to the target layer • **Cancel** - cancels the merging process The source layer is removed from the drawing.
Purge	Ctrl+P	Purges unused items from the drawing. Some items cannot be erased, such as layer 0 and text style Standard. See Purge (6.24.57) command.
Cut	Ctrl+X	Copies the item to the Clipboard, and then deletes it from the drawing following the same procedure as the Delete button.
Copy	Ctrl+C	Copies the item to the Clipboard for pasting elsewhere in the Drawing Explorer.
Paste	Ctrl+V	Copies the item from the Clipboard, and pastes it into the Drawing Explorer.When the item on which the Clipboard content will be pasted is in use, such as a layer with entities, you have options: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 pasteitem-dlg.png}) • **Copy and Replace** - replaces the existing item with the pasted one • **Don't Copy** - cancels the copy operation • **Copy, but keep both** - pastes the Clipboard content with an incremented name, usually suffixed by a 1 • **Cancel** - also cancels the operation

Rename	F2	Renames the selected item; alternatively, click on the item's name twice, slowly, to rename it.
Select All	Ctrl+A	Selects all items.
Invert Selection		Inverts the selected item(s). For instance, if you have one layer selected, then this option unselects it, and then selects all other layers,
Remove Viewport Overrides		(Layers only) Removes overrides from the viewport properties of selected layers.
Isolate Selected Layers		Hides from view all layers that are not selected. Use the Invert Selection option to reverse which layers are visible.
Current		Makes the item current, such as making the selected layer the current (working) layer.

Menu Bar: View	**Description**
Open Drawings	When checked, displays the Drawings pane in the left of the Drawing Explorer dialog box, in which you can select other drawings and items.
Details	When checked, displays the Details panein the center of the Drawing Explorer dialog box, in which you view all related items and possibly edit them.
Preview	When checked, displays the Preview pane atbottom of the Drawing Explorer dialog box, in which the selected item can be viewed.
Editor	When checked, displays the Edit pane in the right of the Drawing Explorer dialog box, in which the properties of items can be changed.
Status Bar	Toggles the display of the status bar at the bottom of the Drawing Explorer dialog box.
Filters	(Layers only.) When checked, displays the Filters pane for layers.
Details	Displays the details of items.
Icons	Displays items as an array of icons; applies only to blocks and external references.
Tree	Displays items in tree form; applies only to layer states, data links, dimension styles, external references, images, PDF underlays and dependencies.
Regen	Updates the drawing display to reflect changes made in this dialog box.
Toggle Regen	Toggles automatic updating of the display.

Menu: Settings	**Description**
Restore Explorer Default Layout	Restores the layout of the Drawing Explorer to the default layout for the selected category. When you makes changes to the layout of the Drawing Explorer, the program remembers them. This option resets all your changes.
Options	(Applies to Blocks and External References only.) Specifies default settings for inserting blocks. See the ExpBlock (6.13.11) command.

Figure 6.927



Drawings Pane	**Description**
Open Drawings	Displays a list of drawings that are currently open.Click a drawing to see a detailed list of its content in the Details pane.
Folders: Add Local Folders	Displays a list of drawing folders that you tend to access often: 1. Click the **Add Local Folder** button to add a folder, including all subfolders. 2. Double click a drawing to open it. 3. To copy items between drawings, drag definitions from one drawing to another; this lets you copy layers, text styles, linetypes, and so one to another drawing.
Folders: Login to Bricsys 24/7	Displays a list of drawings that you have stored online on the Bricsys 24/7 site: 1. Click the **Logon to Bricsys 24/7** button. 2. Enter the user name and password of a valid account: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/login-2-dlg.png}) 3. Select a drawing in a project, and then right-click to download or view the drawing. See the CloudOpen (6.9.31) command.

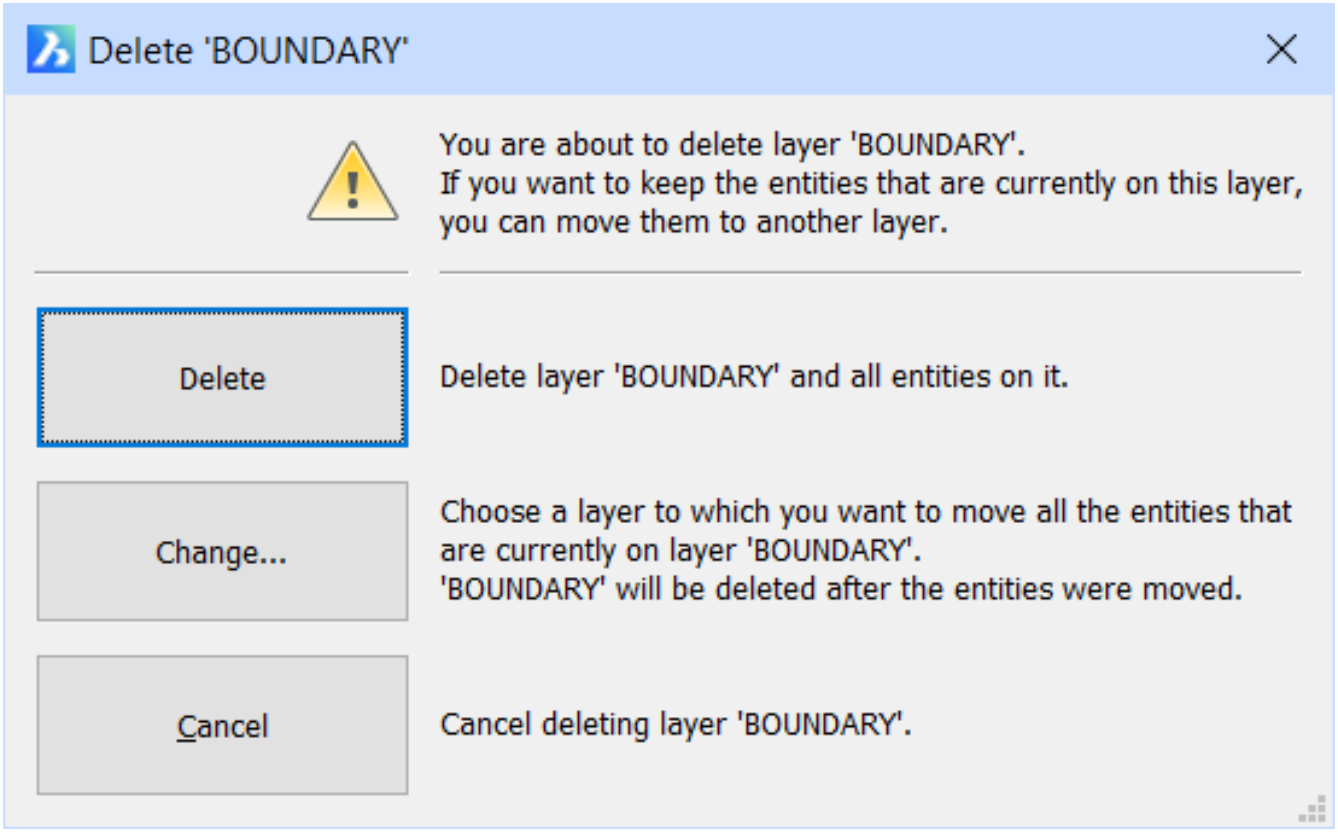
Details	Description
Layers	Reports on the layers defined in the selected drawing.Click to explore the layers; see the Layer (6.20.2) command.
Layer States	Reports on the layer states defined in the selected drawing.Click to explore the layer states; see the LayerState (6.20.7) command.
Linetypes	Reports on the linetypes loaded into the selected drawing.Click to explore the linetypes; see the Linetype (6.20.33) command.
Multiline Styles	Reports on the multiline stylesdefined in the selected drawing.Click to explore multiline styles; see the MlStyle (6.21.27) command.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/ExpMleaderStyles.png})	Reports on the multileader styles defined in the drawing.Click to explore multileader styles; see the MLeaderStyle (6.21.25) command.
Text Styles	Reports on the text stylesdefined in the selected drawing.Click to explore text styles; see the Style (6.27.53) command.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/ExpDimStyles.png})	Reports on the dimension styles defined in the drawing.Click to explore dimension styles; see the DimStyle (6.11.20) command.
Table Styles	Reports on the table styles defined in the selected drawing.Click to explore table styles; see the TableStyle (6.29.6) command.
Datalinks	Reports on the data in the drawing linked to external spreadsheet files.Click to explore data links; see the DataLink (6.10.2) command.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/ExpUcs.png})	Reports on the nameduser coordinate systems (UCS) defined in the selected drawing.Click to explore coordinate systems; see the ExpUcs (6.13.18) command.
Views	Reports on the named views in the selected drawing.Click to explore saved views; see the View (6.31.1) command.
Visual Styles	Reports on the visual styles defined in the selected drawing.Click to explore visual styles; see the VisualStyles (6.31.13) command.
Lights	Reports on the lights placed in the selected drawing.Click to explore lights; see the LightList (6.20.29) command.
Materials	Reports on the materials defined in the selected drawing.Click to explore materials; see the Materials (6.21.11) command.
RenderPresets	Reports on the render presets defined in the drawing.Click to explore render presets; see the RenderPresets (6.26.24) command.
Blocks	Reports onblocks defined in the selected drawing.Click to explore blocks; see the ExpBlocks (6.13.11) command.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/EtypeExternalReferences.png})	Reports on the external references (xrefs) attached to the selected drawing.Click to explore xrefs; see the Xref (6.34.8) command.
Images	Reports on the images attached to the selected drawing.Click to explore the images; see the Image (6.17.3) command.
PDF Underlays	Reports on the PDFunderlays attached to the selected drawing.Click to explore PDF underlays; see the Pdf (6.24.12) command.
Point Clouds	Reports on the point clouds attached to the selected drawing.Click to explore point cloud; see the PointCloud (6.24.32)command.
Dependencies	Reports on the external files that the selected drawing depends on.Click to explore dependencies; see the eTransmit (6.13.9) command.
Page Setups	Reports on the page setups attached to the selected drawing.Click to explore page setups; see the PageSetup (6.24.1) command.
Section Planes	Reports on the section planes defined in the selected drawing.Click to explore the section planes; see the SectionPlaneSettings (6.27.13) command.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/EtypeDrawingDetailStyles.png})	Reports on the view detail styles defined in the selected drawing.Click to explore the view detail styles; see the ViewDetailStyle (6.31.5) command.
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/EtypeDrawingSectionStyles.png})	Reports on the view section styles defined in the selected drawing.Click to explore the view section styles; see the ViewSectionStyle (6.31.11) command.

Procedures

- To open the Drawing Explorer (1.14.2)
- To control the Drawing Explorer layout (1.14.2)
- Opening a drawing (1.14.2)
- Adding a drawing folder (1.14.2)
- To insert a block from an unopened drawing (1.14.2)
- To copy definitions between drawings (1.14.2)
- To merge the content of two layers (1.14.2)

Figure 6.928



Related Commands

DimStyle (6.11.20) - creates and modifies dimension styles, through the Drawing Explorer. eTransmit (6.13.9) - creates a package of a drawing file and all its dependencies, such as: external references, images, font files, plot configuration files, plot style tables and font map files. ExpBlocks (6.13.11) - opens the Blocks section of the Drawing Explorer dialog box. ExpFolders (6.13.12) - opens the Drawing Explorer on the Folders tab. ExpUcs (6.13.18) - creates, modifies, and deletes named UCSs through the Drawing Explorer. Image (6.17.3) - inserts images in drawings through the Drawing Explorer. Linetype (6.20.33) - Loads, sets, and creates linetypes through the Drawing Explorer. Layer (6.20.2) - creates layers, changes their properties, and removes layers through the Drawing Explorer. LayerState (6.20.7) - creates and controls layer states in the Drawing Explorer. LightList (6.20.29) - displays the Drawing Explorer - Lights dialog window. Materials (6.21.11) - creates materials and edits their properties through the Drawing Explorer. MLeaderStyle (6.21.25) - Creates and manages multileader styles through the Drawing Explorer. MlStyle (6.21.27) - creates and edits multiline styles. PageSetup (6.24.1) - creates and edits page setups for plotting drawings in the Drawing Explorer. Pdf (6.24.12) - attaches PDF files to the current drawing through the Drawing Explorer. RenderPresets (6.26.24) - allows to create and edit render presets, and to set the current render preset through a dialog box. SectionPlaneSettings (6.27.13) - Defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog. SheetSet (6.27.31) - Manages sheet sets. Style (6.27.53) - creates and modifies text style with the Drawing Explorer. TableStyle (6.29.6) - creates and edits table styles through the Drawing Explorer - Table Styles dialog box. View (6.31.1) - creates and sets named views through the Drawing Explorer. VisualStyles (6.31.13) - creates and controls visual style definitions in the Drawing Explorer. Xref (6.34.8) - attaches DWG files to the current drawing through the Drawing Explorer.

Figure 6.929

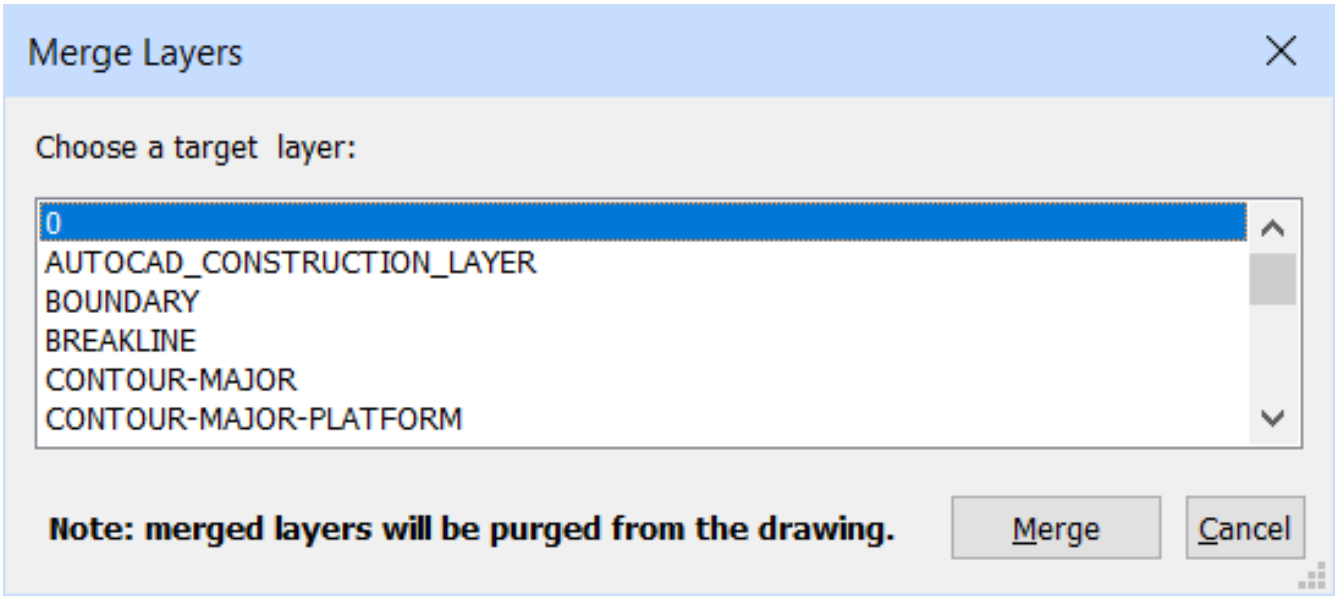


Figure 6.930

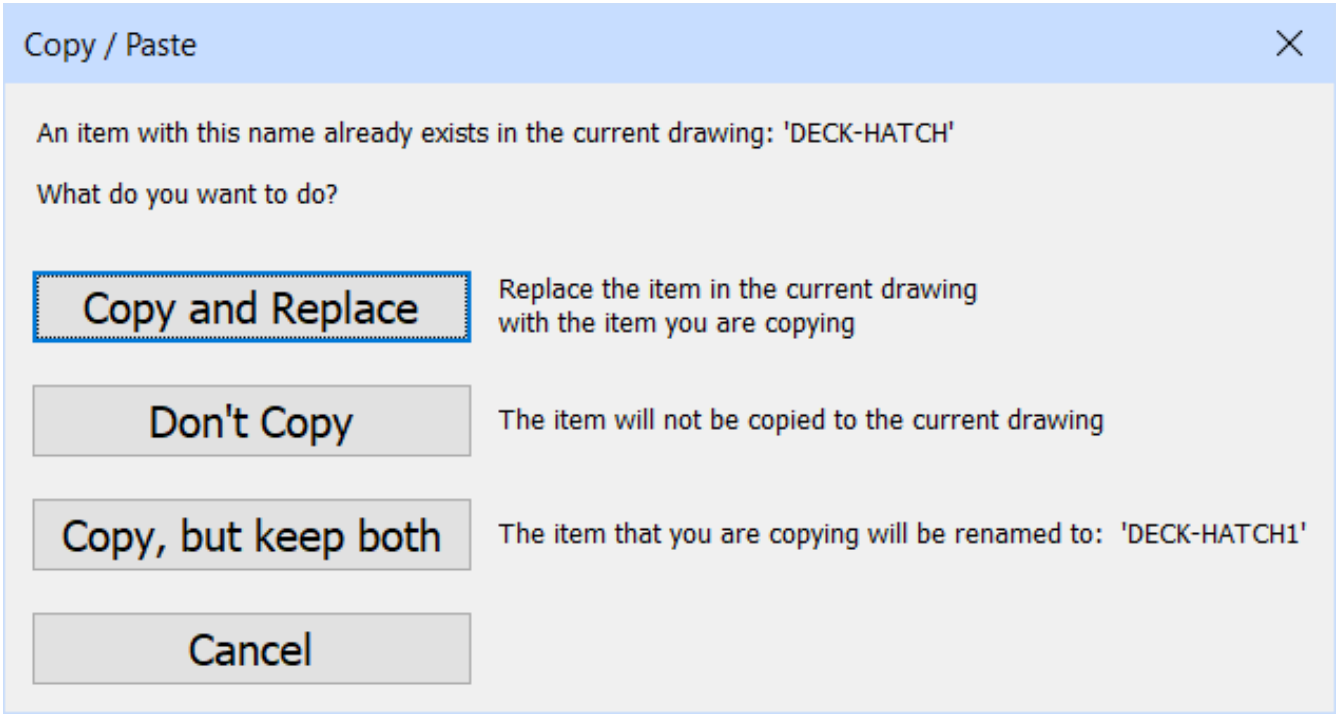


Figure 6.931

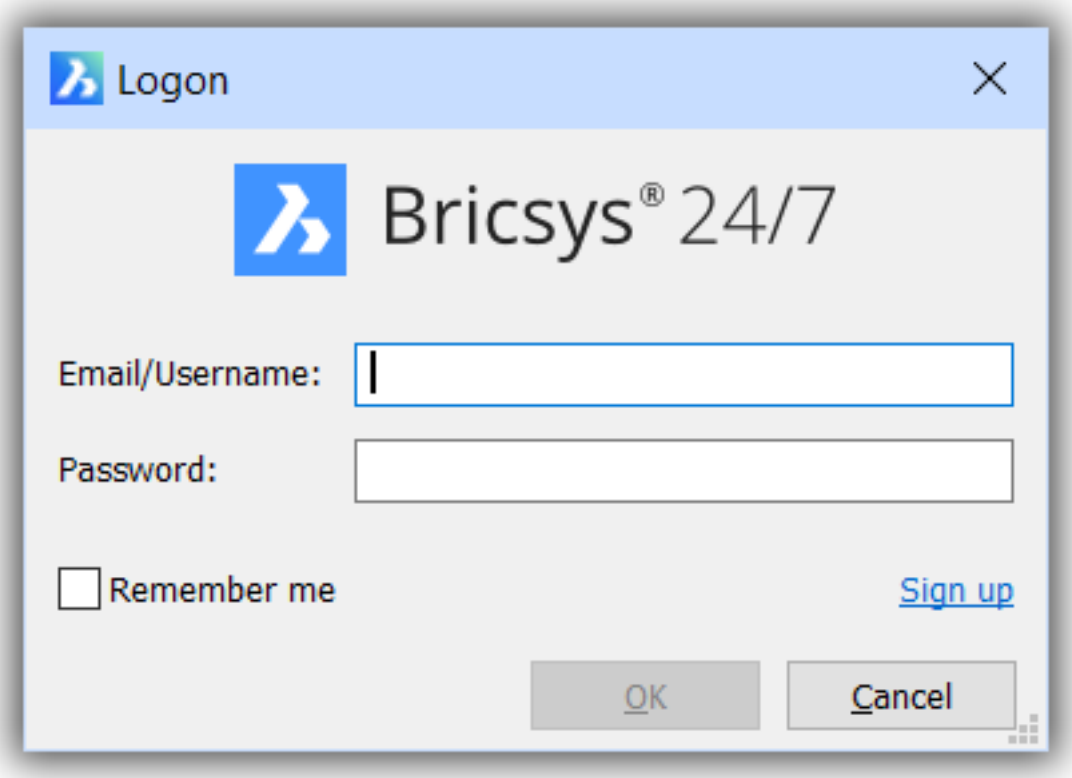


Figure 6.932

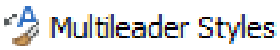


Figure 6.933

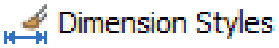


Figure 6.934

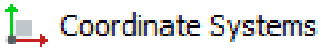


Figure 6.935

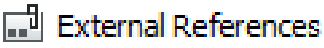


Figure 6.936

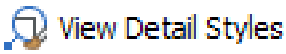


Figure 6.937

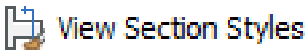


Figure 6.938



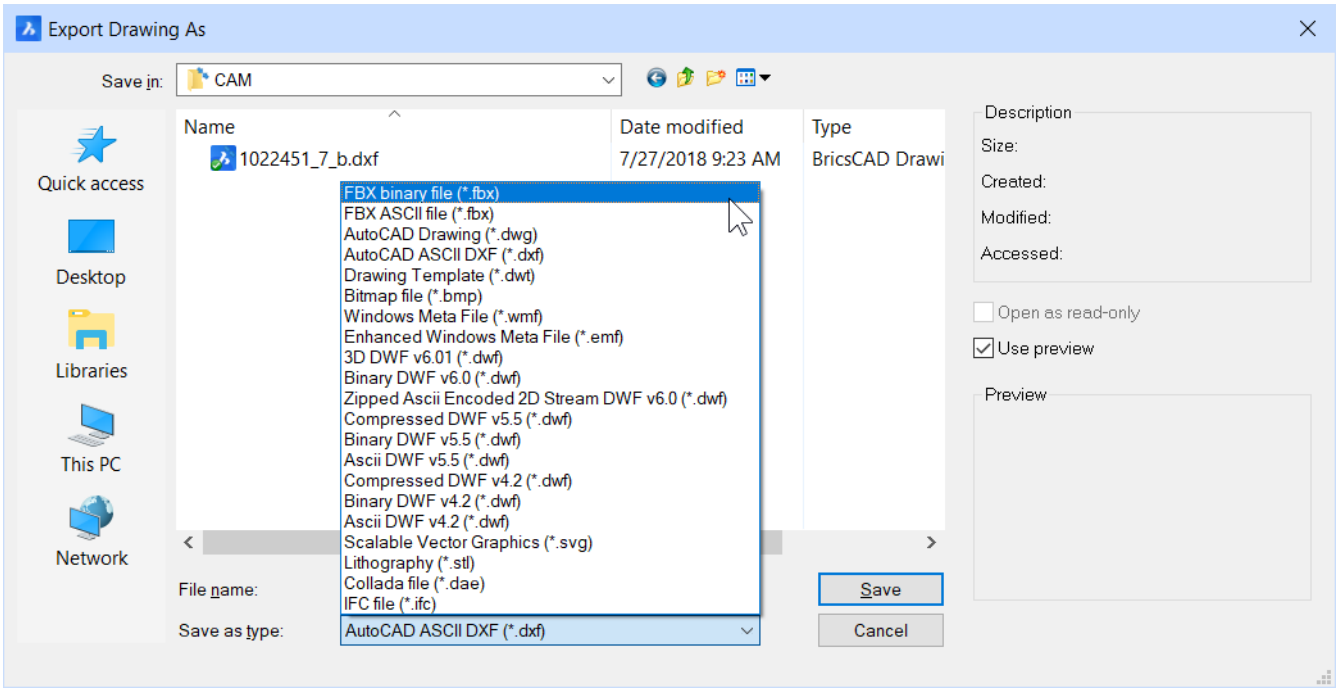
6.13.15 Export

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Exports drawings in a variety of vector and raster formats; exporting drawings to WMF and EMF formats is not supported on Linux.

Accessing the Command

 command bar: **export** alias: **exp, dwfout** menu bar: **File | Export** ribbon:**Home | File | Export** : **Export** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/export-dlg.png}) Choose an export format from the Save As Type droplist, and then click **Save**. To export in PDF format, use the ExportPDF (6.13.17) command.

Figure 6.939



Command Options

Formats are listed in the order in which they appear in the Save as Type droplist, segregated into CAD, vector, and raster formats.

CAD Formats	Extension	Description
FBX binary file	FBX	Mudbox format used to send 3D models to animation and rendering software; binary format creates compact files maintained by Autodesk; see FBXout (6.14.2) command.
FBX ASCII file	FBX	Mudbox format used to send 3D models to animation and rendering software; ASCII format creates human-readalbe files.
AutoCAD Drawing	DWG	CAD drawing format created by Autodesk; to export drawings in earlier version of DWG, use the SaveAs (6.27.3) command.
AutoCAD ASCII DXF	DXF	Drawing interchange format created by Autodesk; to create binary DXF files, use the DxfOut (6.10.44) command.
Drawing Template	DWT	Templates on which new drawings are based.
3D DWF v6.01	DWF	CAD viewing format for 3D models created by Autodesk.
Binary DWF	DWF	Compressed version of DWF; creates more compact files.
Zipped Ascii Encoded 2D Stream DWF v6.0	DWF	ASCII version of DWF compressed into a ZIP file.
Compressed DWF v5.5	DWF	Compressed ASCII version of an older version of DWF; creates more compact files.
Binary DWF v5.5	DWF	Compressed version of an older version of DWF; creates more compact files.

ASCII DWF v5.5	DWF	ASCII version of an older version of DWF>
IFC file	IFC	Saves a BIM model as an IFC file for use with an IFC-certified applications.data.BIM properties (6.7.41) are included in IFC export. The Industry Foundation Classes (IFC) data model is intended to describe building and construction industry
Vector Formats	**Extension**	**Description**
Windows Meta File*	WMF	Vector format created by Microsoft; used by the Clipboard; see the WmfOut command.
Enhanced Windows Meta File*	EMF	Updated version of WMF.
Scalable Vector Graphics	SVG	Used by Web pages to display vector graphics. Scalable Vector Graphics (SVG) is a family of specifications of an XML-based file format for describing two-dimensional vector graphics, both static and dynamic (i.e. interactive or animated).The SVG specification is an open standard that has been under development by the World Wide Web Consortium (W3C) since 1999.
Lithography	STL	3D format used by 3D printers. STL (STereoLithography) is a file format native to the stereolithography CAD software created by 3D Systems. STL is widely used for rapid prototyping and computer-aided manufacturing. STL files describe only the surface geometry of a three dimensional object without any representation of color, texture or other common CAD model attributes. The STL format specifies both ASCII and binary representations. Binary files are more common, since they are more compact. See also the StlOut (6.27.47) command.
Collada file	DAE	3D format used by rendering and animation software. COLLADA (from collaborative design activity) is an interchange file format for interactive 3D applications. It is managed by the nonprofit technology consortium, the Khronos Group, and has been adopted by ISO as a publicly available specification, ISO/PAS 17506.
Raster Format	**Extension**	**Description**
Bitmap*	BMP	Raster format created by Microsoft; used by the Clipboard.

(*) The **PictureExportScale** user preference sets the view size when exporting to picture formats (WMF, EMF, BMP). The following 3D formats are available through a separate module, BricsCAD Communicator; download itfrom the Bricsys website.

Format	Extension	Description
Standard for Exchange of Product data	STE, STP, STEP	STEP (Standard for the Exchange of Product Data) is an ISO standard exchange format used for representing three-dimensional data in a format that can be recognized by multiple programs.
Initial Graphics Exchange Specification	IGS, IGES	A file format which defines a vendor neutral data format that allows the digital exchange of information among Computer-aided design (CAD) systems.
CATIA V4	model	CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systèmes.
CATIA V5	CATPart, CATProduct	
Parasolid	XMT_TXT, XMT_BIN X_T, X_B	Parasolid is a geometric modeling kernel originally developed by ShapeData, now owned by Siemens PLM Software (formerly UGS Corp.), that can be licensed by other companies for use in their 3D computer graphics software products.
VDA-FS	VDA	VDA-FS is a CAD data exchange format for the transfer of surface models from one CAD system to another. Its name is an abbreviation of ”Verband der Automobilindustrie - Flächenschnittstelle”, which translates to the ”automotive industry association - surface data interface”. Standard was specified by the German organization VDA.
3D PDF	PDF	Portable Document Format (PDF) is a file format used to represent documents in a manner independent of application software, hardware, and operating system.

Settings

The import and export procedures using Communicator are controlled through a series of user preferences. See the Communicator section on the Settings (6.27.23) dialog.

Procedures

- Export to PDF settings (1.14.12)
- Exporting a drawing to PDF (1.14.12)

Related Commands

AcisOut (6.4.3) - exports 3D solids and surface entities in SAT format. BimExport (6.7.9) - saves a BIM model as an IFC file for use with an IFC-certified applications. BmpOut (6.6.14) - exports drawings in BMP format. CommunicatorInfo (6.9.39) - checks whether the Communicator product is correctly installed and prints a diagnostic report in the Prompt History window. DwfOut (6.10.47) - exports drawings in DWF format. ExportLayout (6.13.16) - exports all visible objects from the current layout to the model space of a new drawing. ExportPdf (6.13.17) - exports layouts in a drawing to a pdf docucment. IfcExport (6.17.2) - exports a BIM model to IFC, partially or entirely. Import (6.17.10) - imports CAD files in a variety of formats. MSlide (6.21.31) - exports drawings in WMF, EMF, and SLD formats. StlOut (6.27.47) - exports 3D solids and watertight meshes in a format that can be used for stereolithography (3D printing). VmlOut (6.31.16) - exports drawings in VML format. VSlide (6.31.22) - displays files in WMF, EMF, and SLD formats. WmfOut (6.33.12) - exports drawings in WMF, EMF, and SLD formats.

Figure 6.940



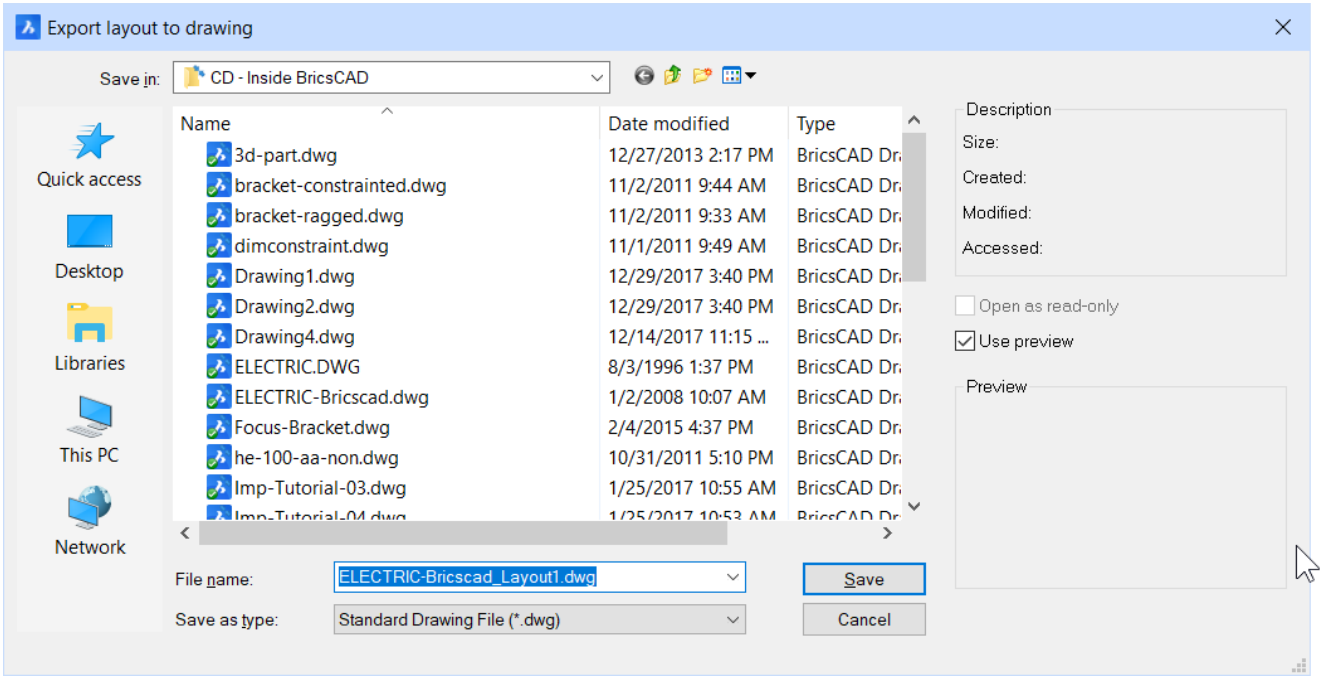
6.13.16 ExportLayout

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Exports all visible objects from the current layout to the model space of a new drawing.The command can only be used in a layout.

Accessing the Command

command bar: **exportlayout** menu: **File | Export Layout to Model** : **ExportLayout** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/exportlayout-dlg.png}) Type a name in the **File name** field, then press the **Save** button. Now that viewports are converted to polylines, and that entities are trimmed by the viewport boundary.

Figure 6.941



Related Commands

Export (6.13.15) - exports the drawing in a variety of raster and vector formats. WBlock (6.33.1) - writes blocks and other drawing parts to files on disk through a dialog box. -WBlock (6.33.2) - writes blocks and other drawing parts to files on disk through the command bar.

Figure 6.942



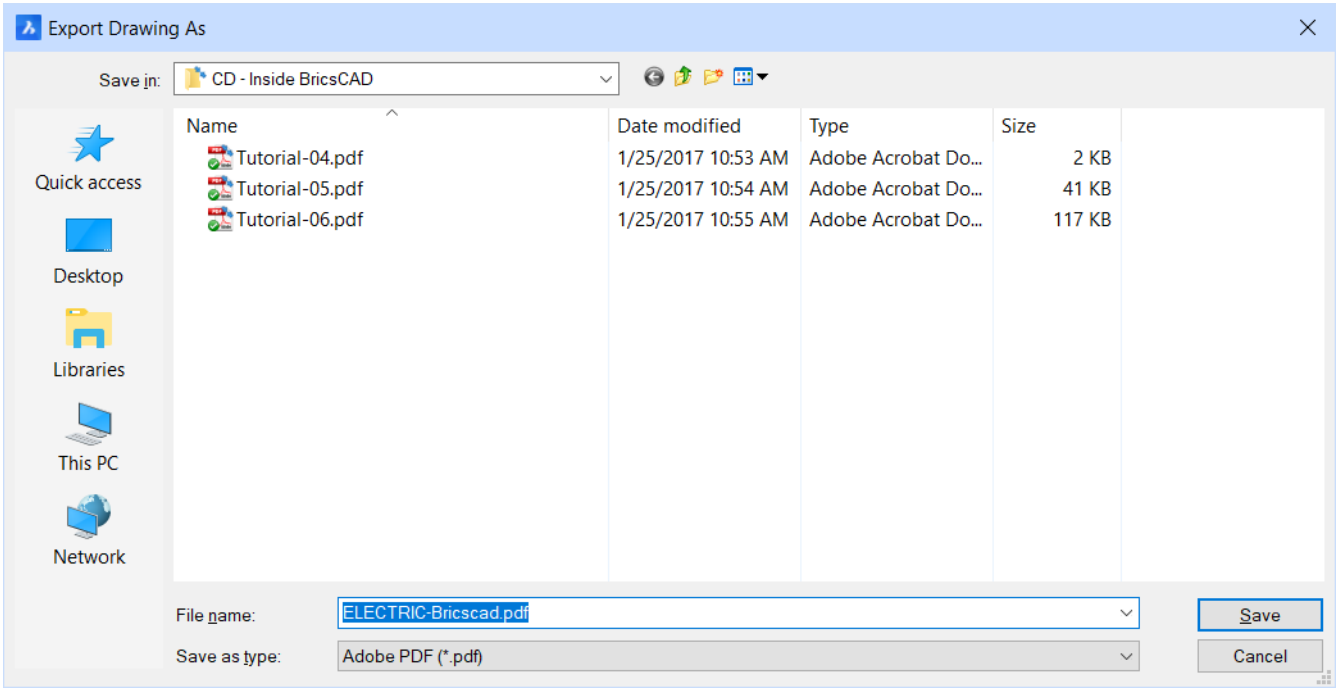
6.13.17 ExportPdf

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Exports the current drawing to a single-page PDF document. (To publish multiple drawings and layouts to a single PDF file, use the Publish command.)

Accessing the Command


command bar: **exportpdf** menu bar: **File| Export PDF** ribbon:**Home | File | Export PDF** : **ExportPdf** Displays a file dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/exportpdf-dlg.png}) Type a name in the **File name** field or accept the default name (**<DrawingName>.pdf**), then click the **Save** button.

Figure 6.943



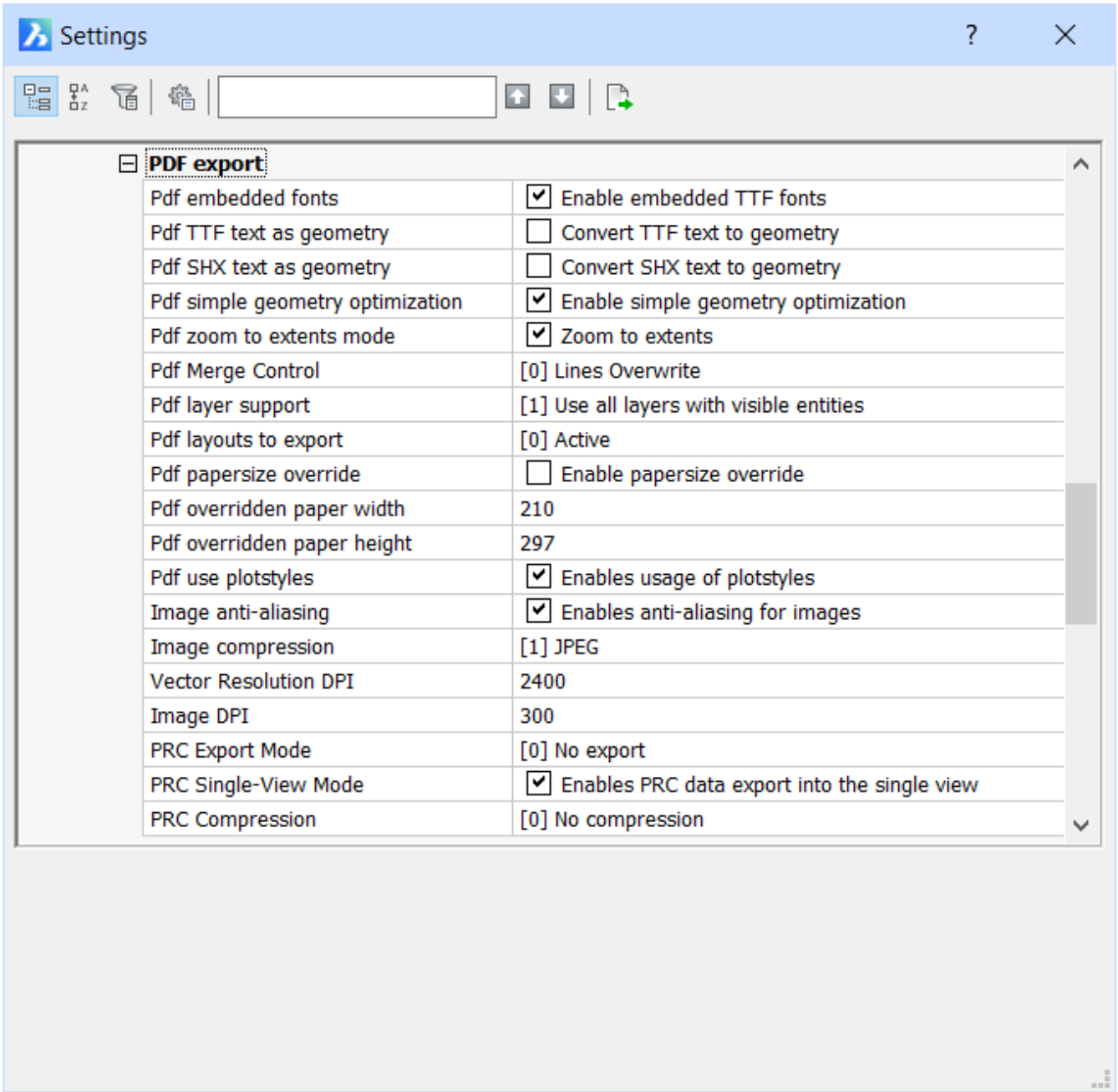
Command Options

Use the PDFoptions command to access the options for exporting PDF files in the PDF Export group in the Settings (6.27.23) dialog box, or else or in the Pdf Options (6.24.55) on the Publish (6.24.55) dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/pdfoptions-dlg.png})

Option	**Description**
Pdf embedded fonts	Determines if True Type fonts are embedded in the PDF file: • **On** - TrueType fonts are included in the PDF file • **Off** - TrueType fonts are excluded, such as when they are covered by a license that prohibits sharing
Pdf TTF text as geometry	Determines ifTrue Type fonts are exploded into geometry: • **On** - converts True Type text to geometry, useful forwhen the TTF files are covered by a license that prohibits sharing, or you want to make it harder to extract text • **Off** - keeps True Type text as fonts
Pdf SHX text as geometry	• **On** - converts SHX text to geometry, which is necessary if the receiving party does not have the same SHX fonts on their computer • **Off** - keeps SHX text as fonts
Pdf simple geometry optimization	• **On** - applies simple geometry optimization to unify separate segments to single polylines, and uses Bezier curve control points • **Off** - does not optimize the file

Pdf zoom to extents mode	• **OFF** - uses the scale and papersize of the page setup. • **ON** - zooms the geometry to the papersize.
Pdf Merge Control	Controls the appearance of crossing lines: • **0 - Lines Overwrite**: The top most line is visible at the intersection. • **1 - Lines Merge**: The colors of crossing lines are blended into a new color.
Pdf layer support	Controls the use of layers in the generated PDF file: • **0** - Do not export layers. • **1** - Only use visible layers • **2** - Use all layers, including layers that are OFF and FROZEN
Pdf layouts to export	Controls which layouts are exported. • **0** - Active layout only • **1** - All layouts in a single multi-sheet PDF file • **2** - Each layout in a separate single-sheet PDF file
Pdf papersize override	Enables papersize override to ignore the limits of the paper's size.
Pdf overridden paper width	PDF width to be used in the papersize override in millimeters.
Pdf overridden paper height	PdfPDF height to be used in the papersize override in millimeters.
Pdf use plotstyles	Enables the usage of plotstyles.
Image anti-aliasing	Enables anti-aliasing for images that require upscaling when exporting.
Image compression	Sets the compression for exported images. • **0** - no compression for cleaner imagers • **1** - JPEG compression for smaller files
Vector resolution DPI	Sets the resolution for vector graphics when exporting. Values between 72 and 40000 are accepted.
Image DPI	Sets the minimal resolution for raster images. The value cannot exceed the value of PdfVectorResolutionDPI.
PRC Export Mode	Determines how 3D models are exported to 3D PDF files: • **0** - 3D models are not exported from the drawing to the PDF file • **1** - 3D models are exported in BREP format • **2** - 3D models are exported as meshes
PRC Single-view Mode	Determines if the 3D model is exported into a single view.
PRC Compression	Determines the type of compression to use to reduce file size: • **0** - no compression • **1** - medium compression • **2** - high compression (smallest file size)

Figure 6.944


Settings
PDF export
Pdf embedded fonts
Enable embedded TTF fonts
Pdf TTF text as geometry
Convert TTF text to geometry
Pdf SHX text as geometry
Convert SHX text to geometry
Pdf simple geometry optimization
Enable simple geometry optimization
Pdf zoom to extents mode
Zoom to extents
Pdf Merge Control
[0] Lines Overwrite
Pdf layer support
[1] Use all layers with visible entities
Pdf layouts to export
[0] Active
Pdf papersize override
Enable papersize override
Pdf overridden paper width
210
Pdf overridden paper height
297
Pdf use plotstyles
Enables usage of plotstyles
Image anti-aliasing
Enables anti-aliasing for images
Image compression
[1] JPEG
Vector Resolution DPI
2400
Image DPI
300
PRC Export Mode
[0] No export
PRC Single-View Mode
Enables PRC data export into the single view
PRC Compression
[0] No compression

Related Commands

Export (6.13.15) - exports drawings in a variety of vector and raster formats. PageSetup (6.24.1) - creates and edits page setups for plotting drawings in the Drawing Explorer. Plot (6.24.26) - plots drawings to printers and to files, through prompts at the command bar; this command is meant for scripts and routines. Print (6.24.46) - prints drawings to plotters and files through a dialog box. -Publish (6.24.56) - prints a saved sheet list (= a list of model space or paper space layouts). Publish (6.24.55) - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.

Figure 6.945



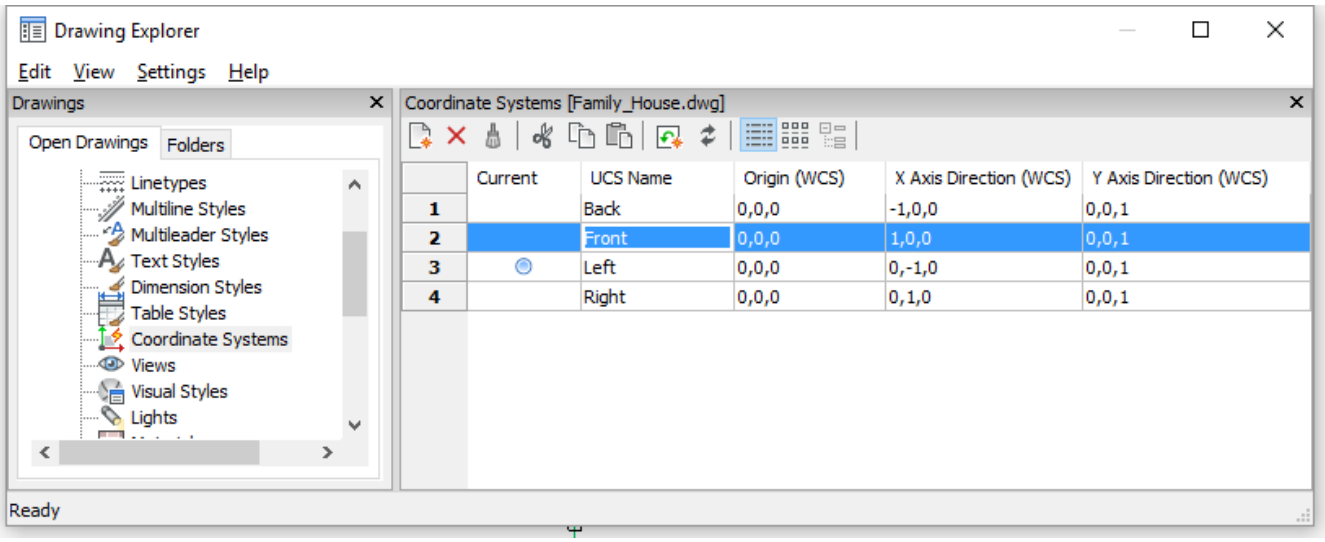
6.13.18 ExpUcs

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates, renames, and deletes named UCSs through the Drawing Explorer (short for ”explore user-defined coordinate systems”).

Accessing the Command

command bar: **expucs** alias: **uc, dducs, explorer (6.13.14)** menu bar: **Tools | Drawing Explorer | Coordinate Systems** : **ExpUcs** Displays the UCS section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc
Create, rename, or delete named UCSs, and then click X. Press F1 to access help.

Figure 6.946



Command Options

Option	Description
UCS Name	Lists the names of UCSs in the drawing; click to rename.UCS is short for "user-defined coordinate system."
Origin (WCS)	Reports the origin (0,0,0) of the UCS in x,y,z coordinates of the WCS; cannot be edited.WCS is short for "world coordinate system."
X Axis Direction (WCS)	Reports the direction of the x axis in WCS coordinates; cannot be edited.
Y Axis Direction (WCS)	Reports the direction of the y axis in WCS coordinates; cannot be edited.
Current	Indicates the current UCS.
Toolbar Icons	
New	Creates a new UCS; dismisses the Drawing Explorer, and then runs the Ucs command. See Ucs (6.30.2) command.
Erase	Deletes the selected UCS; the current UCS cannot be erased.

Related Commands

Ucs (6.30.2) - creates and modifies user-defined coordinate systems. UcsIcon (6.30.3) - controls the appearance of the UCS icon. SetUcs (6.27.25) - sets the UCS to a specified viewpoint. Plan (6.24.25) - changes the viewpoint to the plan view of the WCS or current UCS. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Settings (6.27.23) - sets parameters of UCSs. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.947



6.13.19 Extend

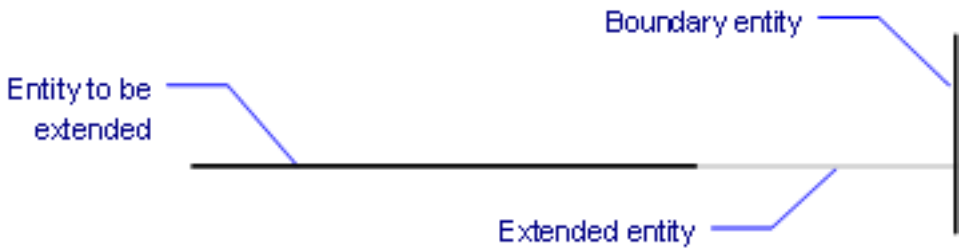
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Extends or trims one or more open entities to a boundary entity. **Note**: Hold down the **Shift** key to switch this command to trim mode.

Accessing the Command



command bar: **extend** alias: **ex** menu bar: **Modify | Extend** ribbon:**Change | Modify | Extend** toolbar: **Modify |**  : **Extend Prompts you in the command bar:** Select boundary entities for extend <ENTER to select all>: **(Choose one or more entities, or else press Enter to choose all)** Entities in set: 1 Select boundary entities for extend <ENTER to select all>: **(Choose more entities, or else press Enter to end entity selection)** Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/ Projection/ eRase]: **(Choose the entity to extend, or enter an option, or else press Shift to pick an entity to trim)** Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/Projection/eRase]: **(Choose another entity to extend, enter an option, or press Shift to pick an entity to trim, or else press Enter to exit the command)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/image177.png})

Figure 6.948



Command Options

Option	Description
Select boundary entities for extend	Choose one or more entities to use as boundaries. These are the entities to which subsequently picked entities are extended.
ENTER to select all	Press **Enter** to choose all entities in the drawing as boundaries. There is no need to pick individual entities as boundaries, except in certain cases.
Select entity to extend	Chooses the entity to extend.When either end of an entity could be extended to a boundary, BricsCAD extends the end closest to your pick point.
Edge mode	Toggles edge mode; prompts you: **[Extend/No extend] <No extend>** - **(enter an option)**: • **Extend** - extends entities to real and implied boundaries in 3D space • **No Extend** - extends entities only to real boundaries
Fence	Allows fence selection, an irregular selection window; prompts you: 1. **First point of fence** - **(pick the first point of the fence.)** 2. **Next fence** - **(pick the next point of the fence; press Enter to exit selection mode.)**
Crossing	Allows crossing rectangle selection; prompts you: 1. **First corner of crossing window** - **(specify the first corner.)** 2. **Second corner of crossing window** - **(specify the second corner.)** The entities that cross the specified rectangle are extended.
Projection	Specifies the how the entity is projected to the boundary; prompts you: **Projection [No projection/xy plane of Ucs/current View] <UCS>** - **(enter an option)**: • **No projection**- extends only those entities that intersect real boundaries in 3D space • **xy plane of UCS** - projects entities and boundaries to the x,y-plane of the current UCS, and then extends the projected entities that would intersect projected boundaries • **Current View** - projects entities into the current view, and then extends them accordingly
Erase	Deletes selected entities; prompts you: **Select entities to delete** - **(select one or more entities)**
Undo	Undoes the previous extend operation.
Shift-select to trim	Switches to trim mode: press and hold the **Shift**key to select the part of an entity to be trimmed at the intersection with the nearest boundary entities. See thethe Trim (6.29.26) command.

Procedures

- To extend entities (1.23.20)

Related Commands

Change (6.9.10) - changes the length of lines. Lengthen (6.20.21) - changes the length of open entities. Stretch (6.27.49) - stretches entities longer and shorter. Trim (6.29.26) - trims entities to a cutting line.

Figure 6.949



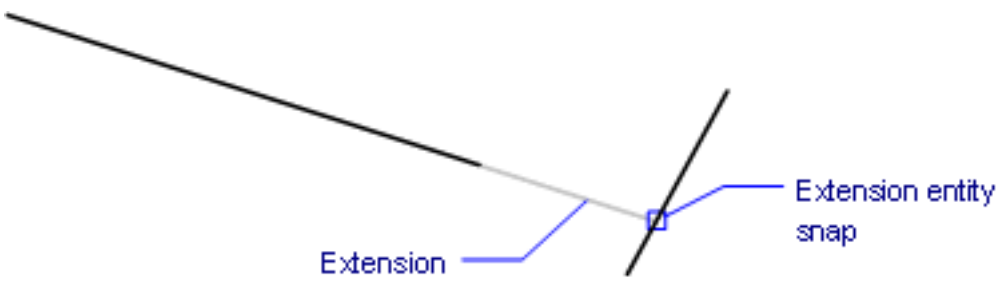
6.13.20 Extension

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles extension entity snap.

Accessing the Command

 command bar: **extension** transparent: **extension** ribbon:**Home | Entity Snaps | Snap to Extension** toolbar: **Entity Snaps** |  : **Extension** Displays no prompts.If extension entity snap is off, entering the command turns it on. And vice versa. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/image178.png}) When extension entity snap is on, the cursor snaps to the intersection of one or two extended entities.

Figure 6.950



Command Options

Option	Description
Extension	Entering this command toggles extension entity snap.This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.951



6.13.21 Extrude

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Extrudes closed and open 2D entities to 3D solids or 3D surfaces.

Accessing the Command



command bar: **extrude** alias: **ext** menu bar: **Model | 3D Solids | Extrude** ribbon:**Model | Solids | Extrude** toolbar: **3d Solids |**  : **Extrude Prompts you in the command bar:** Select entities/subentities to extrude or set [MOde]: **(Choose one or more closed 2D entities or faces of 3D solids)** Entities in set: 1 Select entities/subentities to extrude or set [MOde]: **(Choose more entities, or else press Enter to continue.)** Specify height of extrusion or [Direction/Path/Taper angle] <xx.xx>: **(Enter a number for the height, or pick two points, or press Enter to accept the default height, or else choose an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/ (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/image405.jpg}) Depending on the value of the DELOBJ (1.18.1) (Delete Entity) system variable, the source 2D entities are either retained or deleted, or else you are prompted as to whether you want to keep or delete the source entities.

Figure 6.952

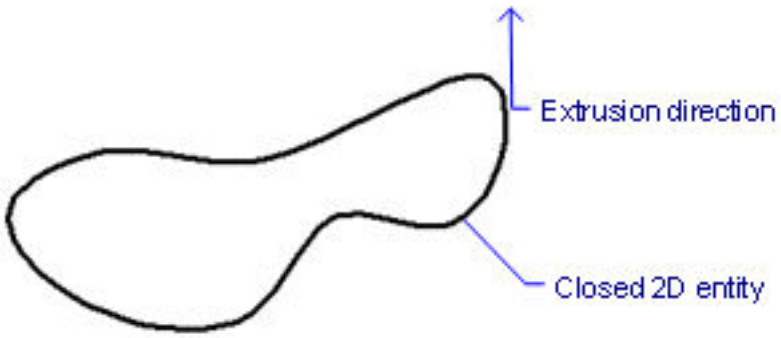
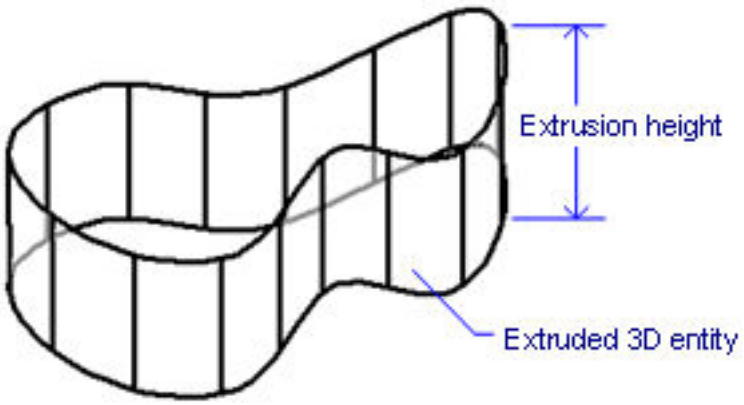


Figure 6.953



Command Options

Option	Description
Select entities	Specifies the entities to be extruded.Choose one or more 2D entities, regions, or faces of 3D solids.The resulting entity is either a 3D solid or a 3D surface: • 2D closed entities (like circles and closed polylines) and faces are extruded to 3D solids • 2D open entities (like lines and arcs) are exstruded to 3D surfaces • Use the **MOde** option extrude closed 2D entities as 3D surfaces When MOde is set to SOlid and you select an open 2D entity, the program does not extrude the entity, as open entities cannot be watertight. Depending on the value of the DELOBJ (1.13.11)variable, the source entities are either retained or deleted, or else you are prompted whether you want the entities deleted or not.
Mode	Determines how closed entities are extruded, as solids or surfaces.Prompts you: **Choose type of created entity [SOlid/SUrface] <SOlid>** - **(enter an option)**: • **SOlid** - returns to extruding closed entities as solids • **SUrface** - extrudes closed entities as surfaces
Specify height of extrusion	Specifies the height of the extrusion.The extrusion height is measured perpendicularly to the source entity. • **Enter a number** - specifies the height; enter a negative number to direct the extrusion downwards from the source entity • **Pick two points** - distance between two pick points specifies the height; for instance, you might pick two points on another entity

Direction	Sets the extrusion direction (up or down) and height at the same time.Prompts you: 1. **Specify start point of direction** - **(pick a point.)** 2. **Specify end point of direction** - **(pick another point.)**
Path	Specifies the extrusion height and taper angle by another entity. The program uses the path to determine how to extrude the source entity. Prompts you: **Select extrusion path or [Taper angle]** - **(pick an open entity to use as the path, such as a line, arc, open spline, or open polyline.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/im The path entity cannot lie in the same plane as the extrusion entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/im
Taper Angle	Specifies the taper angle for the extrusion; a positive angle tapers the extrusion inwardly, as shown by the illustration below. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/im Prompts you: **Specify angle of taper for extrusion <0>** - **(enter a value)**: • **Enter a number** - specify the angle; enter a negative number to taper the extrusion outwards • **Pick two points** - defines the angle geometrically The angle must be shallow enough so that the sloping sides do not intersect after the top of the extrusion.

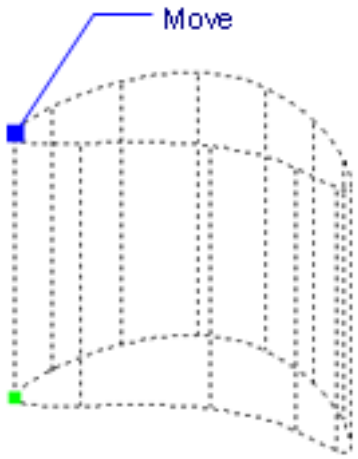
Grips Editing

Extrusion can be edited directly through grips:

1. Select the extruded solid. Notice that it has two grips.
2. Drag a grip to move the extrusion. (Both grips perform the same functions: move.)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/E/image755.png})

Figure 6.954



Related Commands

DmExtrude (6.12.13) - creates 3D solids or surfaces by extruding open or closed 2D entities or regions. Loft (6.20.37) - creates 3D solids passing through two or more cross sections. Sweep (6.27.60) - creates 3D solids or 3D surfaces by sweeping profiles (open or closed 2D entities or 3D subentities) along an open or closed path. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. Revolve (6.26.30) - revolves 2D entities into 3D solids or surfaces. Section (6.27.11) - creates 2D regions from 3D models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Subtract (6.27.56) - removes one solid model from a second. Union (6.30.6) - joins two or more solid models into a single entity.

Figure 6.955

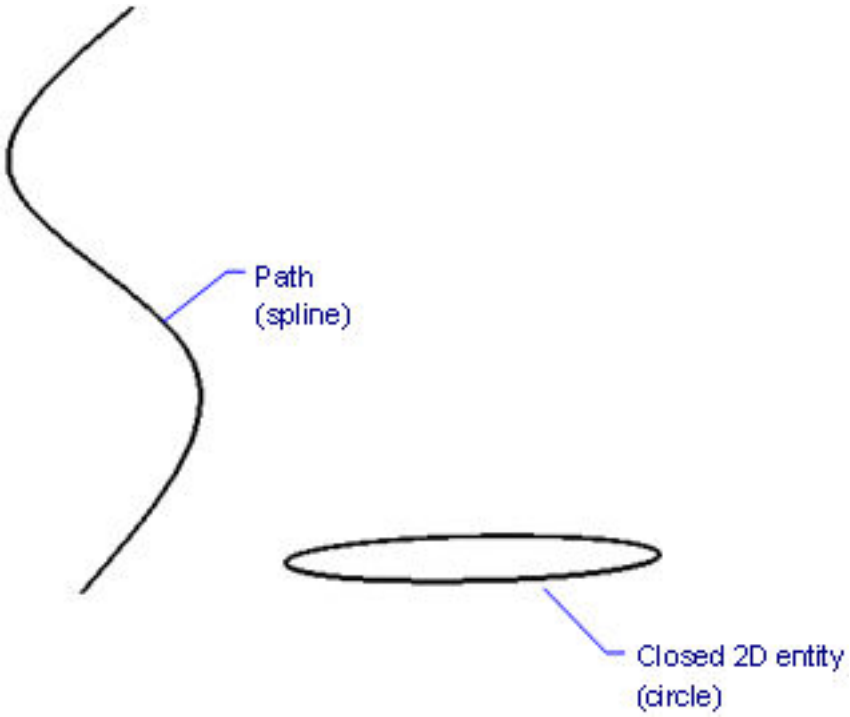


Figure 6.956

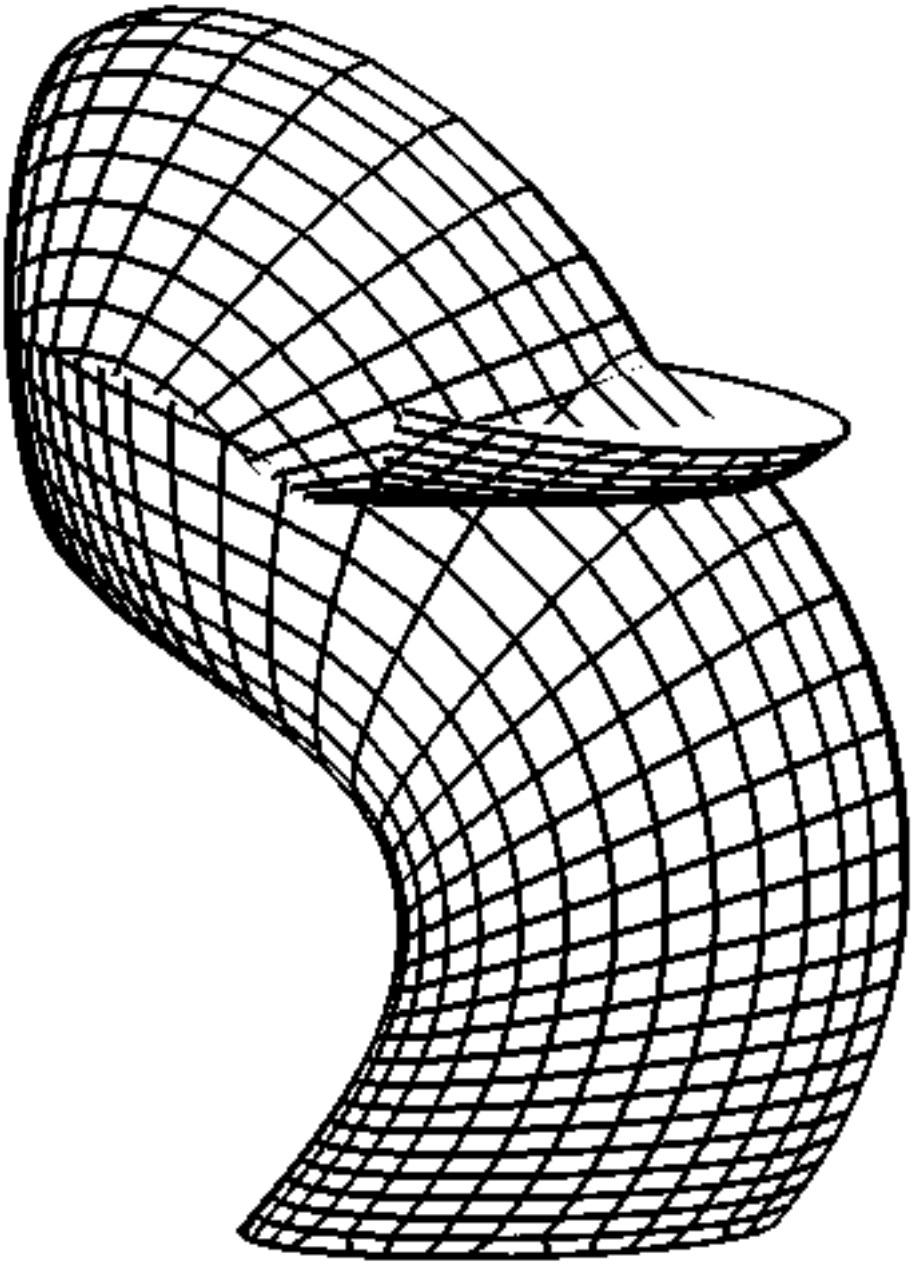
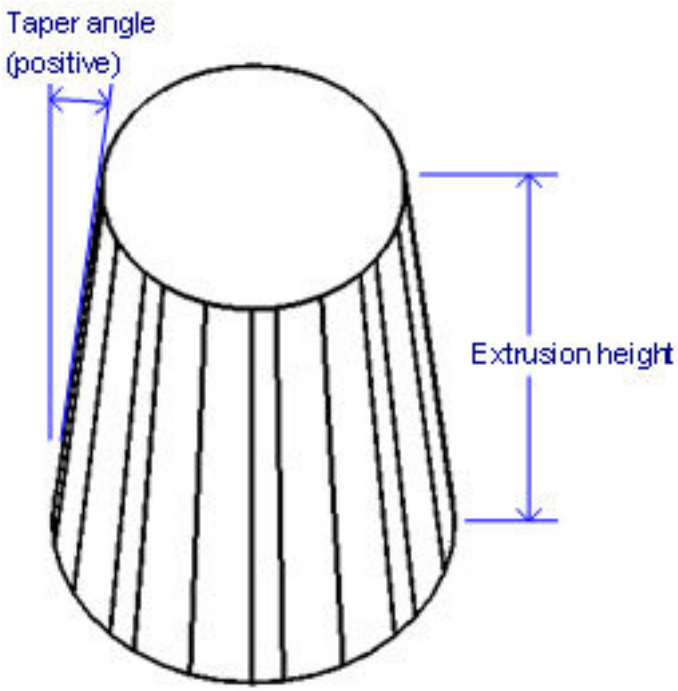


Figure 6.957



6.14 F

6.14.1 -FbxExport

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Exports 3D entities in the current drawing to an FBX file, which is used with rendering and animation programs; operates at the command line.

Figure 6.958



Accessing the Command

Keyboard: **-fbxexport :-FbxExport Prompts you:** What entities to export? [Selected/Visible] <Visible>:**(Enter s or v)** Export options. Entities: yes, lights: yes, cameras: yes, materials: yes What types to export? [Select/All] <Select>:**(Enter s or a)** Export entities? [Yes/No] <Yes>:**(Enter nor y)** Export lights? [Yes/No] <Yes>:**(Enter nor y)** Export cameras? [Yes/No] <Yes>:**(Enter nor y)** Export materials? [Yes/No] <Yes>:**(Enter nor y)** Export options. Entities: yes, lights: yes, cameras: yes, materials: yes How to export textures? [Embedded/Reference to file/Copy of file] <Embedded>:**(Enter an option)** Enter path to export fbx file: [C:\Bricsys\BricsCAD\Drawing1.fbx]: **(Press Enter to accept the path, or press ~ to access the FBX Export dialog box, or else type in a different path)**]:

Command Options

Option	Description
What entities to export?	Determines which entities to exports: • **Selected** - **selects the entities to be exported** • **Visible** - **export all visible entities (excludes those on frozen or off layers, and those not visible in the current viewport)** If you wish to export 2D entities, you must first give them a thickness with the Thickness variable.
What types to export?	Determines entity types to export: • **All** - exports all entity types • **Select** - prompts you to select among entities, lights, cameras, and materials
Export entities?	Determines if entities should be exported: • **Yes** - exports selected entities • **No** - does not export entities; use this option to export cameras, lighting, and materials only
Export lights?	Determines if lights should be exported: • **Yes** - lights are exported • **No** - lights are not exported
Export cameras?	Determines if cameras should be exported: • **Yes** - cameras are exported • **No** - cameras are not exported
Export materials?	Determines if materials should be exported: • **Yes** - materials are exported • **No** - materials are not exported
How to export textures?	Determines how texture files should be exported: • **Embedded** - includes the texture files into the FBX file • **Reference to file** - adds a reference link in the FBX file to the location of the texture files • **Copy of file** - copies the texture files to the destination folder, separately from the FBX file Texture files are not included in FBX files when they are very large, or when a common set of texture files are used by rendering and animation projects.

Enter path to export fbx file:	Specifies the path to the folder in which to place the FBX file: • **Enter** - accepts the path provided • **~** - displays the FBX Export dialog box Select a folder that is convenient for the rendering or animation software to access.

Related Commands

FbxExport (6.14.2) - Exports drawing contents to FBX files. Export (6.13.15) - exports drawing in a variety of file formats.

Figure 6.959



6.14.2 FbxExport

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Exports 3D entities in the current drawing to an FBX file, which is used with rendering and animation programs.

Accessing the Command

keyboard: **fbxexport** menu: **File** | **Export** and then choose FBX format **: FbxExport Prompts you:** What entities to export? [Selected/Visible] <Visible>:**(Enter s or v)** Export options. Entities: yes, lights: yes, cameras: yes, materials: yes What types to export? [Select/All] <Select>:**(Enter s or a)** Export entities? [Yes/No] <Yes>:**(Enter nor y)** Export lights? [Yes/No] <Yes>:**(Enter nor y)** Export cameras? [Yes/No] <Yes>:**(Enter nor y)** Export materials? [Yes/No] <Yes>:**(Enter nor y)** Export options. Entities: yes, lights: yes, cameras: yes, materials: yes How to export textures? [Embedded/Reference to file/Copy of file] <Embedded>:**(Enter an option)** Enter path to export fbx file: [C:\Bricsys\BricsCAD\Drawing1.fbx]: **(Press Enter to accept the path, or press ~ to access the FBX Export dialog box, or else type in a different path)**

Command Options

Option	Description
What entities to export?	Determines which entities to exports: • **Selected** - selects the entities to be exported • **Visible** - export all visible entities (excludes those on frozen or off layers, and those not visible in the current viewport) If you wish to export 2D entities, you must first give them a thickness.
What types to export?	Determines entity types to export: • **All** - exports all entity types • **Select** - prompts you to select among entities, lights, cameras, and materials
Export entities?	Determines if entities should be exported: • **Yes** - exports selected entities • **No** - does not export entities; use this option to export cameras, lighting, and materials only
Export lights?	Determines if lights should be exported: • **Yes** - lights are exported • **No** - lights are not exported
Export cameras?	Determines if cameras should be exported: • **Yes** - cameras are exported • **No** - cameras are not exported

Export materials?	Determines if materials should be exported: • **Yes** - materials are exported • **No** - materials are not exported
How to export textures?	Determines how texture files should be exported: • **Embedded** - includes the texture files into the FBX file • **Reference to file** - adds a reference link in the FBX file to the location of the texture files • **Copy of file** - copies the texture files to the destination folder, separately from the FBX file Texture files are not included in FBX files when they are very large, or when a common set of texture files are used by rendering and animation projects.
Enter path to export fbx file:	Specifies the path to the folder in which to place the FBX file. • **Enter** - accepts the path provided • **~** - displays the FBX Export dialog box Select a folder that is convenient for the rendering or animation software to access.

Related Commands

-FbxExport (6.14.1) - Exports drawing contents to FBX files at the command line. Export (6.13.15) - exports drawing in a variety of file formats.

Figure 6.960



6.14.3 Field

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Inserts field in multiline text, table cells, attributes, and so on. **Note:** A field is variable text that the program updates automatically, displaying such data as drawing properties (like a file name or the saved date), entity properties (like length, area, or layer), user-defined properties, plot settings, variables, and so on.If a field has no value, the the program displays hyphens (----).

Accessing the Command

 command bar: **field** menu bar: **Insert| Insert Field** toolbar: **Insert** |  ribbon: **Draw | Text | Fields** : **Field** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/F/FieldDocumentD Expand a catagory in the **Field Names** pane, then select a field name. The **Format** pane displays the format options for the selected field. Adjust the format, then click **OK**. **The program prompts you in the command bar:** Specify start point or [Height/Justify]:**(Enter an option)**

Command Options

Field category	**Dialog box**
Date&Time	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/F/Fie • Select a date type. – **Create date**: a random date – **Date**: current date – **PlotDate**: the date the drawing was last plotted – **SaveDate**:the date the drawing was last saved • Select a date format in the list or type a format in the **Date format** field.
Document	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/F/Fie • Select a document property (6.10.42). • Select a format
Linked	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/F/Fie • Click the **Hyperlink** button. The**Edit Hyperlink** dialog box displays. • Copy/Paste an URL in the **Link to file or URL** field. or Click the **Browse** (...) button and select a file. • Click the OK button on the**Edit Hyperlink** dialog box.
Objects	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/F/Fie

Plot
Variables
Sheet Set

Option	Description
Height	Specifies the text height.Prompts you: **Specify height <current height>- (enter a value for the height of the field text, or pick two points, or else press Enter to accept the current height.)**
Justify	Sets the text justification.Prompts you: **Enter justification [TL/TC/TR/ML/MC/MR/BL/BC/BR]<Top Left>- (enter a justification option.)**

Procedures

- General procedure to create fields (1.21.23)
- To display an entity property in a field (1.21.23)
- To edit a field (1.21.23)
- To update a field (1.21.23)

Related Commands

AttDef (6.4.27) - defines how a block's attributes will be created through a dialog box. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Dwgprops (6.10.42) - opens the Drawing Properties dialog box, showing the general information and user defined properties stored with a drawing. Mtext (6.21.35) and -MText (6.21.33) - places paragraph text. Updatefield (6.30.10) - updates fields in the selected Mtext entities. Table (6.29.1) - creates a table entity in the drawing.

Figure 6.961

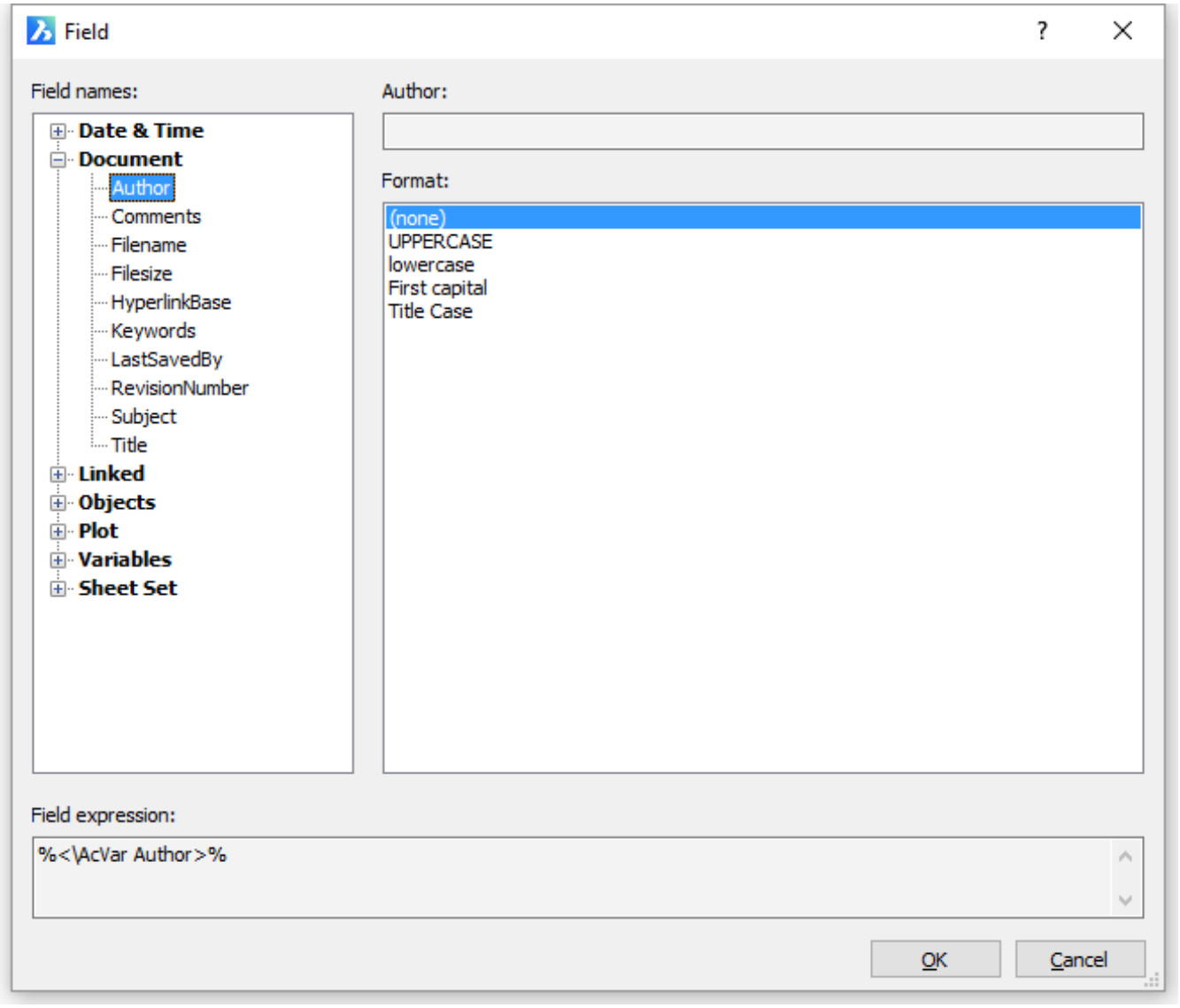

Field
Field names:
Date & Time
Document
Author
Comments
Filename
Filesize
HyperlinkBase
Keywords
LastSavedBy
RevisionNumber
Subject
Title
Linked
Objects
Plot
Variables
Sheet Set
Author:
Format:
(none)
UPPERCASE
lowercase
First capital
Title Case
Field expression:
%<\AcVar Author>%
OK
Cancel

Figure 6.962

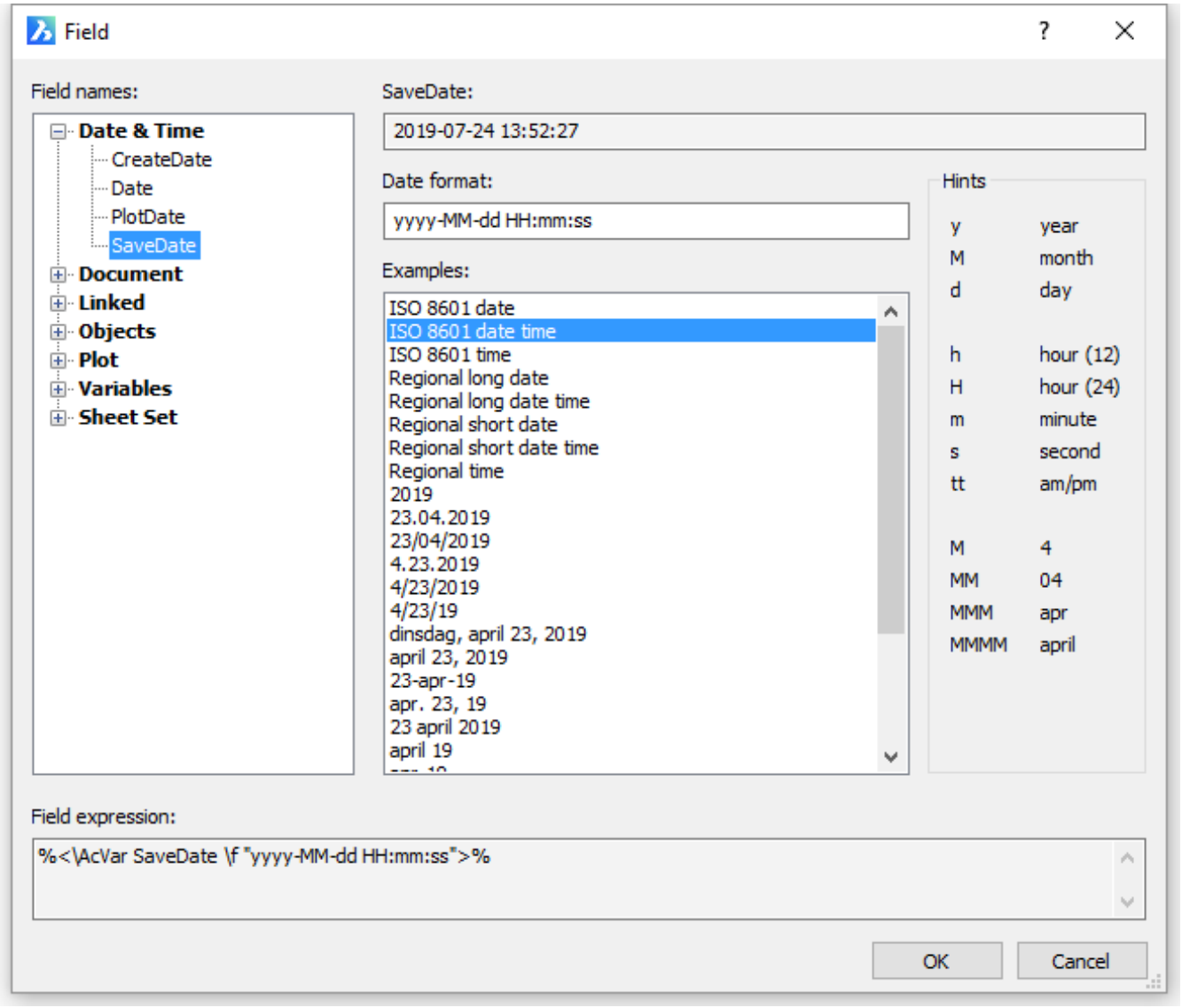

Field
Field names:
Date & Time
CreateDate
Date
PlotDate
SaveDate
Document
Linked
Objects
Plot
Variables
Sheet Set
SaveDate:
2019-07-24 13:52:27
Date format:
yyyy-MM-dd HH:mm:ss
Examples:
ISO 8601 date
ISO 8601 date time
ISO 8601 time
Regional long date
Regional long date time
Regional short date
Regional short date time
Regional time
2019
23.04.2019
23/04/2019
4.23.2019
4/23/2019
4/23/19
dinsdag, april 23, 2019
april 23, 2019
23-apr-19
apr. 23, 19
23 april 2019
april 19
Hints
y year
M month
d day
h hour (12)
H hour (24)
m minute
s second
tt am/pm
M 4
MM 04
MMM apr
MMMM april
Field expression:
%<\AcVar SaveDate \f "yyyy-MM-dd HH:mm:ss">%
OK
Cancel

Figure 6.963

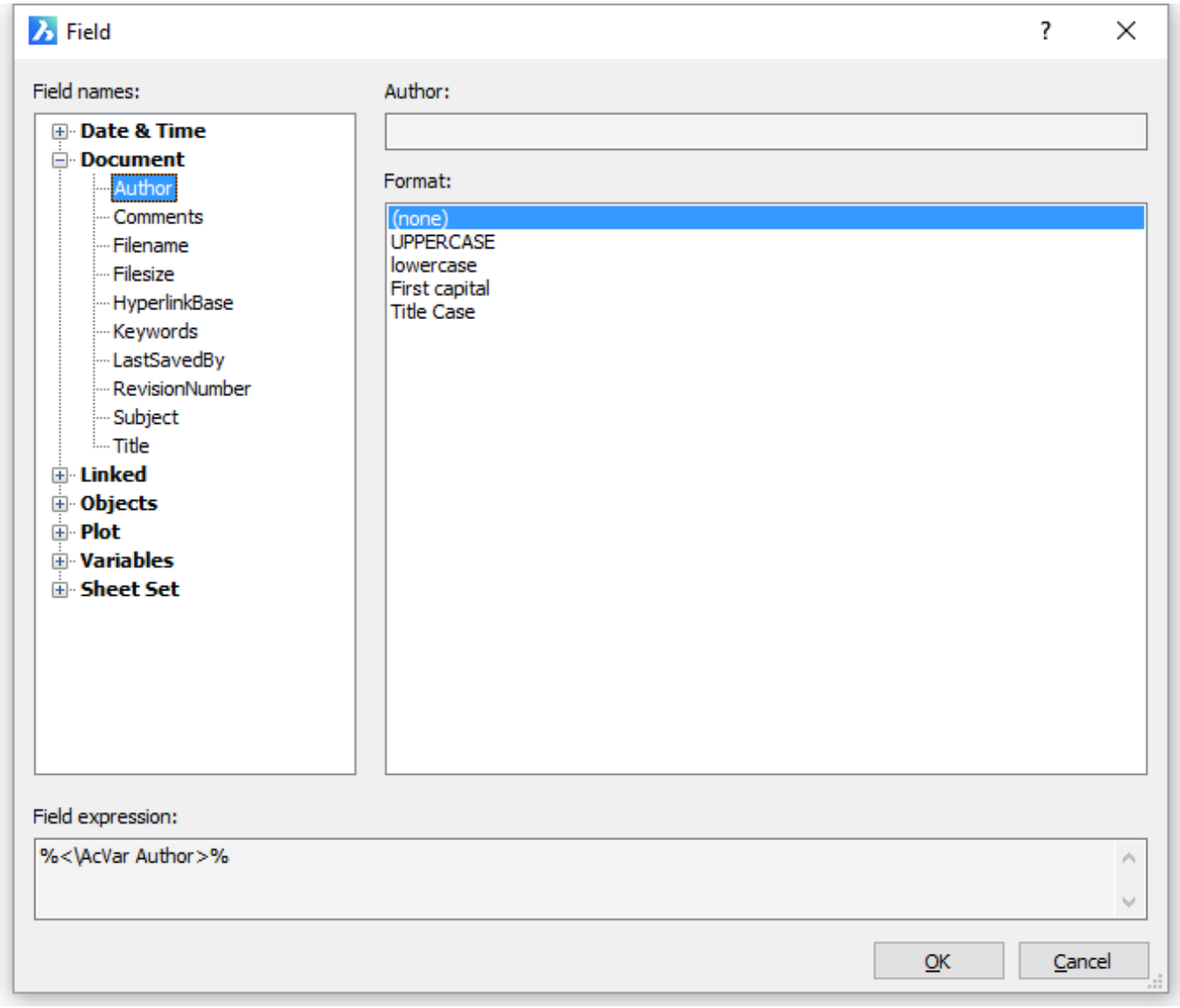

Field
Field names:
Date & Time
Document
Author
Comments
Filename
Filesize
HyperlinkBase
Keywords
LastSavedBy
RevisionNumber
Subject
Title
Linked
Objects
Plot
Variables
Sheet Set
Author:
Format:
(none)
UPPERCASE
lowercase
First capital
Title Case
Field expression:
%<\AcVar Author>%
OK
Cancel

Figure 6.964

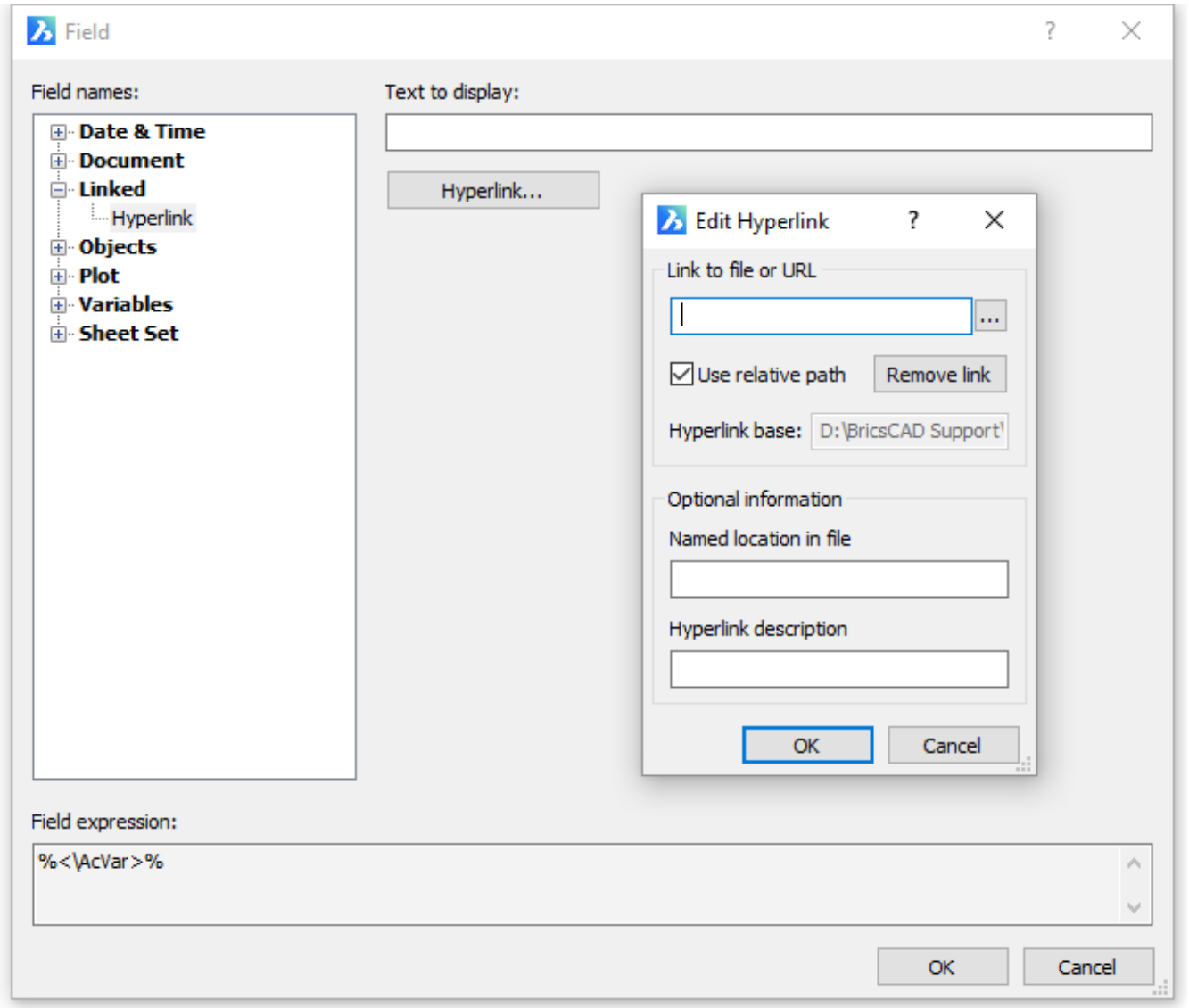

Field
Field names:
Date & Time
Document
Linked
Hyperlink
Objects
Plot
Variables
Sheet Set
Text to display:
Hyperlink...
Edit Hyperlink
Link to file or URL
Use relative path
Remove link
Hyperlink base:
D:\BricsCAD Support\
Optional information
Named location in file
Hyperlink description
OK
Cancel
Field expression:
%<\AcVar>%
OK
Cancel

Figure 6.965

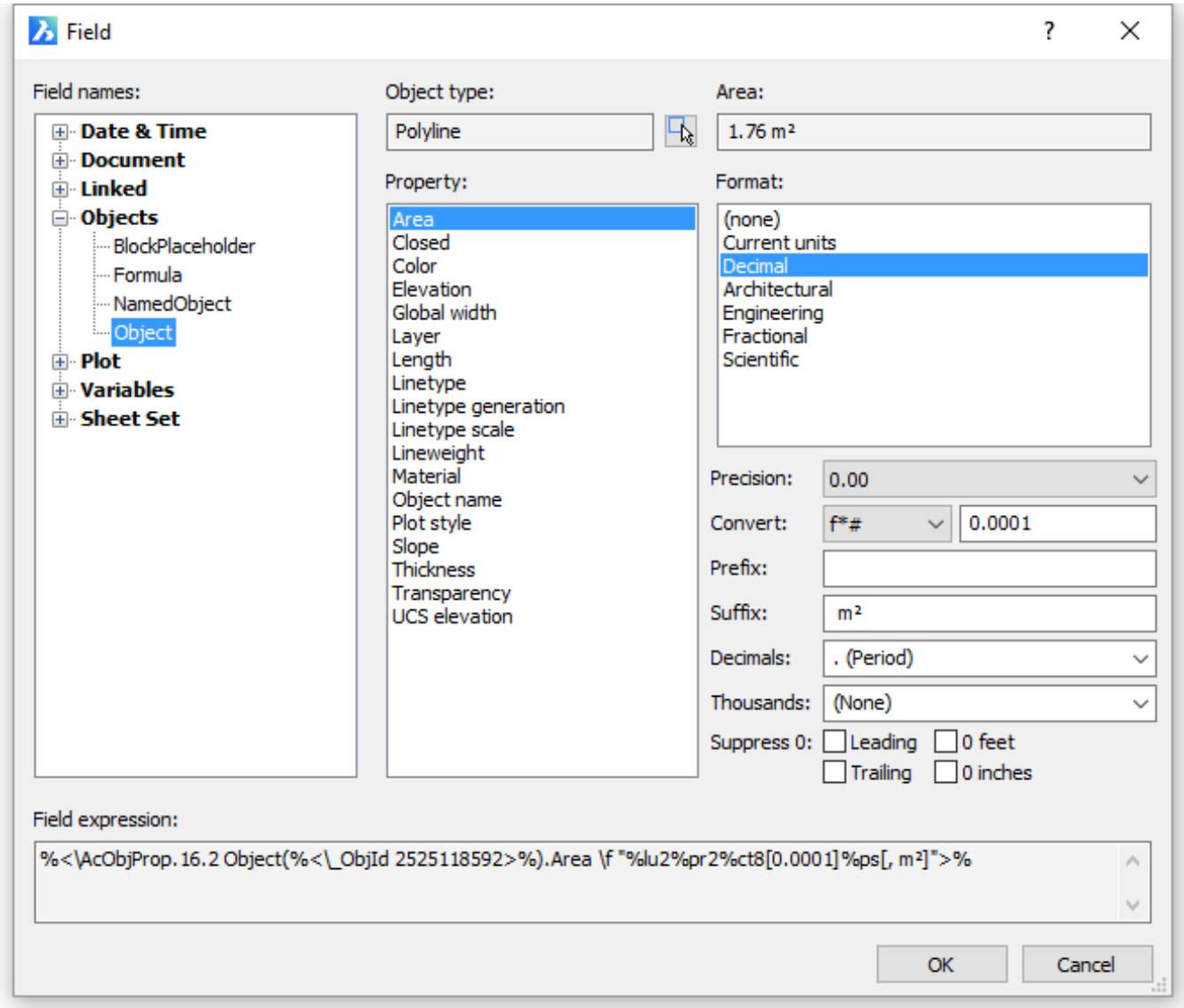

Field
Field names:
Date & Time
Document
Linked
Objects
BlockPlaceholder
Formula
NamedObject
Object
Plot
Variables
Sheet Set
Object type:
Polyline
Area:
1.76 m²
Property:
Area
Closed
Color
Elevation
Global width
Layer
Length
Linetype
Linetype generation
Linetype scale
Lineweight
Material
Object name
Plot style
Slope
Thickness
Transparency
UCS elevation
Format:
(none)
Current units
Decimal
Architectural
Engineering
Fractional
Scientific
Precision:
0.00
Convert:
f*#
0.0001
Prefix:
Suffix:
m²
Decimals:
. (Period)
Thousands:
(None)
Suppress 0:
Leading
0 feet
Trailing
0 inches
Field expression:
%<\AcObjProp.16.2 Object(%<_ObjId 2525118592>%).Area \f "%lu2%pr2%ct8[0.0001]%ps[, m²]">%
OK
Cancel

Figure 6.966



6.14.4 FileOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Opens drawing (DWG), template (DWT), and interchange (DXF) files from the command line, replacing the current drawing.

Accessing the Command

command bar: **fileopen** : **FileOpen Prompts you in the command bar:** Save changes to drawing file? [Yes/No] <Yes>: **(Enter Y or N)** Drawing name: **(Type the name of a drawing, including the path)** If the file extension is not specified, BricsCAD searches for a drawing (DWG) file.

Command Options

Option	Description
Save changes to drawing file?	Specifies whether the current drawing should be saved: • **Yes** - drawing is saved • **No** - drawing is not saved
Drawing name	Type the name of a drawing, including the path; enter **~** to display the Open Drawing dialog box.

Related Commands

Open (6.23.8) - opens drawing (DWG), template (DWT), and interchange (DXF) files.

Figure 6.967



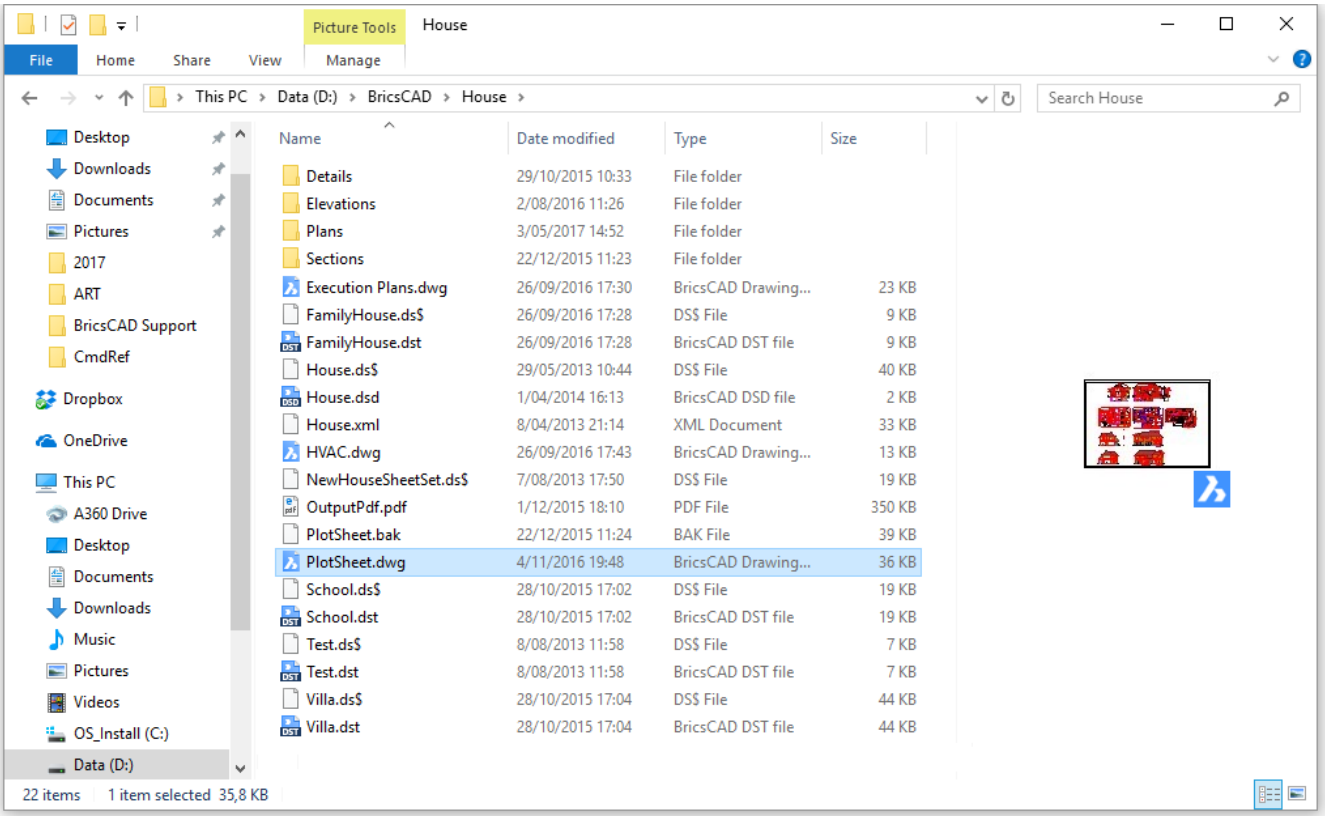
6.14.5 Files

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the operating system's file manager to access files.

Accessing the Command

command bar: **files** : **Files** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Com

Figure 6.968



Command Options

See Windows documentation.

Related Commands

Cal (6.9.1) - runs the Calculator application. Start (6.27.44) - starts other applications. Shell (6.27.32) - opens the operating system command prompt window; runs other applications.

Figure 6.969



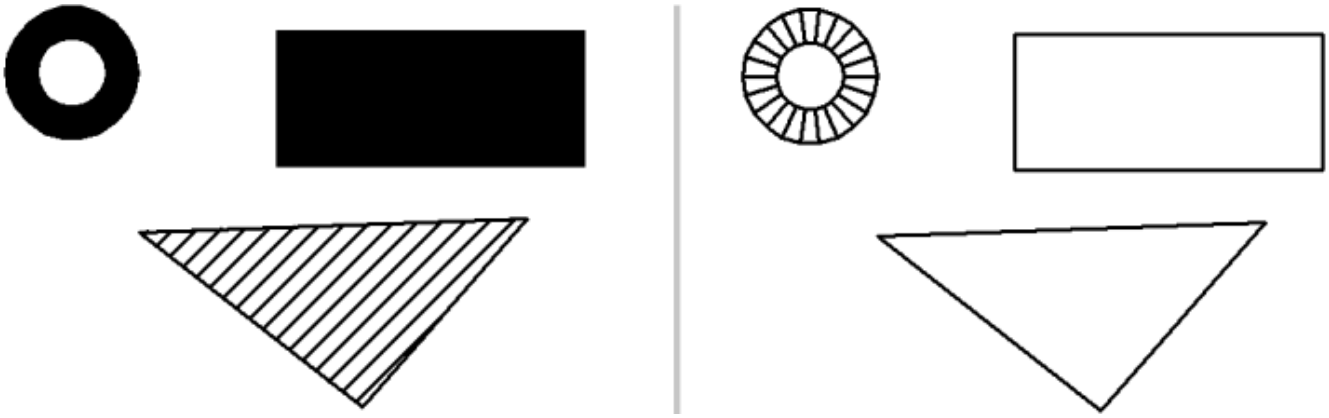
6.14.6 Fill

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the display of filled and hatched areas, such as of wide polylines, hatches, and 2D solids.

Accessing the Command

command bar: **fill** : **Fill Prompts you in the command bar:** FILLMODE is currently on [ON/OFf/Toggle] <On>: **(Enter an option)** To come into effect, this command must be followed by the Regen (6.26.15) command: : **regen** Left: Fill on; right: Fill off. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image199

Figure 6.970



Command Options

Option	Description
OFF	Turns off the display of filled areas.
Toggle	Toggles fill mode; if on, turns it off, and vice versa.
On	Turns on the display of filled areas.

Note: This command was useful for early releases of the program, when computers were not fast enough to display filled areas quickly.

Related Commands

Settings (6.27.23) - changes the FillMode system variable through a dialog box. This setting affects the entities drawn by the following commands: PLine (6.24.27) - draws polylines with width. Donut (6.10.35) - draws fat circles out of polylines. Polygon (6.24.43) - draws regular polygons out of polylines. Rectang (6.26.6) - draws rectangles out of polylines. Solid (6.27.37) - draws solid-filled entities with 3 and 4 sides. Trace (6.29.24) - draws wide lines.

Figure 6.971



6.14.7 Fillet

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Fillets intersections; joins intersecting lines with an arc of any suitable radius.

Accessing the Command

command bar: **fillet** alias: **f** menu bar: **Modify | Fillet** toolbar: **Modify |** : **Fillet Prompts you in the command bar:** Fillet (radius=10): Select first entity or [fillet Settings/ Polyline/ Radius/ Trim/ Undo/ Multiple]: **(Choose an entity or enter an option)** Select second entity (select with pressed SHIFT to make corner): **(Choose the other entity)**

Command Options

Option	Description
Select first entity	Choose one or two of the following entities: • Lines, including parallel lines • All vertices of a single 2D polyline; two polylines cannot be filleted • Rays, including parallel rays • Infinite lines, including parallel infinite lines • Arcs • Circles
Select second entity	Choose the second entity.Press and hold the **Shift** key to create a corner (radius = 0). The selected entities are extended or trimmed at the intersection point. The selected part of the entities is kept. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ If the second entity is parallel to the first entity the entities are connected by a semicircle. If the parallel entities are not of equal length, then the second entity is extended or trimmed when Trim Mode = Trim.
fillet Settings	Displays the Chamfer/Fillet section of the Settings dialog box: • **Fillet radius** - sets the radius of the fillet arc • **Trim mode** - toggles whether the entities are trimmed back to meet the fillet radius (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/F/fille settings-dlg.png})
Polyline	Fillets the all vertices of a polyline.Prompts you: **Select 2D polyline to fillet** - **(choose a single polyline.)** (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Radius	Prompts you to type the Fillet radius in the command bar.

Trim	Sets the trim mode for chamfers and fillets.Prompts you: **[Trim/No] <Trim>** - **(choose an option or press Enter to keep the current mode.)** • **Trim** - trims or extends the selected entities. • **No**- creates the chamfer or fillet, but leaves the selected entities unchanged.
Undo	Undoes the last fillet when in Multiple mode.
Multiple	Lets you create additional fillets using the same settings without having to restart the command.

Procedures

- Filleting two entities or polyline segments (1.23.27)
- Filleting all vertices of a polyline (1.23.27)
- Filleting two parallel lines (1.23.27)

Related Commands

Break (6.6.18) - cuts parts out of entities. Chamfer (6.9.9) - chamfers intersections. DmChamfer (6.12.4) - creates an equal distance chamfer between adjacent faces sharing a sharp edge. DmFillet (6.12.14) - creates a smooth fillet between adjacent faces sharing a sharp edge. Extend (6.13.19) - extends open entities to a boundary. Settings (6.27.23) - specifies parameters for fillets. Trim (6.29.26) - trims entities where they cross other entities.

Figure 6.972

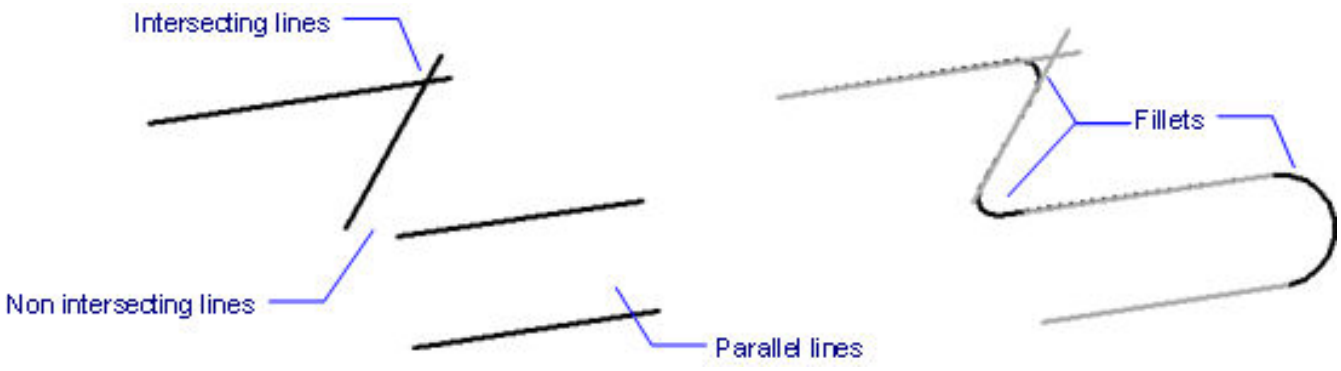


Figure 6.973

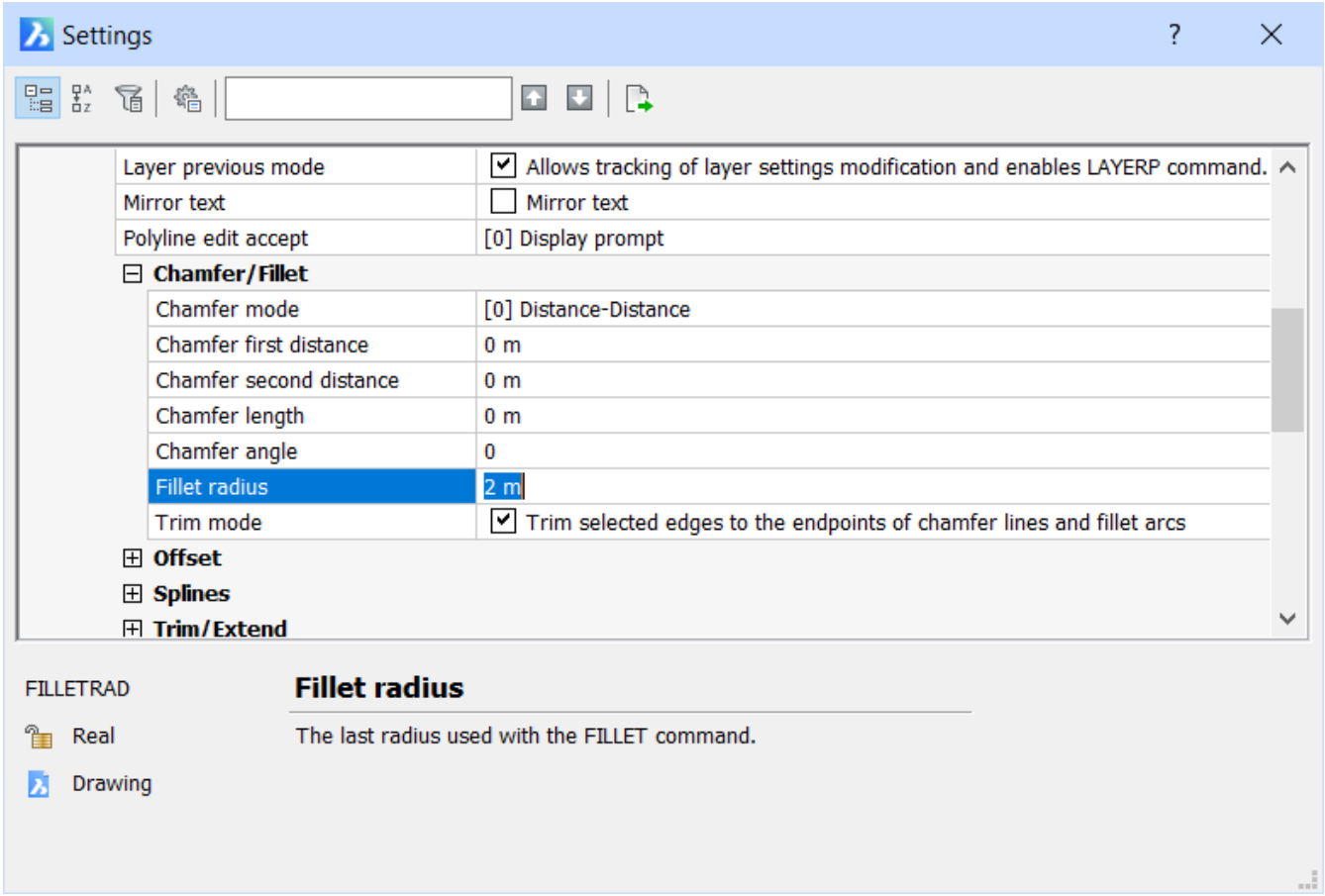


Figure 6.974

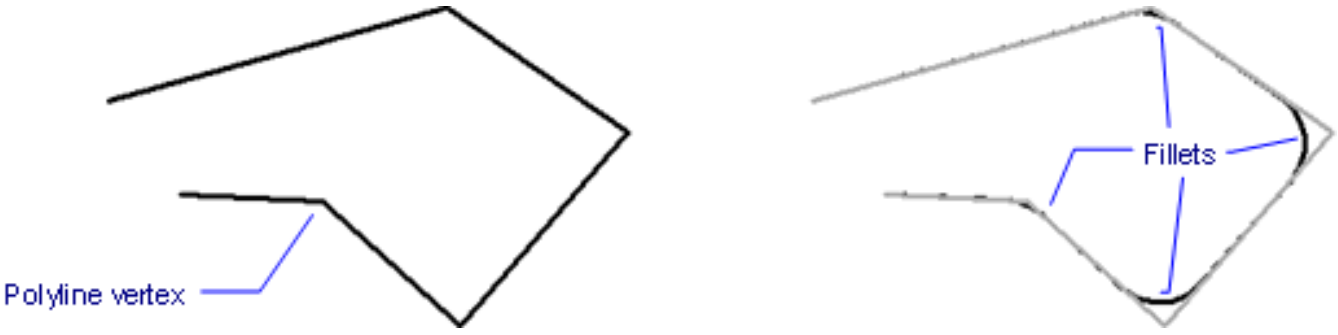


Figure 6.975



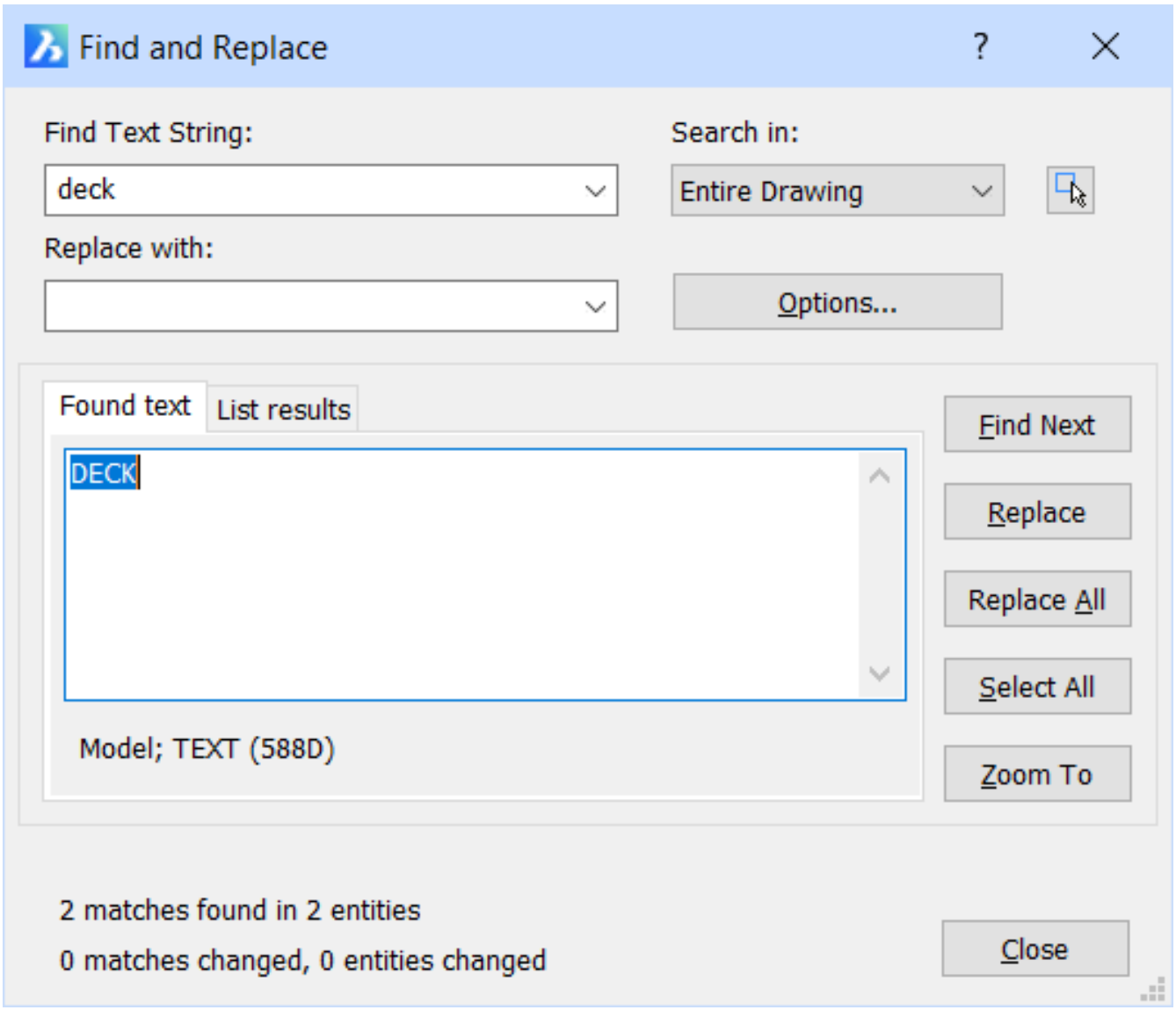
6.14.8 Find

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Searches for text and optionally replaces it; searches for text in blocks, attributes, dimensions, and hyperlinks.

Accessing the Command


command bar: **find** keyboard shortcut: **Ctrl+F** menu bar: **Edit | Find** ribbon:**Draw | Text | Find** : **Find** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/F/find-dlg.png}) Enter text to look for or replace, and then click Find Next.

Figure 6.976


Find and Replace
?
Find Text String:
deck
Search in:
Entire Drawing
Replace with:
Options...
Found text
List results
DECK
Model; TEXT (588D)
Find Next
Replace
Replace All
Select All
Zoom To
2 matches found in 2 entities
0 matches changed, 0 entities changed
Close

Command Options

The shortcut keystrokes listed below work only in this dialog box.

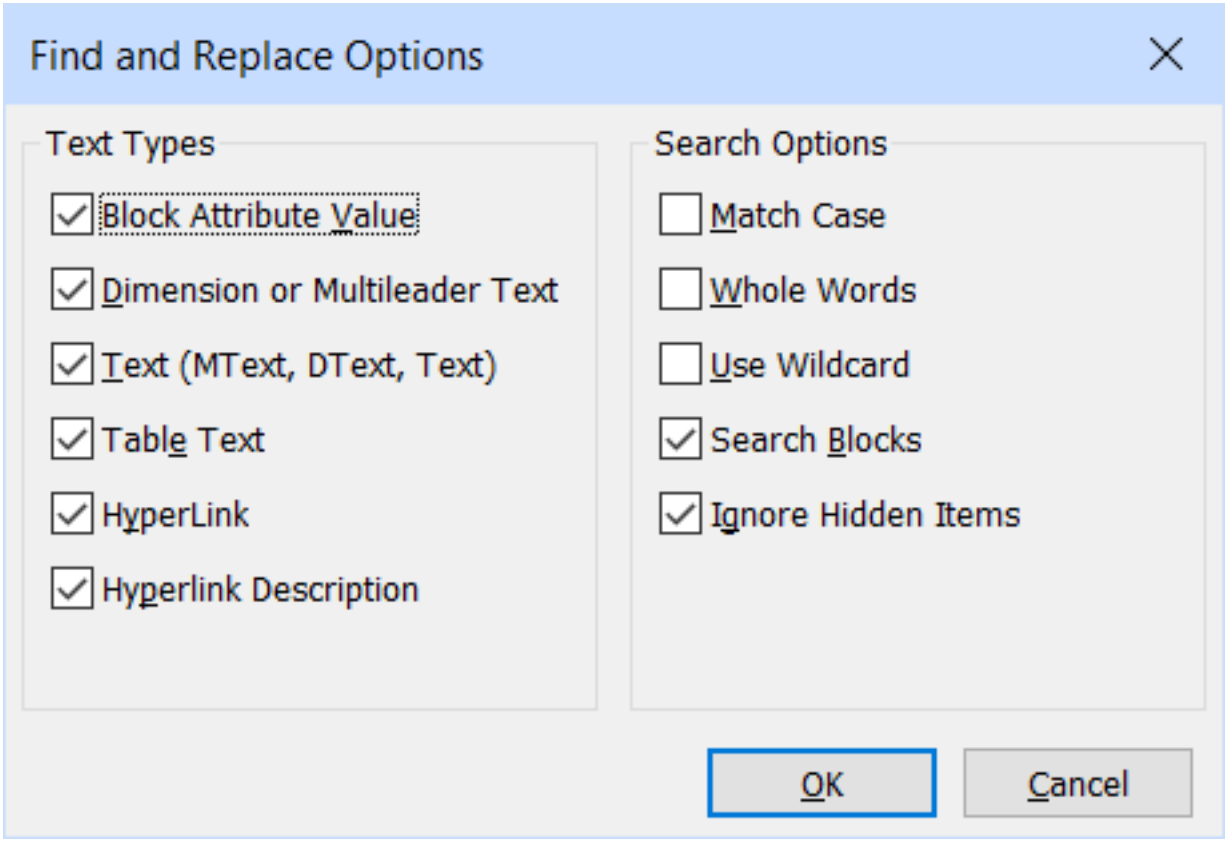
Option	Shortcut Keystroke	Description
Find Text String		Specifies the text to be located.
Replace With		Specifies the text with which to replace the located text.Leave this field empty when only searching for text.
Search In		Specifies the range of search: • **Entire Drawing** - searches all text in the drawing. • **Current Selection** - searches only among the selected entities.
Select Entities		Dismisses the dialog box so that you can select the entities to search; prompts you: 1. **Select entities** - **(choose one or more entities.)** 2. **Select entities** - **(choose more entities, or else press Enter to return to the dialog box.)**
Options		Displays the Find and Replace Options dialog box; allows you to specify the kinds of text entities to search; see below.
Find Next	Alt+F	Finds the next occurrence of the text string.
Replace	Alt+R	Replaces the found text string with the replacement string.
Replace All	Alt+A	Replaces all occurrences of the text string with the replacement string.
Select All		Selects all occurrences of the text string.
Zoom To		Zooms to the part of the drawing containing the found text string.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/F/find-options-dlg.png}) Choose options, and then click **OK** .

Find and Replace Options	Description
Block Attribute Value	When checked, searches the values of block attributes as well.
Dimension or Multileader Text	When checked, searches dimension text as well.
Text (MText, DText, Text)	When checked, searches regular text as well.
Table Text	When checked, searches texts in tables.
Hyperlink	When checked, searches in hyperlinks (URLs) as well.
Hyperlink Description	When checked, searches in hyperlink descriptions as well.

Match Case	Searches the case of the text: • **Yes** - finds text that matches upper and lower case. For example, searching for ”BricsCAD” matches BricsCAD but not bricscad • **No** - finds all text, regardless of upper or lowercase
Whole Words Only	Searches for full words: • **Yes** - finds only text that matches entire words. For example, searching for ”CAD” matches CAD but not BricsCAD • **No** - finds all text, including text embedded in longer words
Use Wildcard	When checked, allows the use of wildcard characters such as * and ?
Search blocks	When checked, searches for matching text strings inside blocks.
Ignore Hidden Items	When checked, ignores content on hidden and frozen layers.

Figure 6.977



Procedures

- To set the Find and Replace options (1.21.22)
- To find text (1.21.22)
- To replace text (1.21.22)

Related Commands

Text (6.29.12) - places single-lines of text in drawings. -Text (6.29.12) - places text and evaluates LISP expressions. MText (6.21.35) and -MText (6.21.33) - places paragraph text. AttDef (6.4.27) - defines attribute text. DdEdit (6.10.14) - edits text. Hyperlink (6.16.12) - adds hyperlinks to entities. Spell (6.27.63) - checks the spelling of words. Properties (6.24.49) - modifies the properties of text.

Figure 6.978



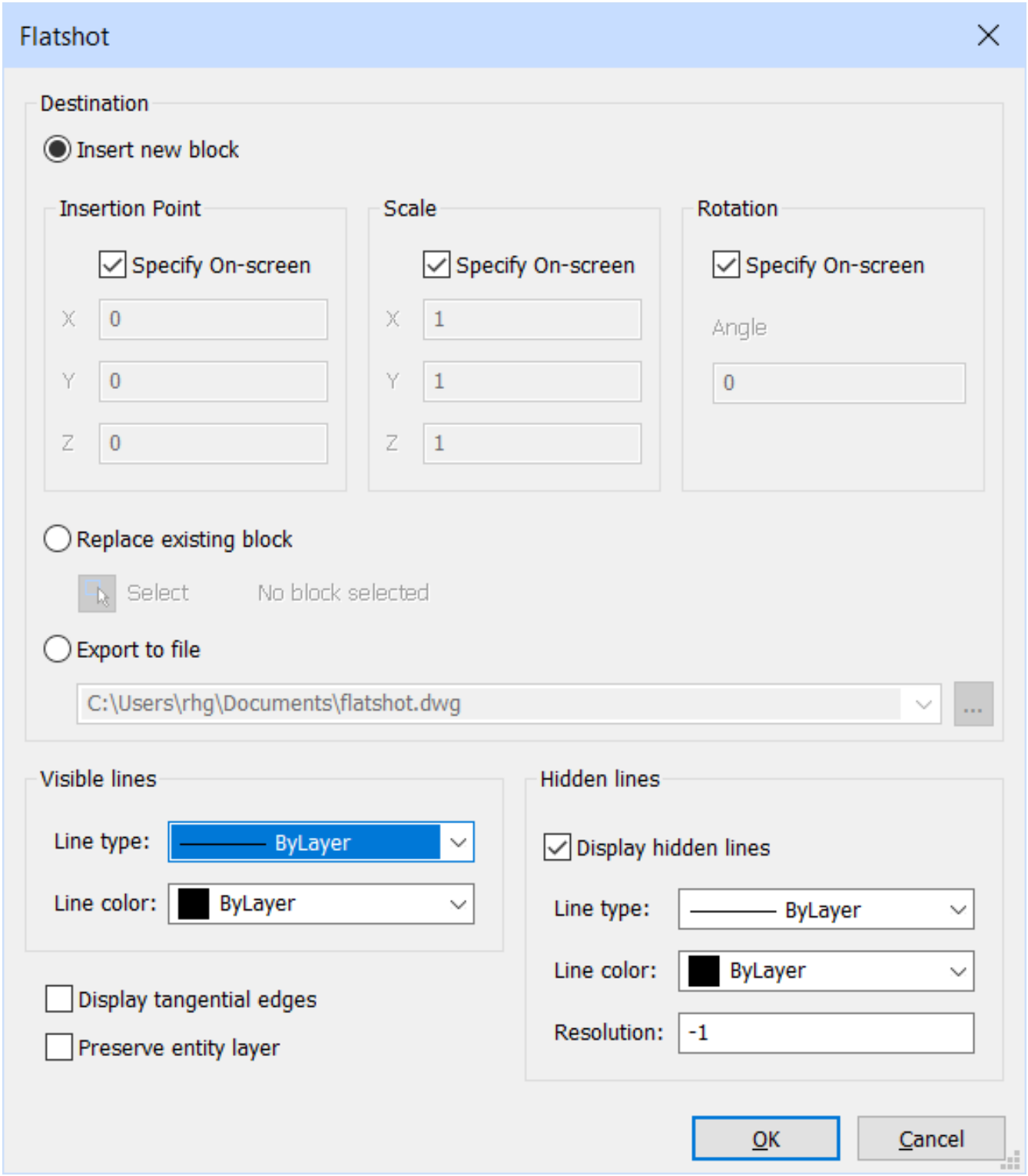
6.14.9 FlatShot

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a flattened, hidden line representation of 3D solids, 3D surfaces, and polyface meshes as a block or a new drawing. **Note:**This command works only in model space.

Accessing the Command

 command bar: **flatshot** menu bar: **Model | Direct Modeling | Sections | Flatshot** ribbon:**Model | Sections | Section Plane | Flatshot** toolbar: **Sections | Flatshot** : **FlatShot** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/F/flatshot-dlg.png})

Figure 6.979


Flatshot
Destination
Insert new block
Insertion Point
Specify On-screen
X 0
Y 0
Z 0
Scale
Specify On-screen
X 1
Y 1
Z 1
Rotation
Specify On-screen
Angle
0
Replace existing block
Select
No block selected
Export to file
C:\Users\rhg\Documents\flatshot.dwg
...
Visible lines
Line type: ByLayer
Line color: ByLayer
Display tangential edges
Preserve entity layer
Hidden lines
Display hidden lines
Line type: ByLayer
Line color: ByLayer
Resolution: -1
OK
Cancel

Command Options

Option	Description
Destination	Determines where the block should be placed; **(choose one)**: • **Insert New Block** - inserts the flatshot block as a new block in the current drawing; specify the insertion point, scale, and rotation angle like a regular block; see the Insert (6.17.13) command • **Replace Existing Block** - replaces a block already inserted in the drawing; click the **Select** button to choose the block graphically • **Export to File** - **(save the flatshot block as a DWG file; clickBrowseto specify the folder in which to save it.)**
Visible lines	Sets the linetype and color for the visible lines; choose the linetype and color from the droplists.
Hidden lines	Toggles the display of hidden lines, and sets their properties. Choose the linetype and color from the droplists, and enter a value for the resolution. Resolution specifies the short entity that will be represented by a hidden line. Enter a negative value so that the program will determine this automatically.The range is 0.01 to 0.000000000001.
Display tangential edges	Toggles the display of tangential edges. A tangential edge is the imaginary transition line between two tangent faces; see figure below. In most cases, leave this option off. Tangential edges removed (left); tangential edges displayed (right). (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Preserve Entity Layer	When on, maintains the layers of entities in the block.

Related Commands

Hide (6.16.10) - removes hidden lines from 3D entities. Solprof (6.27.62) - creates a hidden line representation of 3D solids in a layout viewport. Shademode (6.27.29) - sets the shading style for 3D drawings. ViewBase (6.31.3) - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout. ViewExport (6.31.7) - exports the content of drafting views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing. ViewSection (6.31.10) - creates a cross section view based on a standard drafting view generated by the VIEWBASE command in a paper space layout.

Figure 6.980

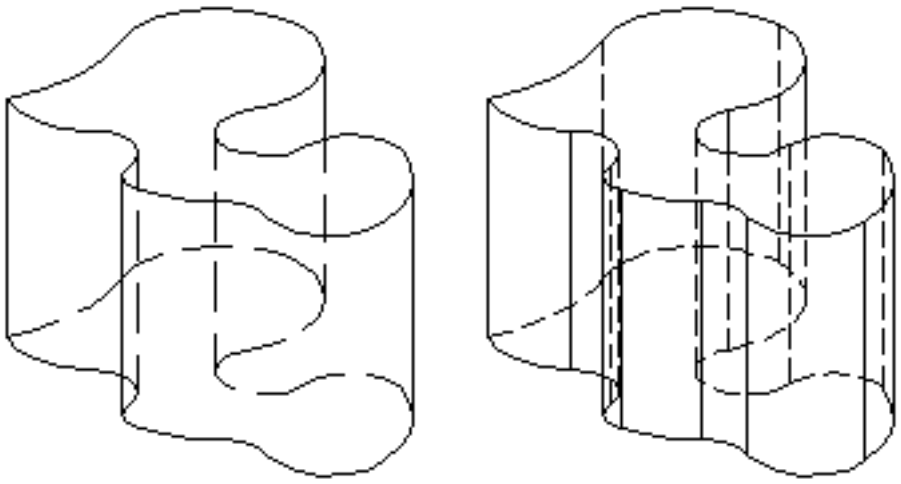


Figure 6.981



6.14.10 Flatten

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Flattens 2D and 3D entities by projecting them onto the XY-plane of the current view.

Accessing the Command

 command bar: **flatten** menu bar: **Modify | Flatten** ribbon:**Change | Modify | Flatten** toolbar: **Modify |**  : **Flatten Prompts you in the command bar:** Select entities [new ucs ELevation/allow EXplode]: **(Choose one or more entities, or else choose an option)** Entities in set: 1 Select entities [new ucs ELevation/allow EXplode]: **(Choose more entities, or else press Enter to end entity selection)** The selected entities are flattened.

Command Options

Option	Description
Select entities to flatten	Choose the entities to be flattened. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ The entities will be projected onto the XY plane of the current view. In top view, the entities will be projected onto the XY-plane of the current coordinate system (WCX or UCS). (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ • Pyramid converted to a square with its diagonals • Cone becomes a circle with a point in the center • 3D polyline and spline are converted to 2D polylines
new ucs ELevation	Specifies the new elevation for the resulting 2D entities; prompts you: **New UCS elevation** - **(enter a value or press Enter to accept the default value.)** The elevation and thickness of the 2D entities can be changed with the Elev (6.13.4) command.
EXplode	Explodes compound entities, such as 3D solids; prompts you: **Allow explode of entities** - **(enter Y or N.)**

Related Commands

Elev (6.13.4) - sets the thickness and elevation of entities. FlatShot (6.14.9) - creates a hidden line representation of all 3D solids in model space as a block or a new drawing. Section (6.27.11) - creates 2D region slices of 3D solid models. SolidEdit (6.27.39) - creates 2D entities from 3D solid models. Solprof (6.27.62) - creates a hidden line representation of 3D solids in a layout viewport.

Figure 6.982

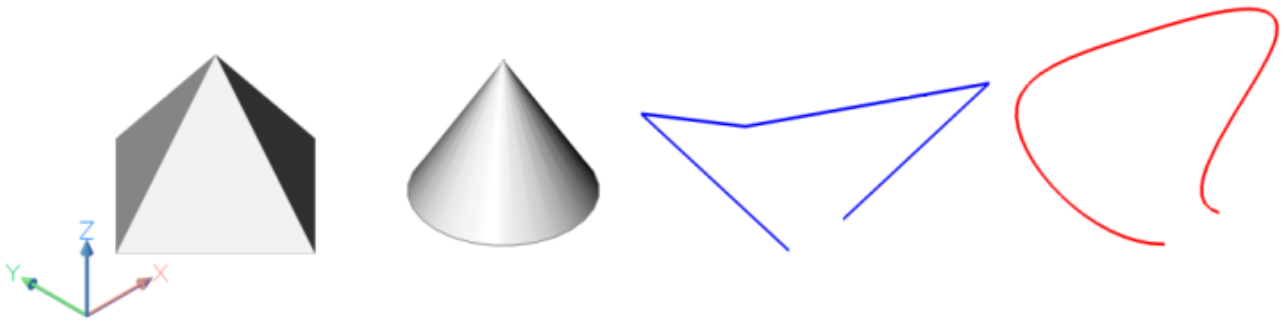
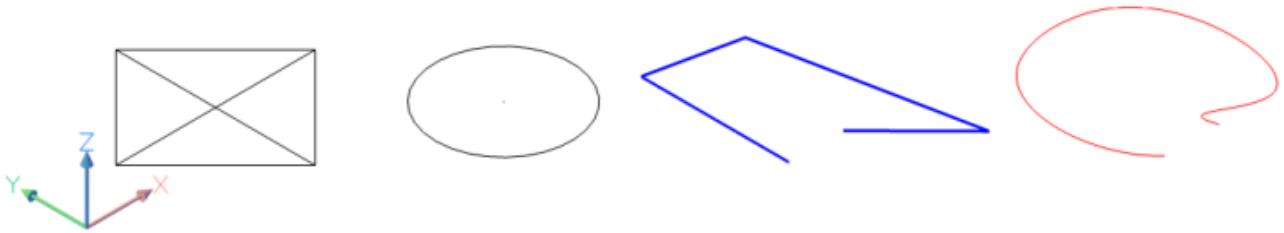


Figure 6.983



6.14.11 From

   Allows you to enter relative coordinates with respect to a reference point whenever a command prompts you to enter a point. **Note:** This command is useful for starting entities offset from another entity, such as inserting a door relative to the end of a wall.

Accessing the Command

 transparent: **from** toolbar: **Entity Snaps** |  : **From Prompts you in the command bar:** Base point: **(Identify the reference point)** Offset or regular point: **(Enter relative coordinates: @ x,y,z)**

Command Options

Option	Description
Base point	Pick a point or enter coordinates to specify the base point
Offset or regular point	Enter the offset by specifying relative coordinates, using the @x,y,z format

Note: From is a command modifier, not a command, and so it is entered only during a drawing or editing command.

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

6.15 G

6.15.1 GcCoincident

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Applies a coincident geometrical constraint to points on two entities. (This command is equivalent to using theCoincident option of the GeomConstraint (6.15.16) command.) **Note:** The second entity that you select is moved or stretched to coincide with the pick point on first entity.

Figure 6.984



Accessing the Command



command bar: **gccoincident** menu bar: **Assembly | 2D Constraints | Coincident** ribbon:**Parametric | 2D Constraints | Coincident** toolbar: **2D Constraints |** : **GcCoincident Prompts you in the command bar:** Select first point or [Entity/Autoconstrain] <Entity>: **(Select a point on an entity, enter an option, or else press Enter to select an entity)** Select point or [Multiple]: **(Select a point, or type M to select multiple points)** Select second point or [Entity] <Entity>: **(Select a point on an entity, or else press Enter to select an entity)**

Command Options

Option	Description
Select first point	Select a valid constraint point on an entity; it is helpful to turn on ESnap to see these constraint points. • **Lines** - endpoints, midpoint • **Arcs and elliptical arcs** - endpoints, center point, midpoint • **Circles and ellipses** - center point • **Polylines** - endpoints, vertices, midpoints of line and arc segments, centers of arc segments • **Splines** - endpoints • **Blocks, xrefs, text, mtext, attributes, tables** - insertion point
Select second point	Select a valid constraint point on another entity.The second point is moved to coincide with the first point.
Entity	Places the constraint on a valid constraint entity: • Line or polyline segment • Circle or ellipse • Arc or elliptical arc • Spline Prompts you: 1. **Select an entity** - **(select an entity; the point you pick on an open entity applies the constraint the end closest to your pick point.)** 2. **Select point or [Multiple]** - **(select another entity.)** The second entity moves or stretches to connect to the first one.
Multiple	Snaps to valid constraint points on additional entities; prompts you: 1. **Select point** - **(pick a point on an entity.)** 2. **Select point** - **(pick a point on another entity, or else press Enter to complete the command.)**

Autoconstrain	Applies the coincident constraint to all entities in a selection set; the entities must be touching. Prompts you: 1. **Select entities** - **(select one or more entities; you can enter all to select all entities in the drawing; the program reports)**: **Entities in set: 3** 2. **Select entities**- **(select more entities, or else press Enter to end entity selection; the program reports)**: **2 coincident constraint(s) applied**

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display them. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en eg-1.png}) Pause the cursor over one of the icons to see the location of the constraint, shown as a large**X**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gcconstraint-eg-2.png}) To remove a constraint, click the tiny **x** on the constraint bar: 

Figure 6.985

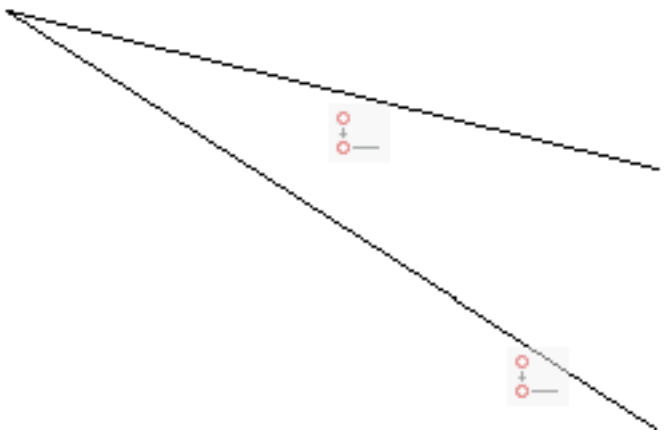
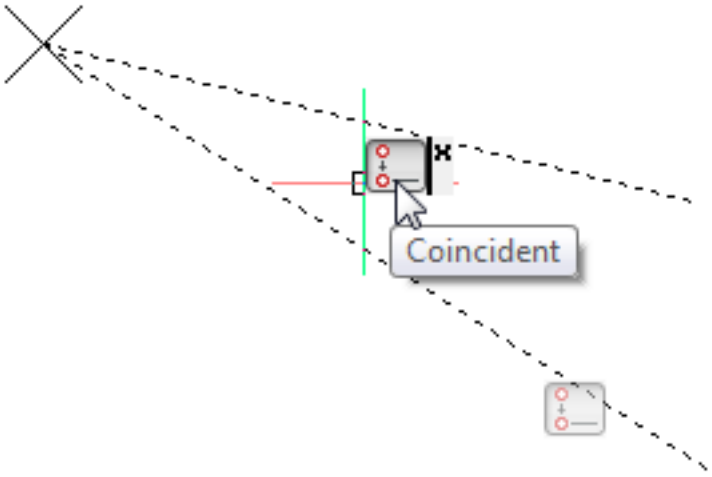


Figure 6.986



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.987



6.15.2 GcCollinear

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains one line or polyline segment to be collinear with another one, which means they lay in the same line. (This command is equivalent to using the Collinear (6.15.16) option of the GeomConstraint (6.15.16) command.) **Note:** The **second** entity you select is straightened out to be collinear to the first.

Accessing the Command

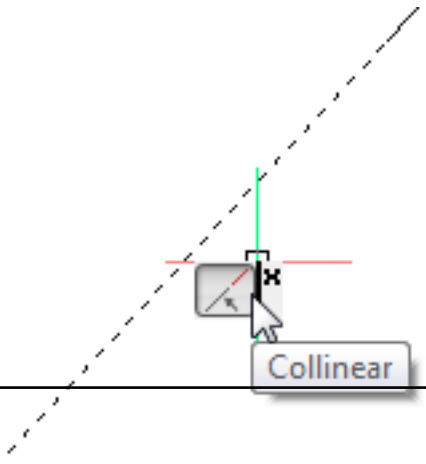
 command bar: **gccollinear** menu bar: **Assembly | 2D Constraints | Collinear** ribbon:**Parametric | 2D Constraints | Collinear** toolbar: **2D Constraints |**  : **GcCollinear Prompts you in the command bar:** Select first entity or [Multiple]: **(Select a line or polyline segment, or press M)** Select second entity: **(Select another line or polyline segment)**

Command Options

Option	Description
Select first entity	Select a line or polyline segment.
Select entity to make collinear to first	Select multiple lines or polyline segments; segments of the same polyline are accepted, which is straightens out the polyline.
Multiple	Makes more than two entities collinear; you can pick only one entity at a time. Prompts you: 1. **Select first entity** - **(pick a line or polyline segment.)** 2. **Select entity to make collinear to first** - **(pick another entity.)** 3. **Select entity to make collinear to first** - **(select more entities, or else press Enter to complete the command.)**

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display them.To remove a constraint, click the tiny **x** on the constraint bar. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gccollinear-x.png})

Figure 6.988



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.989



6.15.3 GcConcentric

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains circles, arcs, polyline arcs, ellipses, and elliptical arcs to have the same center point, making them concentric.(This command is equivalent to using theConcentric option of the GeomConstraint (6.15.16) command.) **Note:** The second entity that you select is moved to be concentric to the first entity.

Accessing the Command

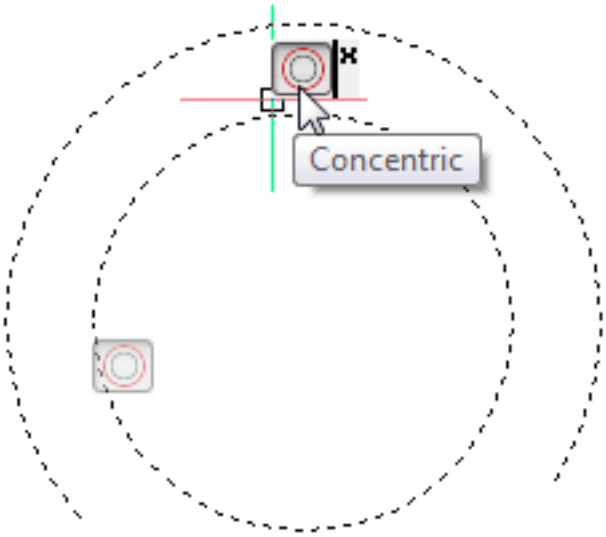
 command bar: **gcconcentric** menu bar: **Assembly| 2D Constraints | Concentric** ribbon:**Parametric | 2D Constraints |** toolbar: **2D Constraints |**  : **GcConcentric Prompts you in the command bar:** Select first entity: **(Select a valid curved entity)** Select second entity: (**Select a second valid curved entity**)

Command Options

Option	Description
Select first entity	Select a valid entity: • Circle • Ellipse • Arc • Polyline arc • Elliptical arc
Select second entity	Select a second valid curved entity; it is moved to be concentric to the first one -- both share the same center point, as shown below.

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display them. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enx.png}) To remove a constraint, click the tiny **x** on the constraint bar.

Figure 6.990



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.991



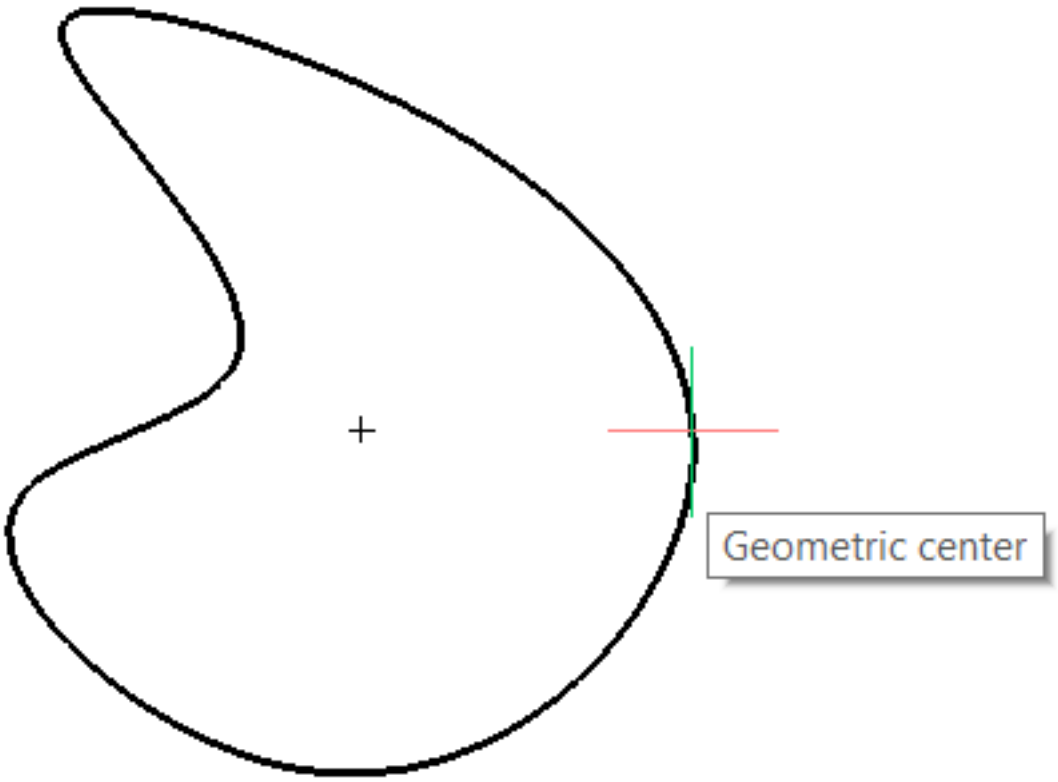
6.15.4 Gcenter

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles Geometric Center Entity snap; snaps to the centroid of a closed polyline, spline, planar 3D polyline, regions, or planar face of a 3D solid.

Accessing the Command

 command bar: **gcenter** transparent: **gce** ribbon:**Home | Entity Snaps | Snap to Geometric Center** toolbar: **Entity Snaps |**  : **Gcenter** Displays no prompts in the command bar; as you move the cursor over a closed entity, displays a blipmark (+) at the geometric center icon. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Ceg.png})

Figure 6.992



Command Options

Option	Description
Gcenter	Entering the command toggles Geometric Center entity snap. When on, it snaps to the centroid of a closed entity or and planar face of a 3D solid. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.993



6.15.5 GcEqual

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains arcs and circles to the same radius, or lines and polyline segments to the same length.(This command is equivalent to using theEqual option of the GeomConstraint (6.15.16) command.)
Note: The second entity that you select is stretched to be the same length (or radius) as the first entity.

Accessing the Command

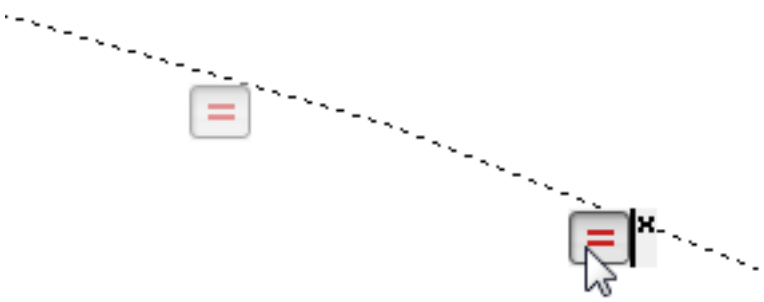
command bar: **gcequal** menu bar: **Assembly| 2D Constraints | Equal** ribbon:**Parametric | 2D Constraints |Equal** toolbar: **2D Constraints |** : **GcEqual Prompts you in the command bar:** Select first entity or [Multiple]: **(Select an entity, or type M)** Select second entity: **(Select a second entity of the same kind -- linear or curved -- as the first entity)**

Command Options

Option	Description
Select first entity	Select a line, arc, circle, or polyline segment.
Select second entity	Selects a second entity, which is made the same length or radius as the first one.
Multiple	Selects more than two entities: 1. **Select first entity** - **(select the initial entity.)** 2. **Select entity to make equal to first** - **(select another entity.)** 3. **Select entity to make equal to first** - **(select another entity, or else press Enter to complete the command.)** All entities selectedmust be of the same kind -- straight or curved -- as the first one. You can select multiple segments of the same polyline.

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display them. To remove a constraint, click the tiny **x** on the constraint bar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gcequal-x.png})

Figure 6.994



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.995



6.15.6 GcFix

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains points or entities to a fixed position. (This command is equivalent to using theFix option of the GeomConstraint (6.15.16) command.) **Note:** This command constrains

- **Point** - one end of the entity is fixed in place; for instance, one end of a line is fixed in place, but the other end can still be moved
- **Entity** - the entire entity is fixed in place and cannot be moved

Accessing the Command

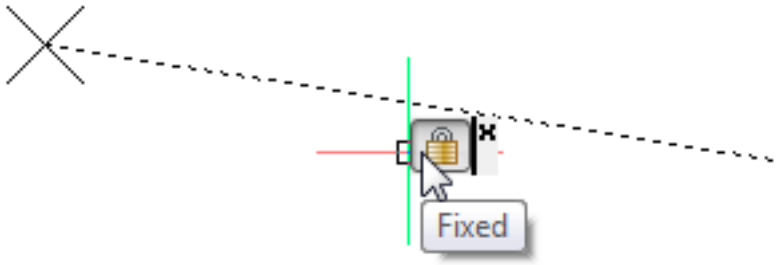
 command bar: **gcfix** menu bar: **Assembly| 2D Constraints | Fix** ribbon:**Parametric | 2D Constraints | Fix** toolbar: **2D Constraints |**  : **GcFix Prompts you in the command bar:** Select point or [Entity]: **(Select a valid constrain point, or else enter E)**

Command Options

Option	Description
Select point	Select a valid constraint point on an entity: • **Lines** - endpoints, midpoint • **Arcs and elliptical arcs** - endpoints, center point, midpoint • **Circles and ellipses** - center point • **Polylines** - endpoints, vertices, midpoints of line and arc segments, centers of arc segments • **Splines** - endpoints • **Blocks, xrefs, text, mtext, attributes, tables** - insertion point The selected point on the entity is fixed in place, and so can no longer be moved.
Entity	Places the constraint on a valid constraint entity: • Line or polyline segment • Circle or ellipse • Arc or elliptical arc • Spline Prompts you: 1. **Select an entity** - **(select an entity; the point you pick on an open entity applies the constraint the end closest to your pick point.)** The **position** of the entity cannot be moved any more, although you can modify the length of, say, a line or polyline segment in either direction.

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display them. Pause the cursor over one of the icons to see the location of the constraint, shown as a large**X**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Command x.png}) To remove a constraint, click the tiny **x** on the constraint bar.

Figure 6.996



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.997



6.15.7 GcHorizontal

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains a line, linear polyline segment, or a pair of points to lie parallel to the X-axis in the current coordinate system.(This command is equivalent to using theHorizontal option of the Geom-Constraint (6.15.16) command.) **Note:** This command moves one end of the segment so that the segment is horizontal. The end that is moved is the end of the segment drawn second.

Accessing the Command

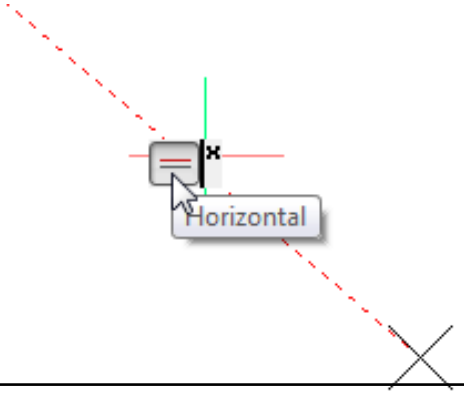
command bar: **gchorizontal** menu bar: **Assembly| 2D Constraints | Horizontal** ribbon:**Parametric | 2D Constraints |Horizontal** toolbar: **2D Constraints |** : **GcHorizontal Prompts you in the command bar:** Select an entity or [2Points] <2Points>: **(Select a line or linear polyline segment, or else press Enter to select two points)**

Command Options

Option	Description
Select an entity	Select a line or a linear polyline segment; the program moves one end to make the entity horizontal -- parallel to the X-axis of the current coordinate system.
2Points	Constrains two pick points to the horizontal; prompts you: 1. **Select first point** - **(pick a point on one entity.)** 2. **Select second point** - **(pick another point on the same entity, or on another entity.)** Pick two valid constraint pointson the same entity or on two different entities. The two selected points are forced to lie on a line which is parallel to the X-axis of the current coordinate system. The program moves the second point vertically so that it lies horizontally to the first pick point; see figure below.

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display the icons. Pause the cursor over one of the icons to see the location of the horizontal constraint, shown by the two large**X**markers: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gchorizontal-2.png}) To remove a constraint, click the tiny **x** on the constraint bar.

Figure 6.998



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.999



6.15.8 GcParallel

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Forces two lines or linear polyline segments to be parallel to each other. (This command is equivalent to using theParallel option of the GeomConstraint (6.15.16) command.) **Note**: The second entity that you select is moved to be parallel with the first entity.

Accessing the Command



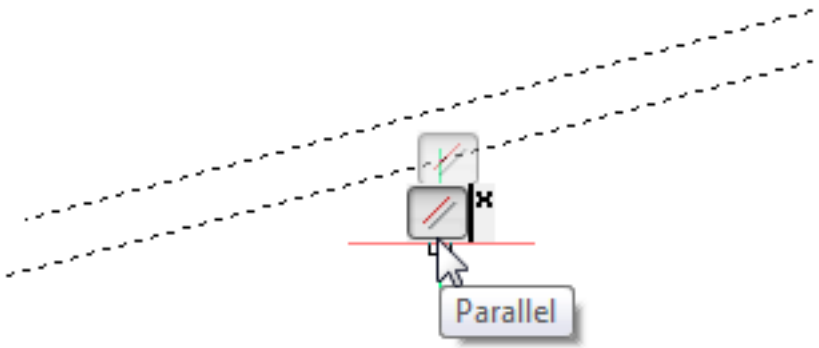
command bar: **gcparallel** menu bar: **Assembly| 2D Constraints | Parallel** ribbon:**Parametric | 2D Constraints | Parallel** toolbar: **2D Constraints |**  : **GcParallel Prompts you in the command bar:** Select first entity: **(Select a line or linear polyline segment)** Select second entity: **(Select a line or linear polyline segment)**

Command Options

Option	Description
Select first entity	Select a line or linear polyline segment.
Select second entity	Select another line or linear polyline segment.

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display them. To remove a constraint, click the tiny **x** on the constraint bar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gcparallel-x.png})

Figure 6.1000



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.1001



6.15.9 GcPerpendicular

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains two lines or polyline segments to lie perpendicularly to each other. (This command is equivalent to using thePerpendicular option of the GeomConstraint (6.15.16) command.) **Note**: The second entity that you select is moved to be parallel with the first entity.

Accessing the Command

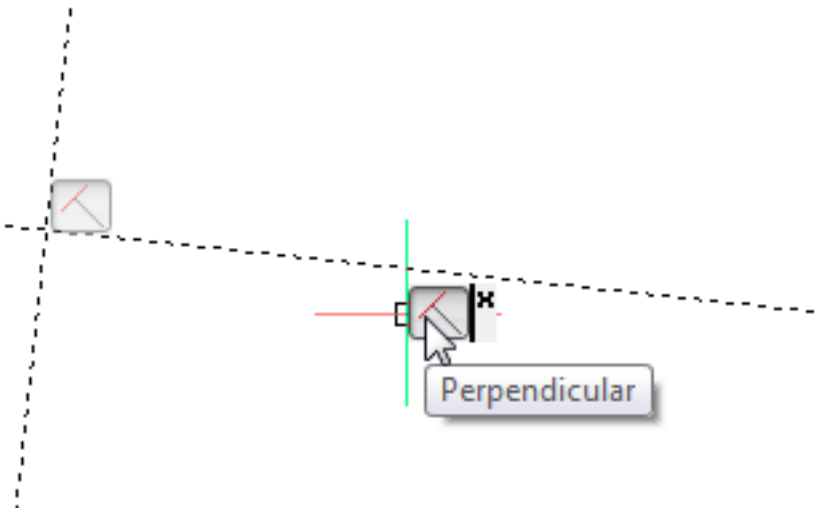

command bar: **gcperpendicular** menu bar: **Assembly| 2D Constraints | Perpendicular** ribbon:**Parametric | 2D Constraints |Perpendicular** toolbar: **2D Constraints |**  : **GcPerpendicular Prompts you in the command bar:** Select first entity: **(Select a line or linear polyline segment)** Select second entity: **(Select another line or linear polyline segment)**

Command Options

Option	Description
Select first entity	Select a line or linear polyline segment.
Select second entity	Select another line or linear polyline segment. You can select two segments of the same polyline.

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display them. To remove a constraint, click the tiny **x** on the constraint bar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gcperpendicular-x.png})

Figure 6.1002



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.1003



6.15.10 GcSmooth

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains two splines to maintain fluid geometric continuity between them. (This command is equivalent to using theSmooth option of the GeomConstraint (6.15.16) command.) **Note:** The second spline that you select is adjusted to smoothly connect to the first spline.

Accessing the Command

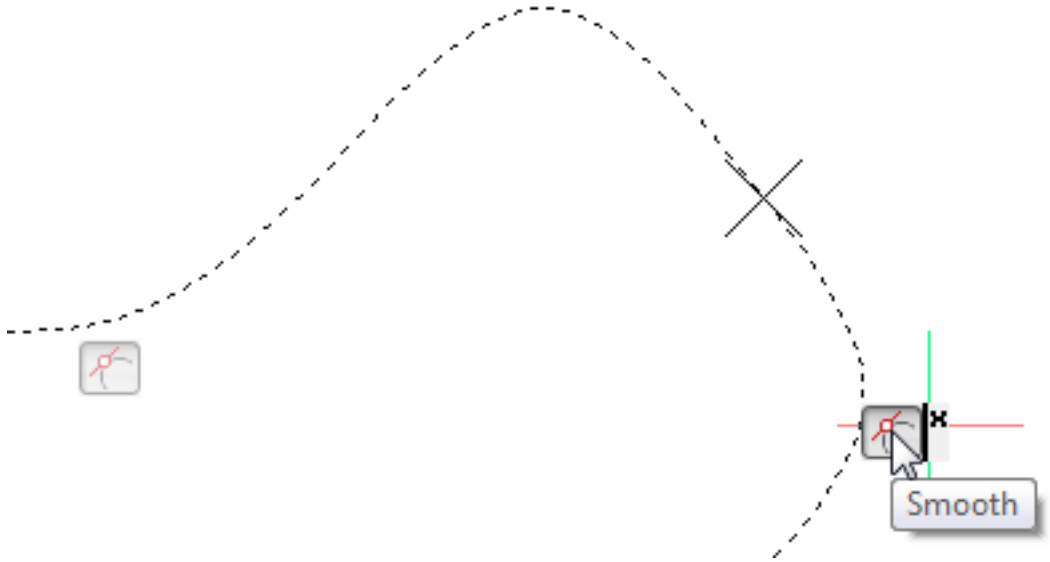
 command bar: **gcsmooth** menu bar: **Assembly| 2D Constraints | Smooth** ribbon:**Parametric | 2D Constraints |Smooth** toolbar: **2D Constraints |**  : **GcSmooth Prompts you in the command bar:** Select first spline curve:**(Select a spline)** Select second entity:**(Select another spline)**

Command Options

Options	Description
Select first entity	Select a spline
Select second entity	Select another spline; this spline bends to meet the first one smoothly; see figure below

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display them. Pause the cursor over one of the icons to see the location of the constraint, shown as a large**X**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Command x.png}) To remove a constraint, click the tiny **x** on the constraint bar.

Figure 6.1004



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity. Resizes 2D and 3D entities.

Figure 6.1005



6.15.11 GcSymmetric

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains two entities or points to lie symmetrically with respect to a selected line. (This command is equivalent to using theSymmetric option of the GeomConstraint (6.15.16) command.) **Note:** The second entity that you select is moved or stretched to be symmetric (mirror-like) to the first entity.

Accessing the Command

 command bar: **gcsymmetric** menu bar: **Assembly| 2D Constraints | Symmetric** ribbon:**Parametric | 2D Constraints |Symmetric** toolbar: **2D Constraints |**  : **GcSymmetric Prompts you in the command bar:** Select first entity or [2Points] <2Points>: **(Select the first entity, or else press Enter to select two points)** Select second entity: **(Select the second entity)** Select symmetry line: (**Select a line)**

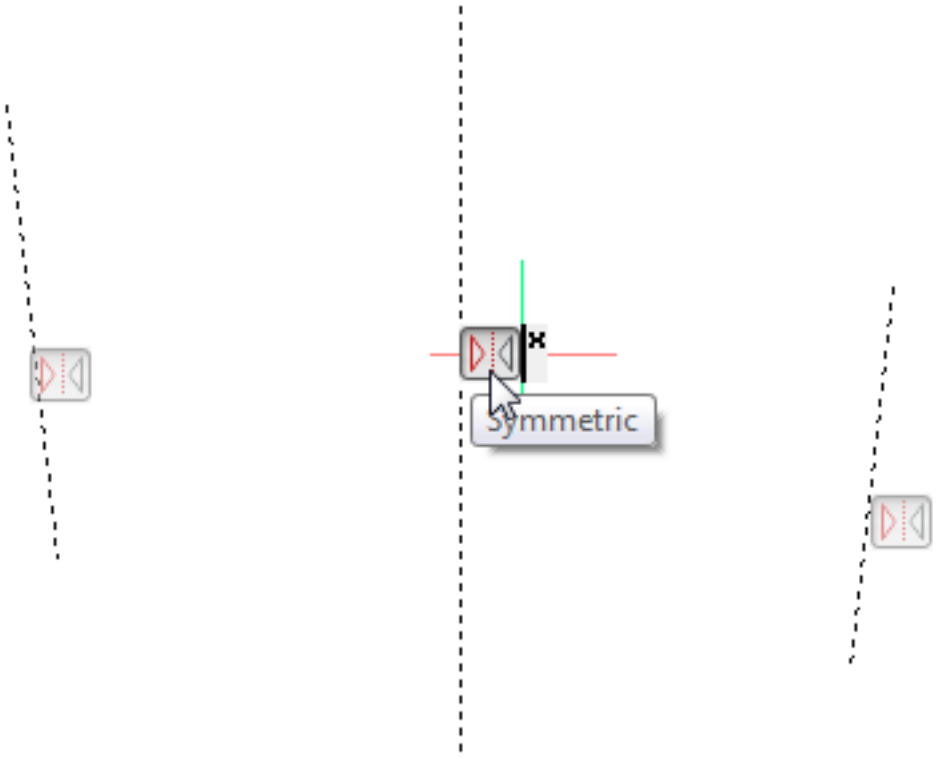
Command Options

Option	Description
Select first entity	Select a valid constraint entity: • Line • Circle • Arc • Ellipse • Elliptical arc
Select second entity	Select another entity of the same type.
Select symmetry line	Select a line, which acts as the line of symmetry to the two entities. The result depends on the entities selected earlier: • **Lines** - the angle of the two entities are made symmetric to the line of symmetry • **Circles**and **arcs** - the center point and radius are made symmetric about the line of symmetry • **Ellipses**and **Elliptical arcs** -the center, major, and minor axes are made symmetricabout the line of symmetry

2Points	Makes two points symmetric about a line of symmetry; prompts you: 1. **Select first point** - **(select a valid point of constraint on an entity)**: Lines - endpoints, midpoint Arcs and elliptical arcs - endpoints, center point, midpoint Circles and ellipses - center point Polylines - endpoints, vertices, midpoints of line and arc segments, centers of arc segments Splines - endpoints Blocks, xrefs, text, mtext, attributes, tables - insertion point 2. **Select second point** - **(select another valid constraint point.)** 3. **Select symmetry line** - **(select a line about which the two points will be made symmetric.)** The second point is moved to lie symmetrically to the first point with respect to the selected line.

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display them. Pause the cursor over one of the icons to see the location of the constraint, shown as a large**X**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commandx.png}) To remove a constraint, click the tiny **x** on the constraint bar.

Figure 6.1006



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.1007



6.15.12 GcTangent

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains two entities to maintain a point of tangency to each other, or to their extensions. (This command is equivalent to using theTangent option of the GeomConstraint (6.15.16) command.) **Note:** The second entity that you select is moved to be tangent with the first entity.

Accessing the Command



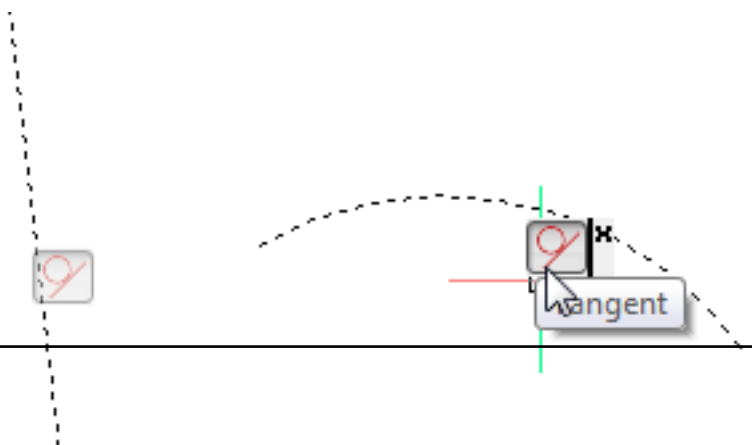
command bar: **gctangent** menu bar: **Assembly| 2D Constraints | Tangent** ribbon:**Parametric | 2D Constraints |Tangent** toolbar: **2D Constraints |**  : **GcTangent Prompts you in the command bar:** Select first entity: **(Select a line, polyline segment, circle, arc, polyarc, ellipse or elliptical arc)** Select second entity: **(Select another circle, arc, ellipse or elliptical arc; or a line or linear polyline segment, if the first object was not a line or polyline segment.)**

Command Options

Option	Command
Select first entity	Select a valid entity: • A linear entity - line or polyline segment • Or a curved entity - circle, arc, polyarc, ellipse, or elliptical arc
Select second entity	Select a second entity: • Linear entities can only be made tangent to curved entities; if you select two lines, the program remarks, "Invalid input" • Curved entities can be made tangent to curved and linear entities The second entity is moved to be tangent to the first, even when they do not touch each other physically; see figure below.

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display them.To remove a constraint, click the tiny **x** on the constraint bar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gctangent-x.png})

Figure 6.1008



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.1009



6.15.13 GcVertical

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Constrains a line, linear polyline segment, or a pair of points to lie parallel to the Y-axis in the current coordinate system. (This command is equivalent to using theVertical option of the GeomConstraint (6.15.16) command.) **Note:** This command moves one end of the segment so that the segment is vertical. The end that is moved is the end of the segment drawn second.

Accessing the Command



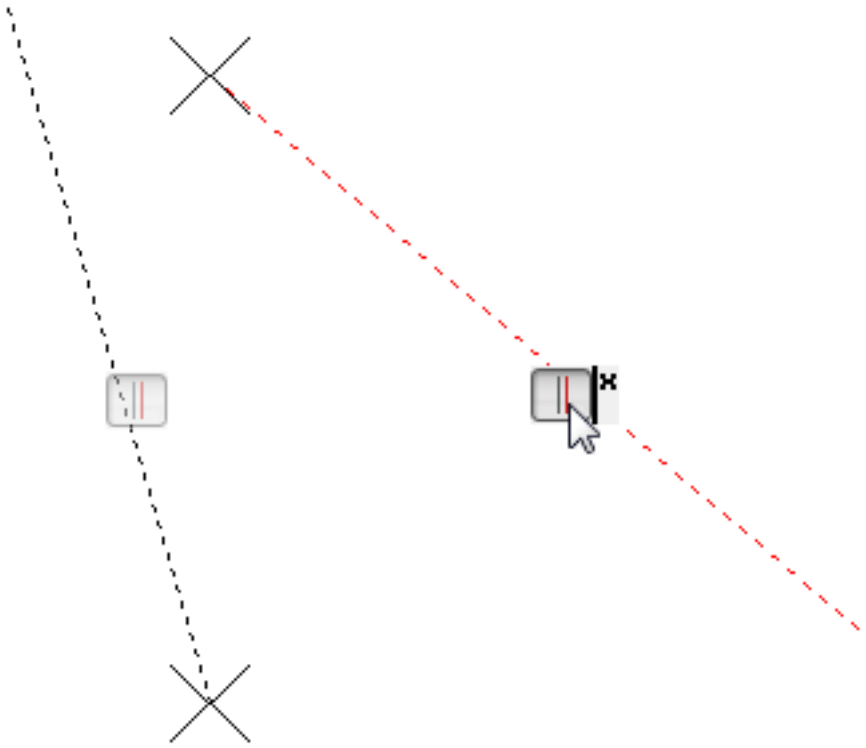
command bar: **gcvertical** menu bar: **Parametric | 2D Constraints | Vertical** ribbon:**Parametric | 2D Constraints |Vertical** toolbar: **2D Constraints |**  : **GcVertical Prompts you in the command bar:** Select an entity or [2Points] <2Points>: **(Select a line or linear polyline segment, or press Enter to select two points on one or two entities)**

Command Options

Option	Description
Select an entity	Select a line or a linear polyline segment; the program moves one end to make the entity vertical -- parallel to the Y-axis of the current coordinate system.
2Points	Constrains two pick points to the vertical; prompts you: 1. **Select first point** - **(pick a point on one entity.)** 2. **Select second point** - **(pick another point on the same entity, or on another entity.)** Pick two valid constraint pointson the same entity or on two different entities. The two selected points are forced to lie on a line which is parallel to the Y-axis of the current coordinate system. The program moves the second point horizontally so that it lies vertical to the first pick point; see figure below.

After you place the constraint, icons appear near the entities indicating the type of constraint. If you do not see them, then use the ConstraintBar (6.9.62) command to display the icons. Pause the cursor over one of the icons to see the location of the vertical constraint, shown by the two large**X**markers: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
2.png}) To remove a constraint, click the tiny **x** on the constraint bar.

Figure 6.1010



Related Commands

GeomConstraint (6.15.16) - applies geometric relationships between entities and/or points on entities. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity.

Figure 6.1011



6.15.14 GenerateBoundary

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Generates a closed polyline or spline around the boundary of a flat, closed area, or the face of a 3D solid, or the area of a hatch pattern.

Accessing the Command

 command bar: **generateboundary** menu bar: **Draw | Boundaries | Generate Boundary** toolbar: **Draw | Boundaries |**  Quad: **2D Editing |**  : **GenerateBoundary Prompts you in the command bar:** Select a planar enclosed area, a planar face, or a hatch to generate a boundary: **(Pick a point inside a closed region, or select a face of a 3D solid, or select a hatches, and press Enter)** Generated boundaries for 1 object(s)

Command Options

Option	Description
Select a planar enclosed point, a planar face, or a hatch to generate a boundary	Pick a point inside one of the following entities: • Closed area • Face on a 3D solid;to select the face of a 3D solid, hold down the **Ctrl** key • Hatch pattern This command does not work with 3D solids that do not have edges, such as spheres and tori. The result of this command: • **Polyline** - when the boundary is polygonal (has corners, like a square) • **Spline** - when the boundary is curved, like a spline. To work with the resulting boundary polyline or spline entity, such as to move or copy it, use **L** (last) selection.

Related Commands

Boundary (6.6.15) - creates a polyline that forms a boundary inside a closed area through a dialog box. -Boundary (6.6.16) - creates a polyline boundary inside closed areas through the command bar. HatchGenerateBoundary (6.16.5) - generates a boundary around a hatch or gradient fill.

Figure 6.1012



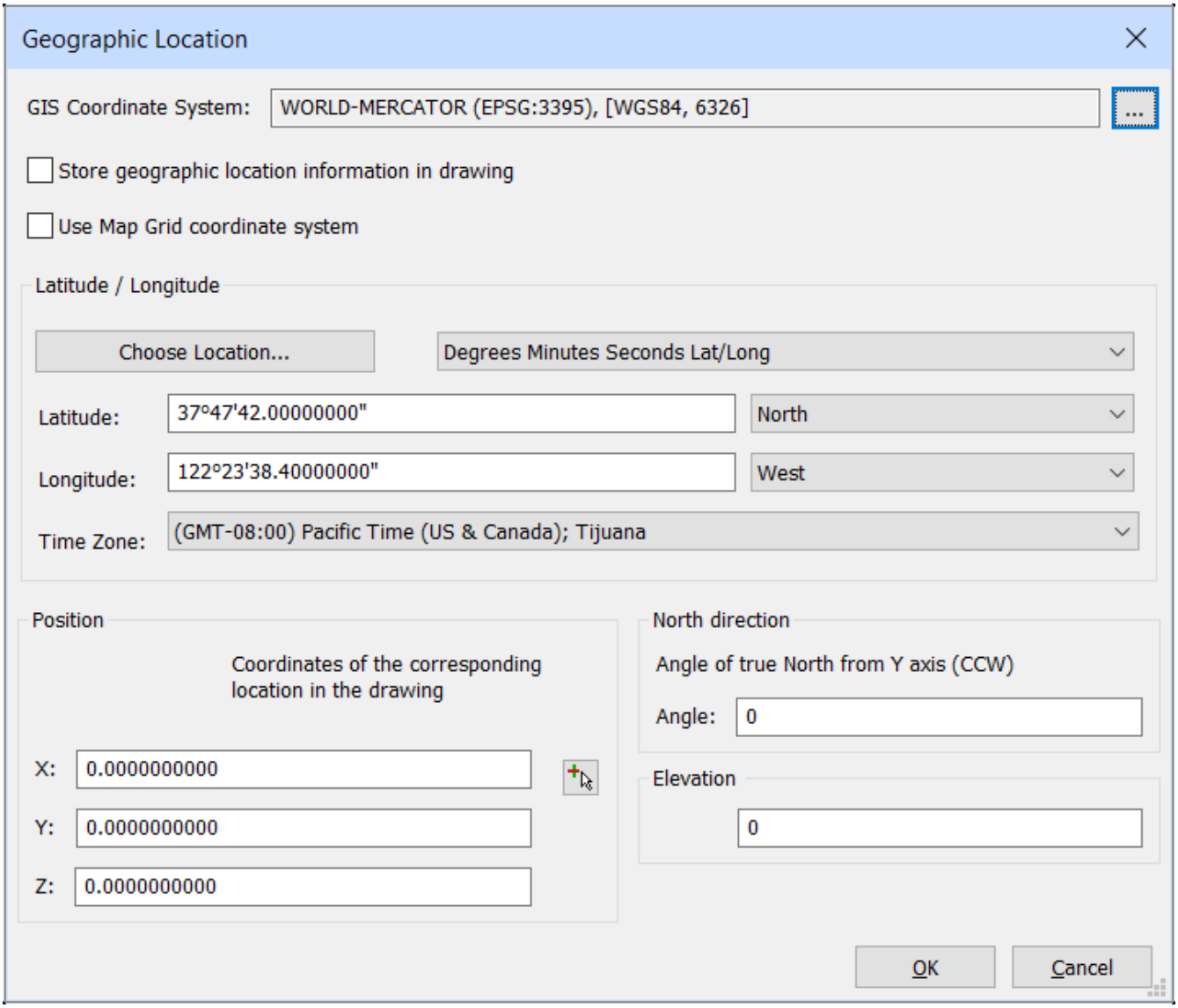
6.15.15 GeographicLocation

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Sets the geographic location of the drawing by applying the longitude and latitude to a point in the drawing. **Note**: This command cannot be used in a unitless drawing; first use the INSUNIT variable to set the units.

Accessing the Command

 command bar: **geographiclocation** alias: **geo** menu bar:**View | Rendering | Geographic Location Insert | Geographic Location** ribbon:**View | Sun | Geographic Location** toolbar:**Render |**  **: geographiclocation** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/geographiclocation-dlg.png}) Enter options, pick a point in the drawing, and then click **OK**. To display geographic coordinates, in the status bar right-click the x,y,z coordinates and then chooseGeographic. Click **Geographic Location System** in the status bar to change the way the coordinate system is represented in the drawing.

Figure 6.1013



Command Options

Option	Description
GIS Coordinate System	Reports the GIS system being used by the drawing. To change the system 1. Click the **Browse** button 2. Choose a system from the droplist (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/browse-dlg.png}) 3. Click **OK**
Store geographic location information in the drawing	When checked, a red dot indicates the position in the drawing; see Position option. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/geeg.png})
Use Map Grid coordinate system	Uses the location data from the GIS Coordinate System, and so grays out all other options in this dialog box.
Choose Location	Displays the Choose Geographic Location dialog box: 1. Choose a city from the droplist 2. Optionally, also choose a time zone 3. Click **OK** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/chooselocation-dlg.png}) 4. The program asks you if it is okay to change to the new time zone.Click **Yes** or **No**. You can also set the time zone independently in the Time Zone droplist; see below.
Location Format	Choose a format for the coordinates from the droplist: • Degrees Minutes Seconds • Decimal degrees
Latitude	Sets the latitude in either Degrees/Min./Sec. or decimal format. The valid range is 0 to 90. Choose **North** or **South** from the latitude option button. The value is saved to the LATITUDE (1.13.11) system variable.
Longitude	Sets the longitude of the drawing in either Degrees/Min./Sec. or decimal format. The valid range is 0 to 180. Choose **East** or **West** from the longitude option button. The value is saved to the LONGITUDE (1.13.11) system variable.
Timezone	Choose a time zone from the Time Zone droplist: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/getimezones-dlg.png}) The setting is saved to the TIMEZONE (1.13.11) system variable.

Position	Specifies the position of the location in the drawing: • **X, Y Z** - **(enter the coordinates; the z is optional and can remain 0.)** • **Pick Point**  - **(click the button, and then pick a point in the drawing.)**
North Direction: Angle	Specifies the angle of the sun from north, in the context of the world coordinate system.The setting is saved to the NORTHDIRECTION (1.13.11) system variable.
North Direction: Elevation	Enter an elevation for the long-lat point; the value can be positive or negative.

Procedures

- Managing geographic datasets (1.15.5)

Related Commands

Render (6.26.22) - calculates a photorealistic image of the 3D model. Sunproperties (6.27.57) - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog. MapConnect (6.21.3) - allows to set up a connection with a Web Map Service through a dialog box.

Figure 6.1014

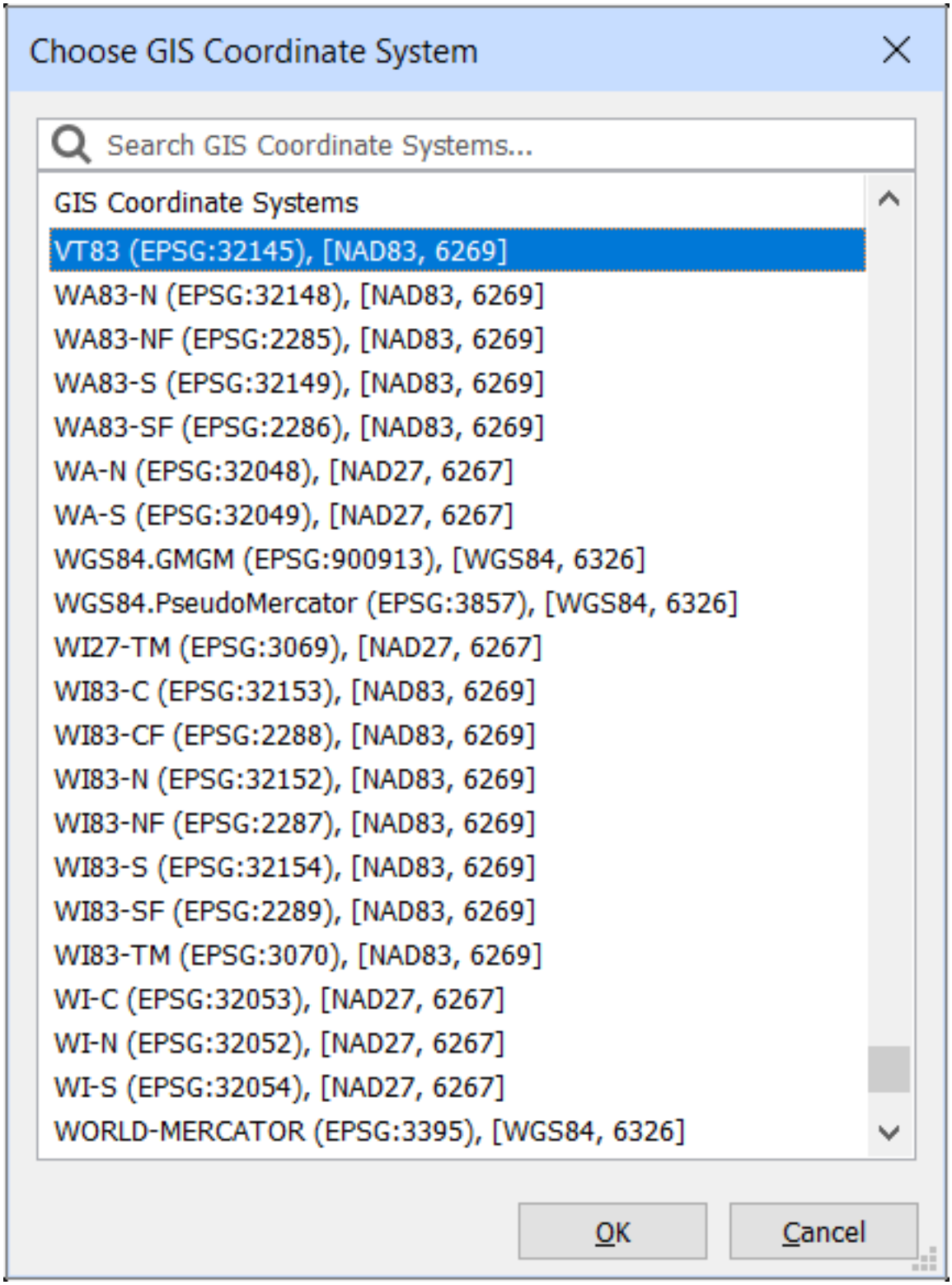


Figure 6.1015



Figure 6.1016

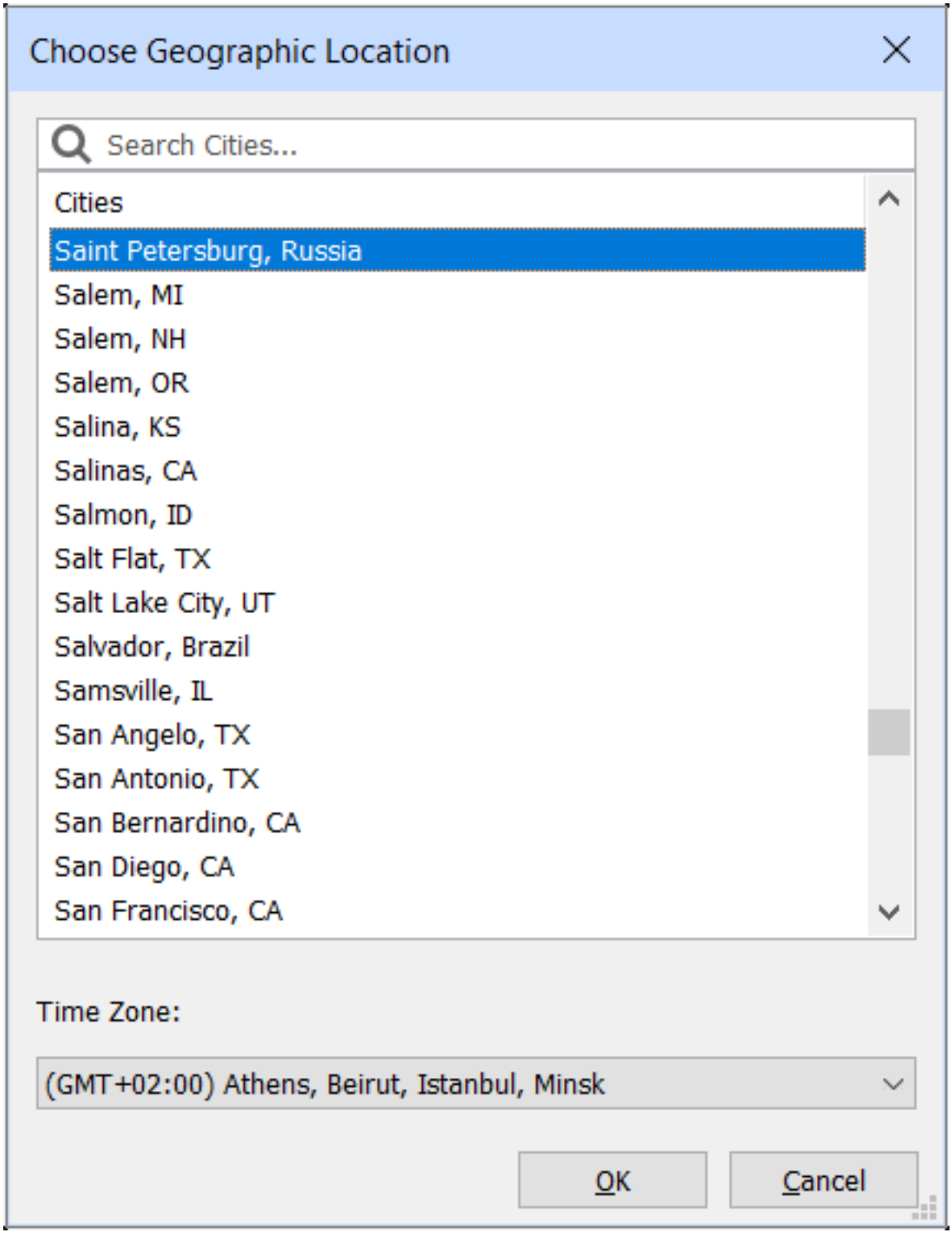


Figure 6.1017



Figure 6.1018



6.15.16 GeomConstraint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Applies geometric relationships between entities, on entities, and on valid constraint points. Constraints keep entities in a fixed position, such as perpendicularly or vertically. **Note:** Geometric constraints can be applied to the following entities and their constraint points:

Entity Type	Valid Constraint Points
Lines	Endpoints, midpoint
Arcs, elliptical arcs	Endpoints, center point, midpoint
Circles, ellipses	Center point
Polyline segments	Endpoints, vertices, midpoints
Polyline arcs	Endpoints, vertices, midpoints, center points
Splines	Endpoints
Inserted entities: blocks, xrefs, text, mtext, attributes, tables	Insertion points

Accessing the Command

command bar: **geomconstraint** : **GeomConstraint Prompts you in the command bar:** Enter constraint type [Horizontal / Vertical / Perpendicular / PArallel / Tangent / SMooth / Coincident / CONcentric / COLlinear / Symmetric / Equal / Fix] <Coincident>: **(Enter an option).**

Command Options

Option	Description
Horizontal	Constrains entities or pairs of points to lie parallel to the X-axis of the current coordinate system. See the GcHorizontal (6.15.7)command.
Vertical	Constrains entities or pairs of points to lie parallel to the Y-axis of the current coordinate system. See the GcVertical (6.15.13)command.
Perpendicular	Constrains two entities to lie perpendicularly to each other. See the GcPerpendicular (6.15.9)command.
PArallel	Forces two entities to be parallel to one other. See the GcParallel (6.15.8)command.
Tangent	Constrains two entities to maintain a point of tangency to each other or their extensions.See the GcTangent (6.15.12)command.
SMooth	Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. See theGcSmooth (6.15.10)command.
Coincident	Applies a coincident geometrical constraint to two points or constrains a point to an entity. Seethe GcCoincident (6.15.1)command.
CONcentric	Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide .See the GcConcentric (6.15.3)command.
COLinear	Forces entities to be collinear. See the GcCollinear (6.15.2)command.
Symmetric	Constrains two entities or points to lie symmetrically with respect to a selected line. See the GcSymmetric

Equal 	Constrains circularentities to the same radius, or linear entities to the same length. See the GcEqual (6.15.5)command.
Fix 	Constrains points and entities at a fixed position. See the GcFix (6.15.6)command.

Related Commands

Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. GcCoincident (6.15.1) - applies a coincident geometrical constraint to two points or constrains a point to an entity. GcCollinear (6.15.2) - forces lines to be collinear. GcConcentric (6.15.3) - constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. GcEqual (6.15.5) - constrains arcs and circles to the same radius, or lines to the same length. GcFix (6.15.6) - constrains points and entities at a fixed position. GcHorizontal (6.15.7) - constrains lines or pairs of points to lie parallel to the X-axis of the current coordinate system. GcParallel (6.15.8) - forces two lines to be parallel to each other. GcPerpendicular (6.15.9) - constrains two lines to lie perpendicularly to each other. GcSmooth (6.15.10) - forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. GcSymmetric (6.15.11) - constrains two entities or points to lie symmetrically with respect to a selected line. GcTangent (6.15.12) - constrains two entities to maintain a point of tangency to each other or their extensions. GcVertical (6.15.13) - constrains lines or pairs of points to lie parallel to the Y-axis of the current coordinate system.

Figure 6.1019



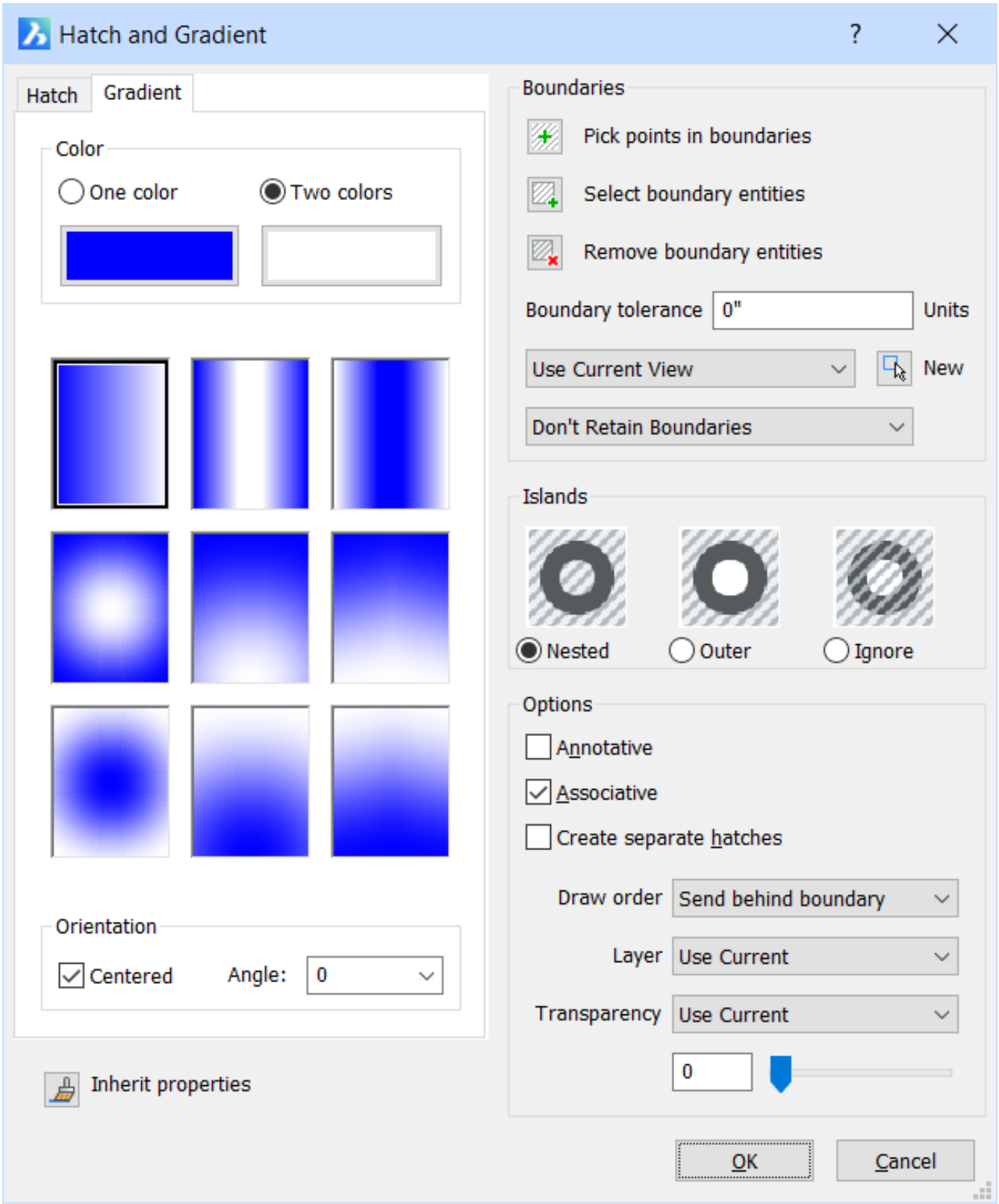
6.15.17 Gradient

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Fills closed areas with solid fills in gradients of one or two colors in a variety of patterns; displays the Hatch and Gradient dialog box. 

Accessing the Command

command bar: **Gradient** menu bar: **Draw | Gradient** toolbar: **Draw |**  : **Gradient** Displays the Gradient tab of a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gradient-dlg.png}) Choose options, and then click **OK**. The program flood fills the selected area with the gradient effect.

Figure 6.1020



Command Options

Color Options	Description
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/GradientColor1.png})	**One color** - click the colored tile to choose a different color, then move the slider to change the underlying color between white, the selected color, and black.
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/GradientColor2.png})	**Two colors** - click the colored tiles to choose different colors.
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/GradientName.png})	Click a gradient fill pattern: name: 1. Linear 2. Cylindrical 3. Inverted Cylindrical 4. Spherical 5. Hemispherical 6. Curved 7. Inverted Spherical 8. Inverted Hemispherical 9. Inverted Curved
Orientation	**Orientation**- determines how the fill is placed in the boundary: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ • **Centered** - centers gradient fill in the area to be filled; when off, starts the fill from the upper left of theboundary. • **Angle**- select an angle of the gradient fill, relative to the current UCS; choose from increments on 15 degrees, or else enter any other value for the angle.

Boundaries

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gra boundaries-dlg.png}) **Pick points in boundaries** - selects closed areas (boundaries) in which to place the gradient pattern; temporarily dismisses the dialog box, and then prompts you at the command bar: ”Select a point to define a boundary or hatch area.” Notice that the program immediately draws the gradient so that you can preview its colors, angle, and origin. When done selecting areas, press Enter. You cannot pick areas that already contains patterns or areas with gaps larger than the boundary tolerance. **Select boundary entities** - selects entities making up a pattern boundary; temporarily dismisses the dialog box, and then prompts you at the command bar ”Select entities.” You can fill areas already containing hatching. **Remove boundary entities** - prompts you to remove entities from the detected boundary set. **Boundary tolerance** - specifies how large a gap can be for this command to fill an open boundary.

- **0** - (Default) The tolerance is set by the application, based on the current view size.When zoomed in closely, boundary detection will fail; when zooming out further so the contour ’looks’ closed, the boundary will become detected
- **Any Value** - Defines the maximum gap in drawing units.

The value is saved in the HPGAPTOL (1.13.11) system variable. Specifies where BricsCAD should search for entities that make up the hatch boundary:

- **Current Viewport** - searches all entities in the current viewport.
- **Existing Set** - searches only the current selection set for possible boundaries.

New - creates a new selection set of entities making up the boundary; prompts you at the command bar, ”Select entities”. **Boundary retention** - determines how boundaries are handled:

- **Don’t retain boundaries** - erases the temporary boundary after the command is finished
- **Retain boundaries as polylines** - converts the boundary into a polyline entity, which can be used afterwards for other purposes
- **Retain boundaries as regions**- converts the boundary into a region entity

Islands

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gra islands-dlg.png}) **Nested** - alternating areas are filled. **Outer**- only the outermost area is filled. **Ignore** - all outer and interior areas of filled.

Options

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gra options-dlg.png}) **Annotative** - toggles the annotative property of fills:

- **On** - annotative, so that the fill changes its scale automatically when the annotative scale factor changes
- **Off** - not annotative, so that the fill retains its scale factor, no matter how the annotative scale factor changes

Associative - toggles the associativity property of fills:

- **On** - associative, so that the fill updates itself automatically when the boundary changes
- **Off** - not associative, so that the fill retains is area when the boundary is changed

The associative and annotativeoptions are mutually exclusive: when one is on, the other is turned off. **Create separate hatches** - creates separate gradient entities for each region in the selection set. **Draw Order** - specifies where the fill should appear relative to visually overlapping entities:

- **None** - draw order is ignored.
- **Send to Back** - fill is behind (underneath) overlapping entities.
- **Send to Front** - fill is in front (on top) of overlapping entities.
- **Send Behind Boundary** - fill is placed underneath the boundary.
- **Bring in Front of Boundary** - fill is placed in on top of the boundary.

Layer - specifies the layer onto which to place the fill:

- **Use Current** - places the fill on the current layer
- **Other layer names** - selects one of the drawing's layers

Transparency - specifies how transparent the fill should be (saved in the HPTRANSPARENCY (1.13.11) system variable):

- **Use Current** - uses the level of transparencyspecified by the CETRANSPARENCY (1.13.11) system variable.
- **By Layer** - use the transparency specified by the layer onto which the fill will be placed
- **By Block** - use the transparency specified by the block of which the fill belongs
- **Specify Value** - use the slider to specify a value between 0 (fully opaque) to 90 (nearly fully transparent)

Inherit properties	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gra inerhentproperties-dlg.png}) **Inherit properties** - copies properties of another gradient in the current drawing; prompts you at the command bar, "Select hatch entity to copy properties from." This button is useful for quickly setting options of a new fills based on existing fills.

Procedures

- General procedure to create hatches and gradient fills (1.17.28)
- To define gradient fill properties (1.17.28)
- To define the boundary for a hatch or gradient fill (1.17.28)
- To edit a hatch or gradient fill (1.17.29)
- Defining the MAXHATCH setting (1.17.26)

Grips Editing

Gradient fills can be edited directly through grips:

1. Select the hatch. Notice that it has one grip.
2. Drag the grip to move the hatch.

Related Commands

Hatch (6.16.1) - places hatch patterns through a dialog box. -BHatch (6.6.5) and -Hatch (6.16.2) - places hatch patterns through the command bar. Boundary (6.6.15) and -Boundary (6.6.16) place a polyline boundary inside closed areas. HatchEdit (6.16.3) - edits hatches and gradient fills in a dialog box. -HatchEdit (6.16.4) - edits the boundaries of hatch and gradient entities; displays prompts in the command bar. Properties (6.24.49) - changes the properties of hatch patterns. Settings (6.27.23) - specifies the default parameters for hatch patterns. Solid (6.27.37) - draws 3- and 4-side solid-filled areas.

Figure 6.1021

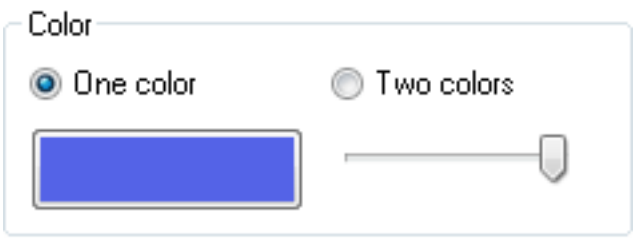


Figure 6.1022



Figure 6.1023

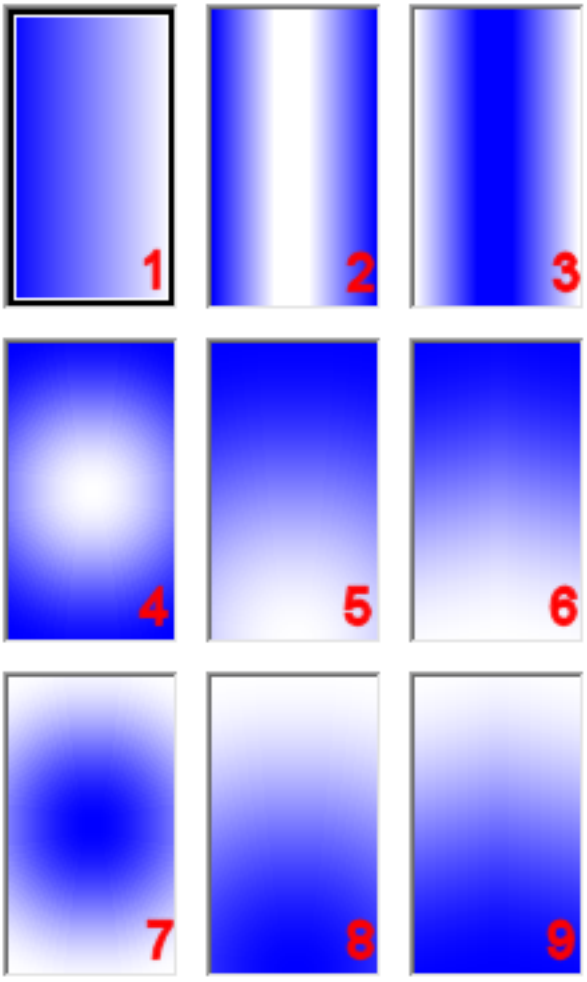


Figure 6.1024

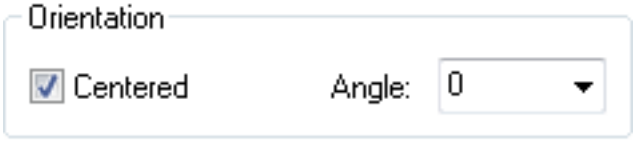


Figure 6.1025

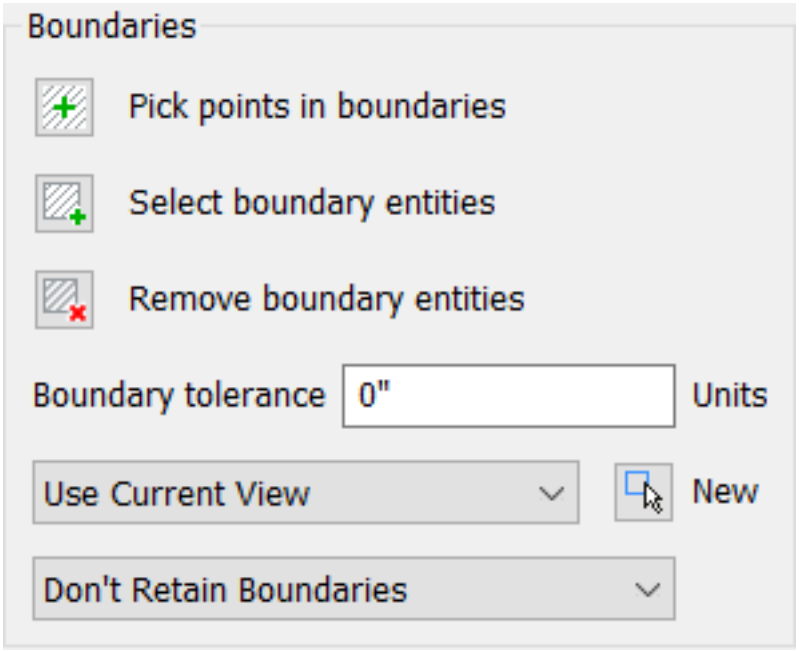


Figure 6.1026

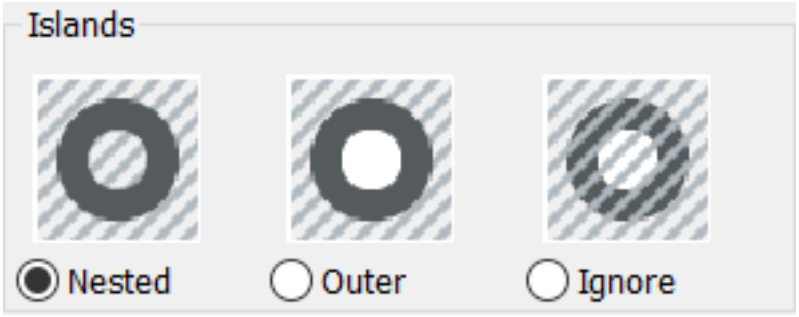


Figure 6.1027

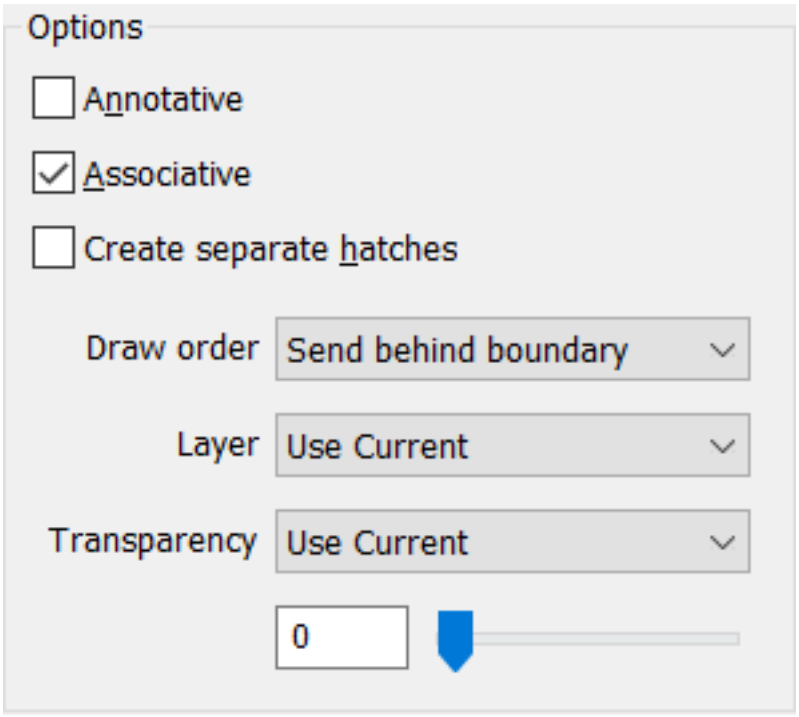


Figure 6.1028

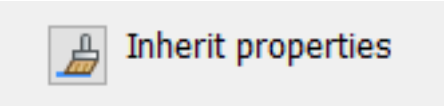


Figure 6.1029



6.15.18 GradientBkgOff

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Switches off the gradient background color in the current viewport off, restoring the default view background. **Note:** To turn on the background gradient color, use the GradientBkgOn (6.15.19) command.

Accessing the Command

command bar: **gradientbkgoff** : **GradientBkgOff** Displays no prompts in the command bar. Removes the background gradient colors.

Command Options

No Options

Related Commands

Background (6.6.1) - sets the model space background for the current viewport and saved views. GradientBkgOn (6.15.19) - restores the default gradient background of the current viewport. View (6.31.1) - creates and sets named views through the Drawing Explorer. -View (6.31.2) - creates and sets named views through the command bar.

Figure 6.1030



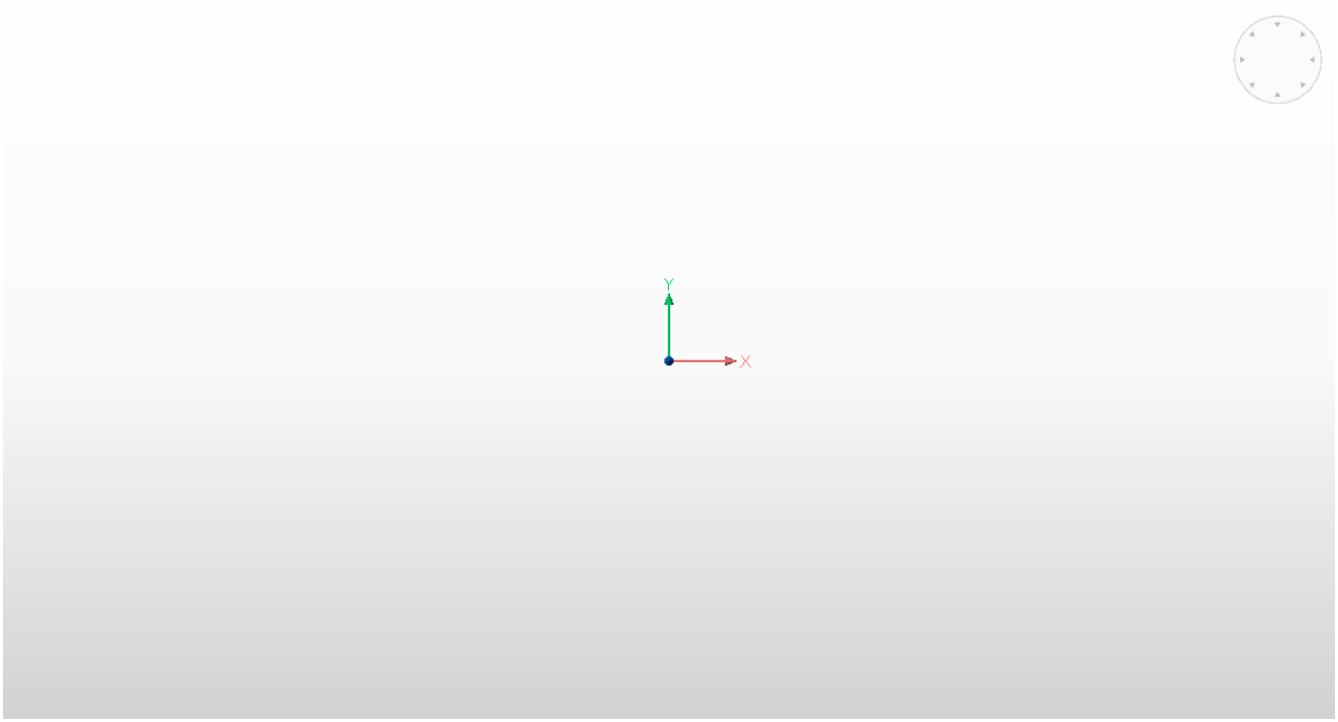
6.15.19 GradientBkgOn

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Restores the default gradient background of the current viewport. **Note:** This command works only when the visual style is not ”2D Wireframe.”

Accessing the Command

command bar: **gradientbkgon** : **GradientBkgOn** Displays no prompts in the command bar, but turns on the background gradient colors. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/gradientbkon-eg.png})

Figure 6.1031



Command Options

No options.To establish the colors of the background gradient, use the following variables:

- **GradientColorTop** -sets the color for the top 1/3 of the gradient background
- **GradientColorMiddle** - sets the color for the middle 1/3 of the gradient background
- **GradientColorBottom**- sets the color for the lower 1/3 of the gradient background

After changing the colors, re-run the GradientBkgOn command.

Related Commands

Background (6.6.1) - sets the model space background for the current viewport and saved views. GradientBkgOff (6.15.18) - switches the gradient background color of the current viewport off. Restores the default view background. View (6.31.1) - creates and sets named views through the Drawing Explorer. -View (6.31.2) - creates and sets named views through the command bar.

Figure 6.1032



6.15.20 GraphScr

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Switches from the programs Text Prompt window to the drawing window (short for ”graphics screen”).

Accessing the Command

command bar: **graphscr** transparent: **'graphscr** keyboard shortcut: **F2** : **GraphScr** Displays no prompt in the command bar; brings the drawing window (graphics screen) to the foreground.

Command Options

No options.

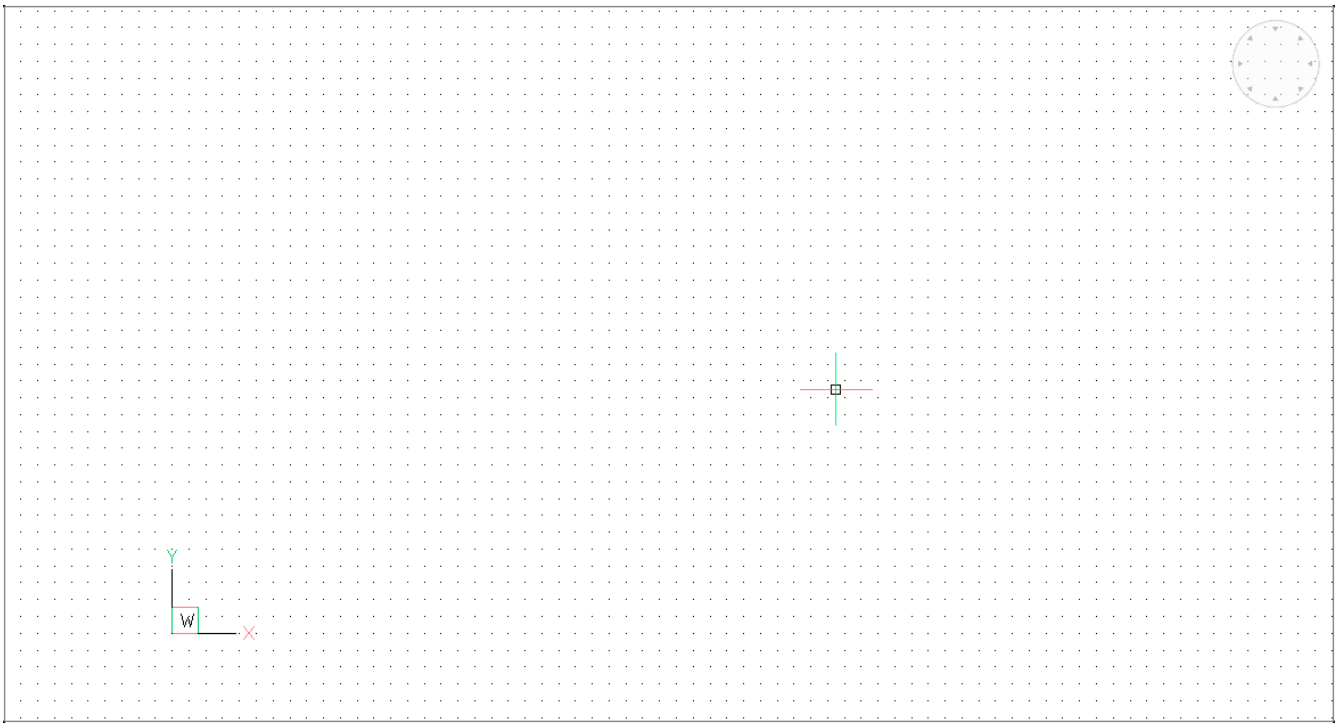
Related Commands

TextScr (6.29.13) - brings the text window to the foreground.

Figure 6.1033



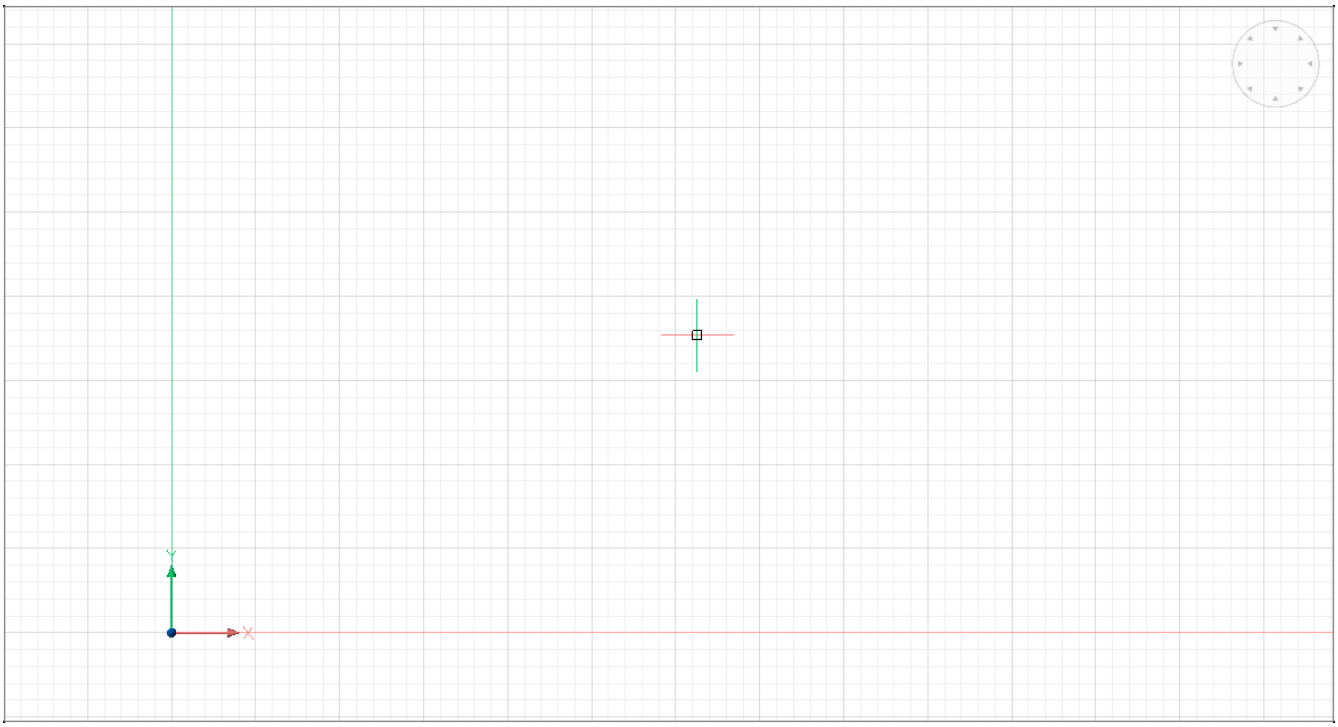
Figure 6.1034



6.15.21 Grid

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the display of the grid, and sets some of its properties. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/grid-dots-eg.png}) **Note:**When the grid is on, it is made of dots or lines in model, paper, and block editor spaces, but is always displayed as lines when the visual style is other than ”2D Wireframe” and always displayed as dots when the grid is rotated. In additions, the grid lines emanating from the origin are usually colored red in the x direction and green in the y direction. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/grid-lines-eg.png})

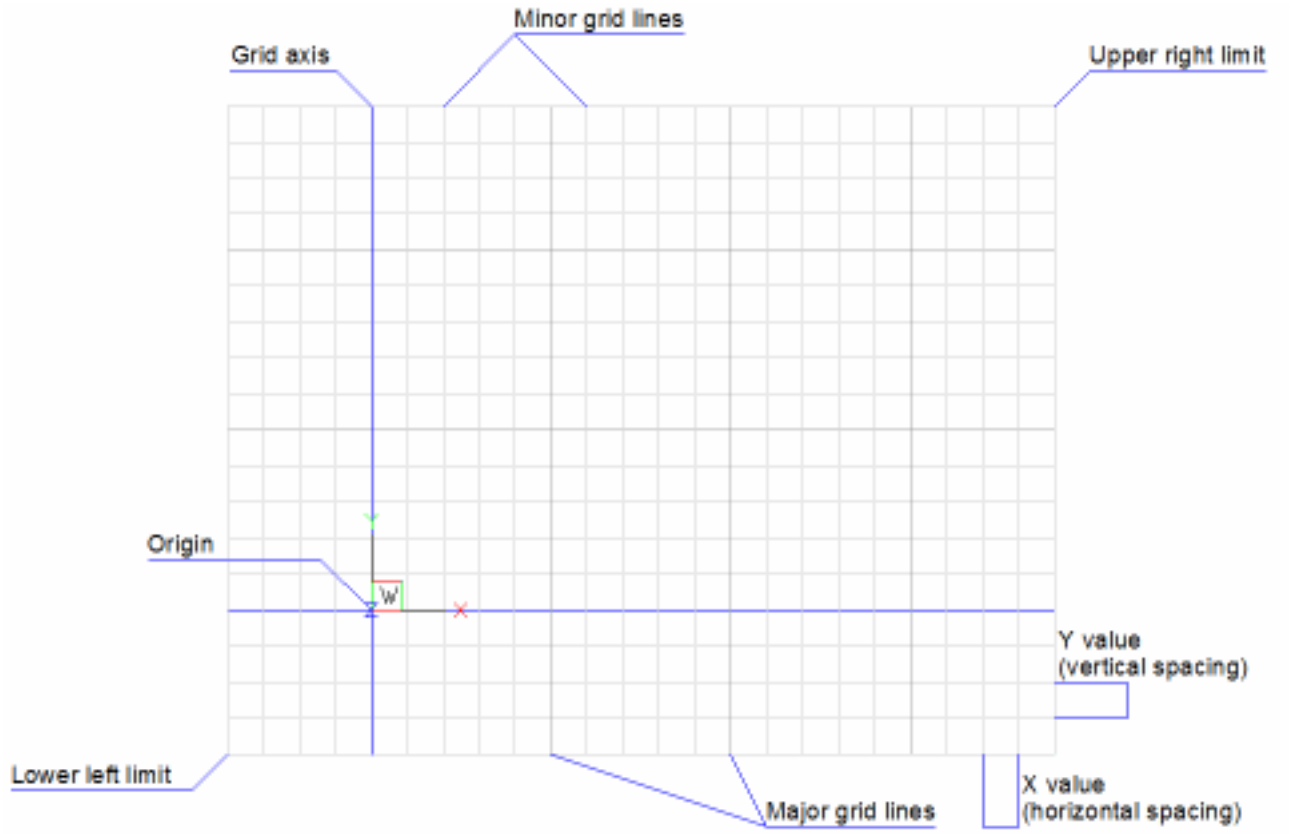
Figure 6.1035



Accessing the Command

 command bar: **grid** transparent: **'grid** alias: **g** keyboard shortcuts: **F7** **Gtrl+G** menu bar: **Settings | Reference Grid** ribbon:**Settings | Reference Grid** status bar: **GRID** toolbar: **Settings |**  : **Grid Prompts you in the command bar:** Grid is off. Enter grid spacing or [grid ON/sync to Snap/Aspect] <10>: **(Enter an option)** (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image204.png}) The extents of the grid are controlled by the Limits (6.20.30) command when the **Display beyond limits area** setting of the GRIDDISPLAY (1.13.11)variable is turned off.

Figure 6.1036



Command Options

Option	Description
Enter grid spacing	Specifies the spacing between grid lines or dots; makes the x and y grid spacing the same; enter a positive number larger than 0.
grid OFF **or** grid ON	Turns the display of the grid off and on, respectively.
Sync to snap	Synchronizes the grid spacing with the current snap spacing, so that the two are the same. The grid spacing stays remains synced to the current snap spacing until you use the Grid Spacing or Aspectoptions to make the grid spacing different from the snap spacing. When SNAPTYPE = 2, adaptive grid snap (1.15.12) is used and SNAPMODE is denied.
Aspect	Sets the aspect ratio of the grid; prompts you: 1. **Horizontal Spacing- (specify the x distance between grid lines or dots.)** 2. **Vertical Spacing- (specify the y distance between grid lines or dots.)** To change the grid's look to isometric, use theSnap (6.27.38)command's Style option.

Procedures

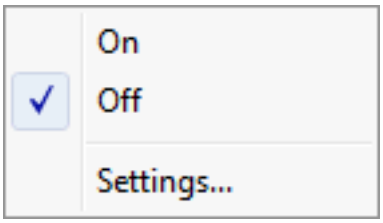
Using snap and grid (1.15.11)

Status Bar Menu

Right-click **GRID** on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help
~~(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image791.png})~~

Shortcut Menu	Description

Figure 6.1038



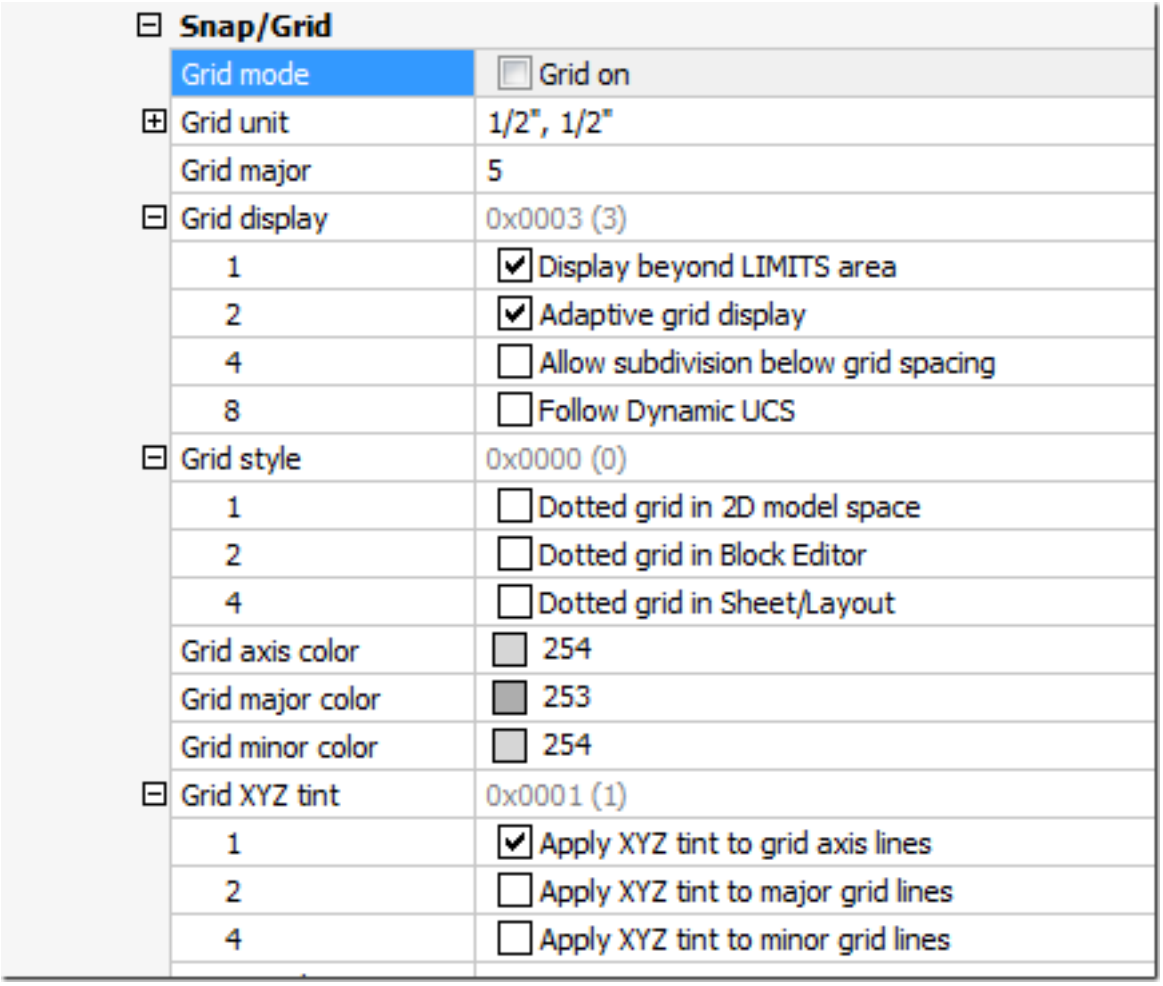
Settings Command: Snap/Grid Section

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/grid-settings.png})

Setting	Description
Grid mode	Toggles grid display: • **On** - turns on grid display • **Off** - turns off grid display
Grid unit	Specifies the distance between grid lines or dots in the drawings's units: • **X** - spacing between dots or lines in the x direction • **Y** - spacing between dots lines in the y direction There is no grid in the z direction.
Grid Major	Specifies the frequency of major versus minor grid lines; enter a number.
Grid Display	Specifies how the grid is displayed: • **Display beyond limits area** - displays the grid all over the graphic screen. Whenthis option is off, the grid's display is limited to the area defined by the LIMMIN (1.13.11) and LIMMAX (1.13.11) system variables, set by the Limits (6.20.30) command. • **Adaptive grid display** - displays fewer grid lines when the grid spacing is too small to display. When this option is off, the program displays a message in the command window when it is not possible to display the grid properly. • **Allow subdivision below grid spacing** - displays more grid lines as you zoom in. When this option is off, the distance between the minor grid lines equals the current grid spacing. • **Follow dynamic UCS** - the grid displays in the XY-plane of the dynamic UCS. When this option is off, the program displays the grid only in the XY-plane of the current UCS.

Grid Style	Specifies how the program displays the grid; when on, the grid is displayed as dots; when off, the grid is displayed as lines. • **Dotted grid in 2D model space** - dots are display in model space when the visual style is set to 2D Wireframe. In all other view modes the grid always displays with lines (see the Shademode (6.27.29) command). • **Dotted grid in Block Editor** -dots are displayed in the block editor environment • **Dotted grid in Sheet/Layout** - dots are displayed in paper space layouts.
Grid color	Specifies the color of the grid axis lines and the major and minor grid lines: • **Grid axis color** - specifies the color of axis lines, the x and y grid lines that emanate from the origin; override by the **Grid XYZ Tint** settings, when it is on • **Grid major color** -specifies the color of the primary grid lines • **Grid minor color** -specifies the color of the sub-grid lines
Grid XYZ tint	Specifies whether the XYZ tints are applied over the grid line colors. • **Apply XYZ tint to axis lines** - turns on the color for the axis lines • **Apply XYZ tint to major grid lines** - turns on the color for the primary grid lines • **Apply XYZ tint to minor lines** - turns on the color for the minor grid lines XYZ tints are defined by the COLORX(usually red), COLORY(usually green), and COLORZ(usually blue) variables.
Snap angle	Defines the rotation of the grid (as well as the snap and the cursor crosshairs) for the current viewport, relative to the current UCS. If the snap angle is different from zero, then only dotted grids display (grid lines do not display).

Figure 6.1039



Related Commands

Snap (6.27.38) - sets the snap spacing. Limits (6.20.30) - limits the extent of the grid dots. Settings (6.27.23) - sets grid parameters through a dialog box.

Figure 6.1040



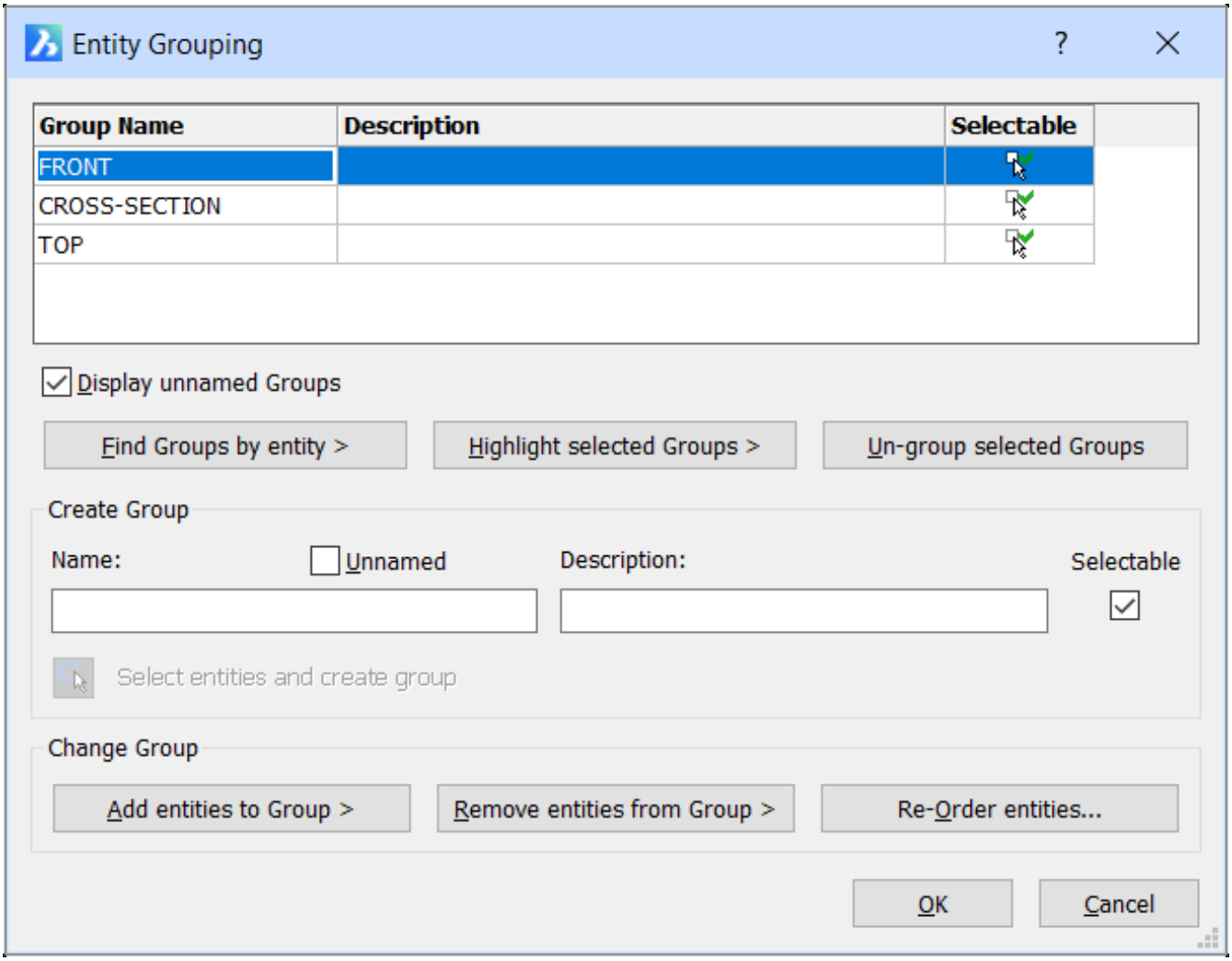
6.15.22 Group

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates, modifies, and deletes named groupings of entities.

Accessing the Command


command bar: **group** menu bar: **Tools | Group** ribbon:**Change | Arrange | Group** toolbar: **Tools |**  : **Group** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/group-dlg.png}) Create, edit, pr erase a named group, and then click **OK**.

Figure 6.1041



Command Options

Option	Description
Display Unnamed Groups	Toggles the display of unnamed groups: • **Off**- displays only named groups created by users • **On**- also displays unnamed groups created by the program
Find Groups by Entity	Reports the name(s) of groups when you select entities in the drawing.Temporarily removes the dialog box, and then prompts: 1. **Select entities** - **(choose one or more entities.)** 2. **Select entities** - (**choose more entities, or press Enter to continue.)** 3. Displays the Group Member List dialog box with one or more group names: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/groupmemberlist-dlg.png}) Click **OK** to close this dialog box and to return to the original dialog box.
Highlight Selected Groups	Highlights entities that belong to the selected group. 1. Choose one or more groups in the Group Name list 2. Click the **Highlight Selected Groups** button 3. Notice that the dialog box disappears and that entities are highlighted: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/objectgrouping-dlg.png}) 4. Click **OK** to return to the Entity Grouping dialog box.
Un-group Selected Groups	Removes group status from selected entities: 1. Select groups in the Group Names list 2. Click the **Un-group Selected Groups** button The names disappear from the list.

Create Group Section	Description
Name	Specifies the names of groups.
Unnamed	Toggles whether the group has a name: • **On** - allows unnamed groups to be created; the Name field is grayed out. • **Off** - requires that groups be named Unnamed groups are assigned the prefix *A following by an incremented number, such as *A2.
Description	Specifies an optional description of the group.

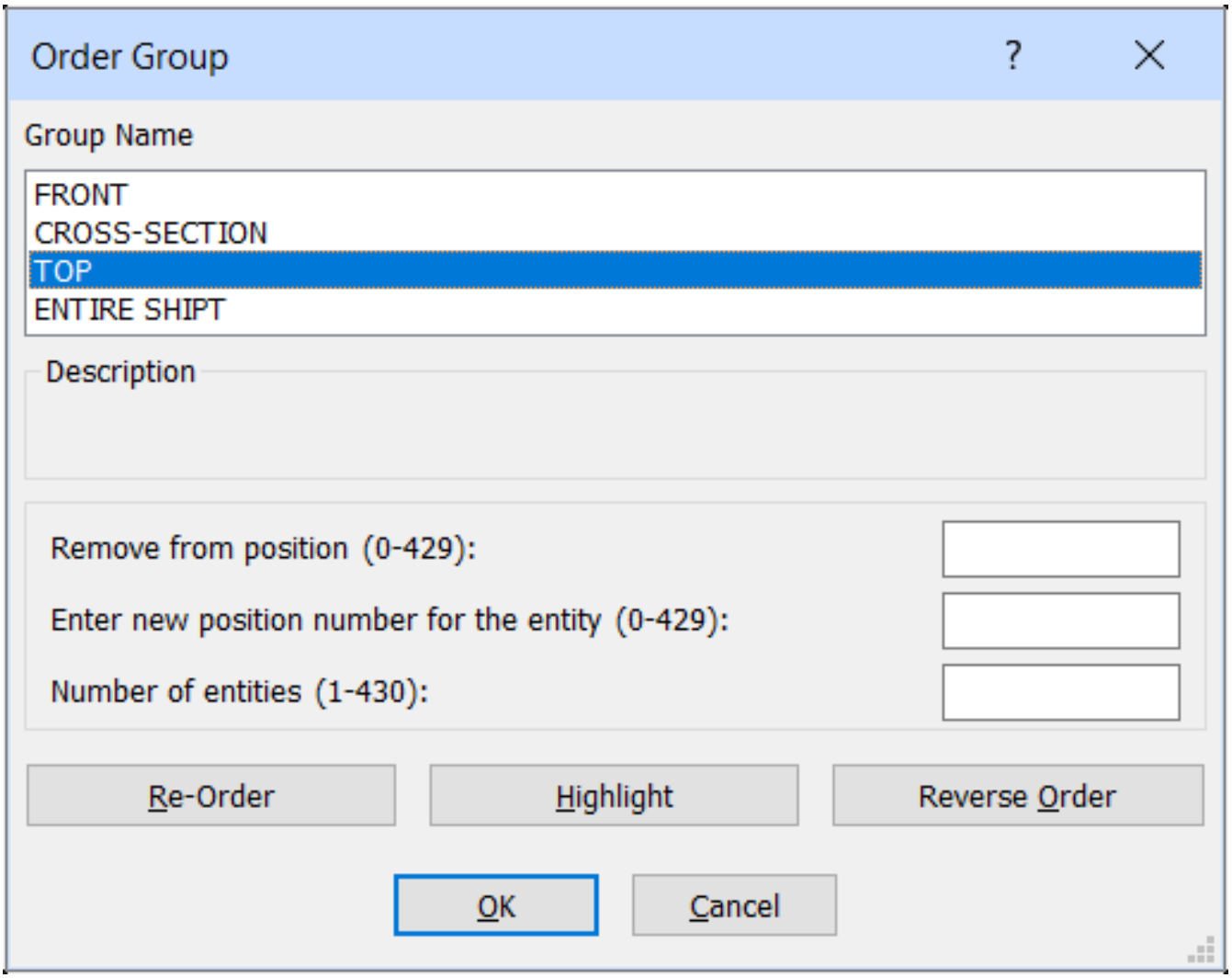
Selectable	Determines how groups are selected when the PICKSTYLE system variable is set to 1 or 3. • **On** - you can select the group but not the entities that make it up • **Off** - you can select individual members of the group
Select Entities	Dismisses the dialog box temporarily, and then prompts you at the command bar: 1. **Select entities** - **(choose one or more entities.)** 2. **Select entities** - **(choose more entities, or press Enter to return to the dialog box.)** Back in the dialog box, the selected entities become part of the group.

Change Group Section	**Description**
Add Entities to Group	Adds entities to the selected group; prompts you at the command bar: 1. **Select entities** - **(choose one or more entities to add to the group.)** 2. **Select entities** - **(choose more entities, or press Enter to return to the dialog box.)**
Remove Entities from Group	Removes entities from the selected group; prompts you at the command bar: 1. **Select entities** - **(choose one or more entities to remove from the group.)** 2. **Select entities** - **(choose more entities, or press Enter to return to the dialog box.)**
Re-Order Entities	Changes the order of entities in groups; displays the Order Group dialog box; see below.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/G/group-reorder-dlg.png})

Order Group Options	**Description**
Group Name	Lists the names of named and unnamed groups.
Remove From Position	Specifies the position number of the entity to be re-ordered.
Enter New Position Number for the Entity	Specifies the new position number for the group.
Number of Entities	Specifies the range of entities to reorder.
Re-Order	Applies the re-ordering dictated by the previous three fields.
Highlight	Highlights entities in the group; dismisses the Order Group dialog box, and then displays the following one: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ • **Next** - highlights the next entity of the group. • **Previous** - highlights the previous entity. • **OK** - returns to the Order Group dialog box.
Reverse Order	Reverses the order of entities in the group.

Figure 6.1042



Grips Editing

Groups can be edited directly through grips in two ways: Selectable - select one entity in the group, and all entities are selected (show highlights and grips). (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/Cm Not Selectable - select one entity and only that entity is selected, such as the ellipse shown below. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image759.png})

Figure 6.1043

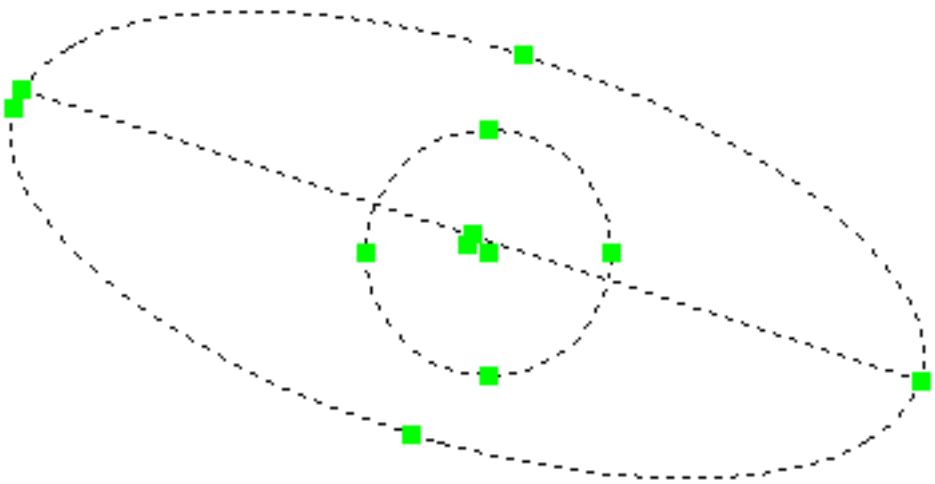
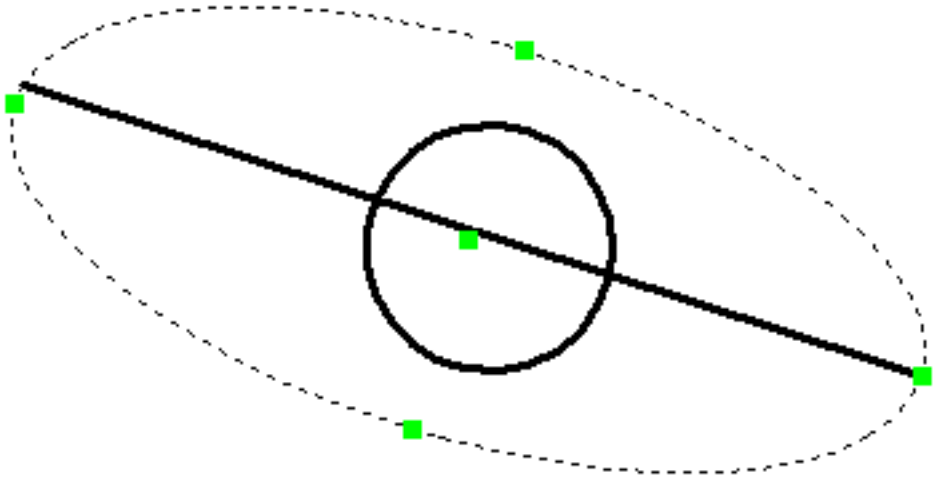


Figure 6.1044



Procedures

- To create a group (1.22.20)
- To modify a group (1.22.20)
- To ungroup entities (1.22.20)
- To change the order of entities (1.22.20)

Related Commands

-Group (6.15.23) - creates and modifies named groups of entities, at the command line. Block (6.6.11) - groups entities into a block (symbol), through a dialog box -Block (6.6.12) - groups entities into a block (symbol), at the command line.

Figure 6.1045

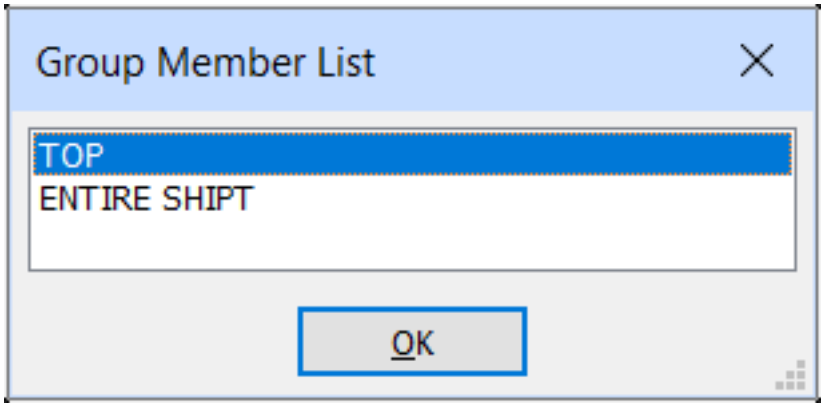


Figure 6.1046

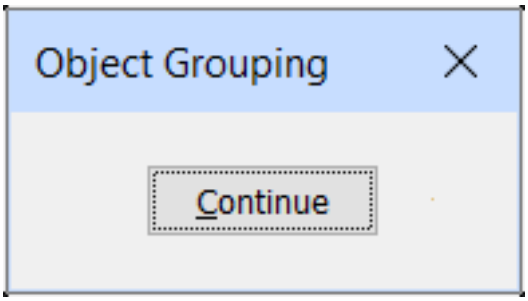


Figure 6.1047

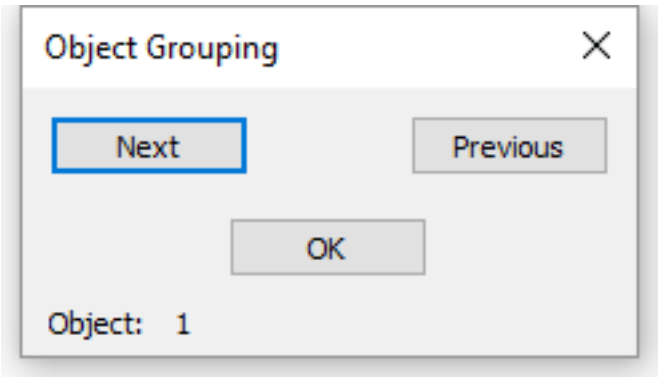


Figure 6.1048



6.15.23 -Group

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and modifies named groups of entities at the command line.

Accessing the Command

command bar: **-group** : **-Group Prompts you in the command bar:** Enter group option [? to list/Order/Add/Remove/Explode/ REName/ Selectable/ Create] <Create>: (**Enter an option)**

Command Options

Option	Description
? to list	Lists named and unnamed groups in the drawing, and then exits the command. Sample listing: Group name: SIDEVIEW Group description: Side View of Boiler Selectable: Yes Unnamed groups are prefixed with *A, followed by an incremented number, such as *A3.
Order	Reverses the order of entities in a group; prompts you: 1. **Enter a group name or ? to list** - **(specify the name of a group.)** 2. **Reverse order** - **(enter Y or N.)**
Add	Adds entities to a group; prompts you: 1. **Enter a group name or ? to list** - **(specify the name of a group.)** 2. **Select entities** - **(choose one or more entities, and then press Enter.)**
Remove	Removes entities from a group; prompts you: 1. **Enter a group name or ? to list** - **(specify the name of a group.)** 2. **Select entities** - **(choose one or more entities, and then press Enter.)**
Explode	Ungroups the selected group; prompts you: • **Enter a group name or ? to list** - **(specify the name of a group.)**
REName	Renames groups; prompts you: 1. **Enter a group name or ? to list** - **(specify the name of a group.)** 2. **Enter new name** - **(specify a different name for the group.)**

Selectable	Toggles the selectability of groups; prompts you: 1. **Enter a group name or ? to list** - **(specify the name of a group.)** 2. **This group is selectable, do you want to change it**- **(enter Y or N.)**
Create	Creates new groups; prompts you: 1. **Enter a group name or ? to list [* to create unnamed]** - **(enter a name, or press * for an unnamed group.)** 2. **Enter description** - **(enter a description, or press Enter for no description.)** 3. **Select entities** - **(choose one or more entities.)** 4. **Select entities** - **(choose more entities, or press Enter to create the group.)**

Related Commands

Group (6.15.22) - creates and modifies named groups of entities, through a dialog box Block (6.6.11) - groups entities into a block (symbol), through a dialog box -Block (6.6.12) - groups entities into a block (symbol), at the command line.

Figure 6.1049



6.15.24 Grading

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Creates a graded surface on existing toposurfaces.

Accessing the Command


Keyboard: **grading** Menu:**BIM | Create Grading** Toolbar: **BIM Site Design** Quad Tab: **Model | Create Grading** Ribbon Tab | Panel:**Data | Site Design|Create Grading : Grading Prompts you:** Select entity: **(Select the entity to define a graded surface.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/bimgradingselecter
Select target surface or [slope/Offset]: **(Select the target surface to create a graded surface or choose an option.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/bimgradingselecttargetsurface.png})
Move mouse to change the slope of the grading: **(Move the mouse to adjust the slope of the grading.)**

Figure 6.1050

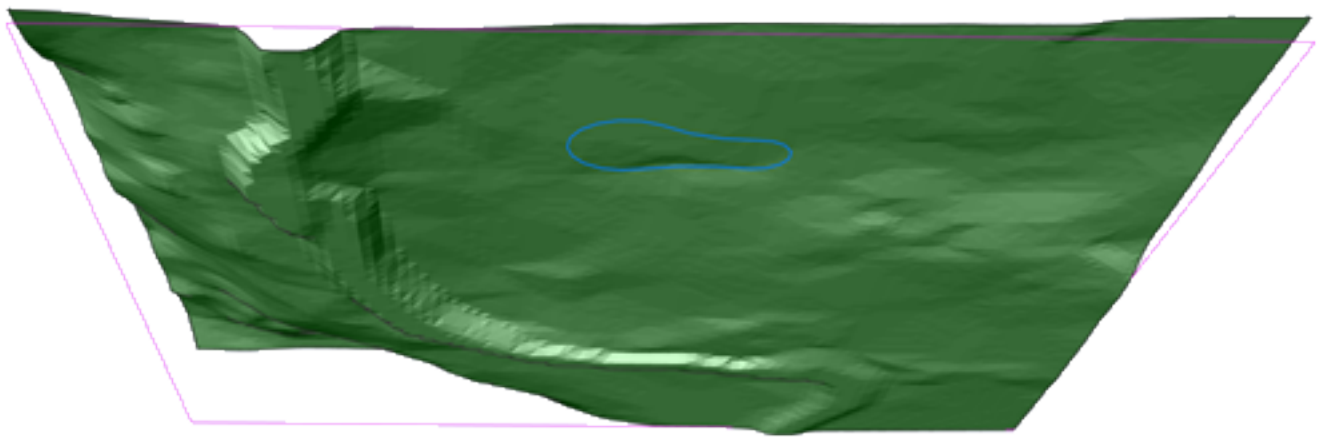
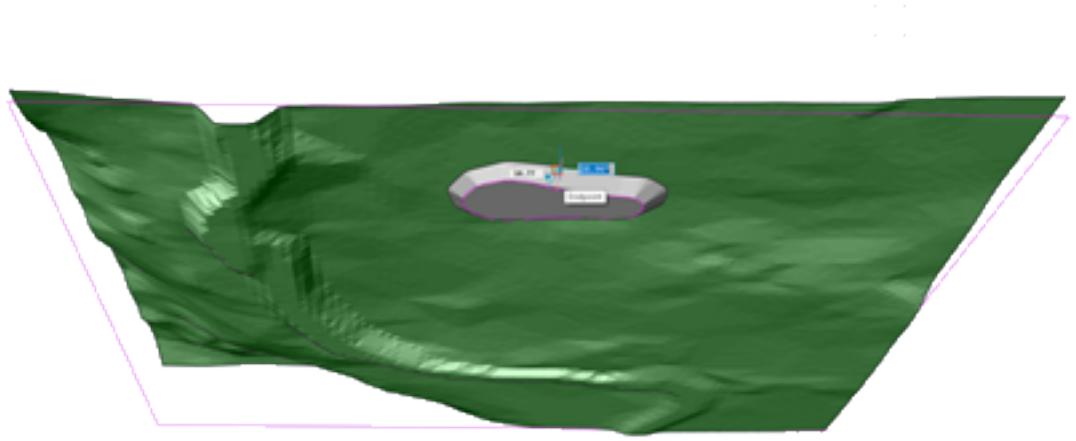


Figure 6.1051



Command Options

Option Description After selecting an entity slope/Offset Sets either the slope or offset option of the grading. Type "O" in the command line Prompts you: **Move mouse to change the offset of the grading or press tab key to change the slope: Note:**that for the offset-slope projection method requires input entity, offset, and slope.

Procedures

Grading a surface to create pond (1.18.9) Displaying the properties of a TIN surface (1.18.9)

Related Commands

Site (6.27.41) -Creates a topographical surface from an imported points file, or placing new points. SiteEdit (1.18.10) - allows to add/remove boundaries, breaklines, and points to a surface.

6.16 H

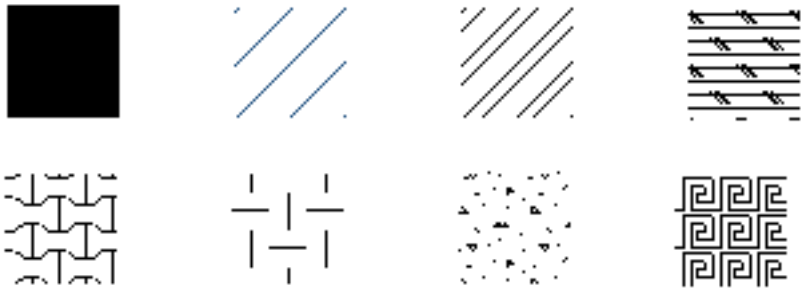
6.16.1 Hatch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Fills closed 2D areas with repeating patterns or solid colors; displays the Hatch and Gradient dialog box. **Note:** 3D entities cannot be hatched. Examples of hatch patterns: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image86.png})

Figure 6.1052



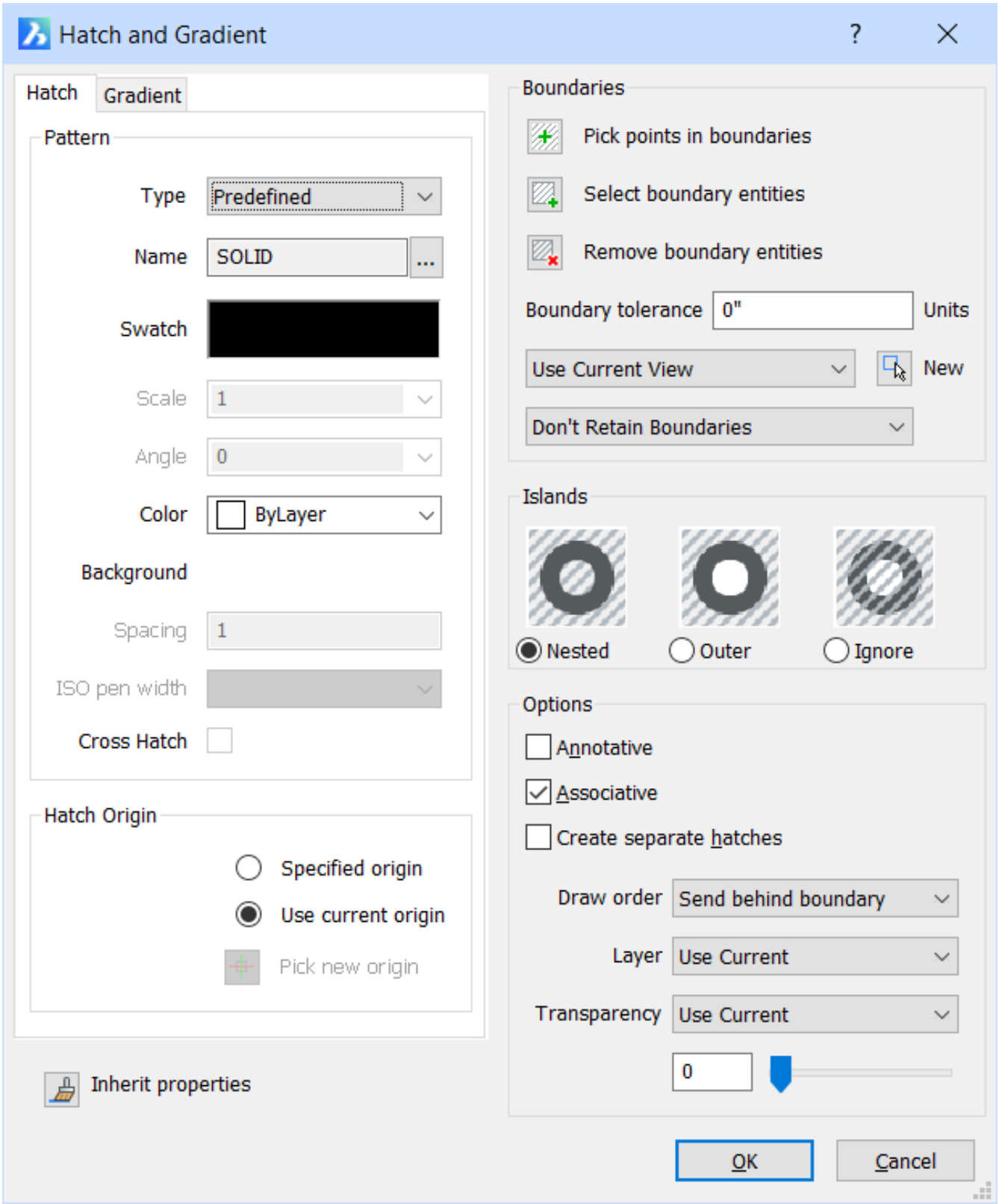
Figure 6.1053



Accessing the Command


command bar: **hatch** aliases: **h, bh (6.6.5)**menu bar: **Draw | Hatch** ribbon:**Draw | Hatch | Hatch** toolbar: **Draw**
|  : **Hatch** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/H/hatdlg.png}) Choose options, and then clickOK.

Figure 6.1054



Command Options

Pattern Options	Description
Type	Select a source of the hatch pattern from the droplist: • **User Defined** - constructs patter from the the user-define parameters of Angle, Spacing, Cross Hatch, Color, and Background Color • **Predefined** - uses the patterns defined in the iso.pat (metric units) or default.pat (imperial units) pattern definition files provided with the program • **Custom** - uses a pattern defined by *.pat files (limited to one pattern definition per file) created by users; the program searches for *.pat files in the folders that are defined in the SRCHPATH (1.13.11) (Support File Search Path) variable
Name	Specify the name of the predefined hatch pattern by one of these methods: • Type a name • Click the **Browse** button to display the Hatch Pattern Palette dialog box, choose a pattern, and then click **OK** This option is unavailable for the User-defined pattern.
Swatch	Click to display the Hatch Pattern Palette dialog box: 1. Choose a pattern 2. Click **OK** to return to the previous dialog box This option is unavailable for the User-defined pattern.
Scale	Specifies the scale factor of the pattern: • Click the droplist and then choose a preset scale factor • Enter a number for the scale factor The scale factor of hatch patterns is generally the same as that for text and linetypes.
Angle	Specifies the angle of the pattern: • Choose a preset angle from the droplist • Else enter a custom angle This angle is measured relative to the positive x-axis of the current UCS.
Color	Specifies the color of the hatch lines: • **Use Current** - uses the current color specified by the CeColor variable. • Choose a color available in the droplist • **Select Color** - **(choose a color from the complete palette; see Color command.)**

Background Color	Specifies the background color under the hatching: • **None** - uses no color. • **Select Color** - **(choose a color from the complete palette; see Color command.)**
Spacing	Specify the distance between hatch lines; this option is available for only user defined patterns
ISO pen width	Specifies the lineweight; this option is available only for ISO hatch patterns
Cross Hatch	Determines whether the pattern is crossed: • **On**- the pattern is repeated at 90 degrees to the original • **Off**- the pattern is not crossed This option is available for only user defined patterns

Hatch Origin Section	**Description**
Specified origin	Uses a point other than the drawings's origin (0,) for the lower-left corner where the pattern begins: 1. Click the **Pick New Origin** button 2. **Select origin point** - **(pick a point in the drawing.)** Use this option to begin the hatch pattern at the corner of a wall or other location where the placement of the pattern is critical.
Use current origin	Uses the drawing's origin (0,0) as the origin of the hatch pattern.
Pick new origin	Dismisses the dialog box temporarily, and then prompts you at the command bar: 1. Click the**Pick New Origin**button 2. **Select origin point**- **(pick a point in the drawing)** This option is available for only the specified (user defined) origin.

Boundaries Section	**Description**

Pick points in boundaries	Specifies the closed areas (boundaries) in which to place the pattern: 1. Click the **Pick Points in Boundaries** button; the program temporarily dismisses the dialog box 2. **Select a point to define a boundary or hatch area [or Select entities/Undo]** - **(pick a point inside a closed area, or enter an option): Select Entities** - prompts you to select entities that form the boundary **Undo** - removes the previously selected area from the selection set Notice that the program draws the hatch immediately, allowing you to preview its scale factor, angle, origin, and other parameters 3. **Select a point to define a boundary or hatch area [or Select entities/Undo]** - **(pick another point, or else press Enter to return to the dialog box.)** You cannot pick an area that already contains a hatch pattern, or areas that are not closed (whose gap is larger than the value specified by Boundary Tolerance).
Select boundary entities	Selects the entities that make up the hatching boundary to constrain the extent of the pattern: 1. Click **Select Boundary Entities** button; the program temporarily dismisses the dialog box 2. **Select entities** - **(use any selection method to choose one or more entities.)** 3. **Select entities** - **(select more entities, or else press Enter to return to the dialog box.)** This option allows you to hatch closed areas, including ones that already contain a hatch. It hatches open areas whose gap is less than the value specified by Boundary Tolerance.
Remove boundary entities	Remove entities from the detected boundary set: 1. Click **Remove Boundary Entities** button; the program temporarily dismisses the dialog box 2. **Select entities** - **(use any selection method to choose one or more entities.)** 3. **Select entities** - **(select more entities to remove, or else press Enter to return to the dialog box.)**

Boundary Tolerance	Specifies the largest gap the program ignores when hatching a boundary that is not fully closed: • **0** - (Default) The tolerance is set by the application, based on the current view size.When zoomed in closely, boundary detection will fail; when zooming out further so the contour 'looks' closed, the boundary will become detected • Any Value - Defines the maximum gap in drawing units. The value is saved in the HPGAPTOL (1.13.11) system variable.
Current View/ Boundary Set	Specifies where the program should search for entities that make up the boundary: • **Use Current View** - searches all entities in the current viewport • **Use Boundary Set** - searches only the current selection set for possible boundaries
New	Creates a new selection set of entities that make up the pattern boundary: 1. Click the **New** button; the program temporarily dismisses the dialog box 2. **Select entities** - (**use any selection method to choose one or more entities.)** 3. **Select entities** - **(select more entities, or else press Enter to return to the dialog box.)** The next step is to click the **Pick Points in Boundaries** button to choose the hatch area.
Retain boundaries	Determines what happens to boundaries: • **Don't Retain Boundaries** -removes the boundary after the hatch is created • **Retain Boundaries as Polylines** - retains boundaries and turns them into polylines • **Retain Boundaries as Regions** -retains boundaries and turns them into regions (Pro or Platinum versions only)

Islands Section	**Description**
Nested	When a closed hatch boundary contains other boundaries, BricsCAD hatches alternating areas.
Outer	When a closed hatch boundary contains other boundaries, BricsCAD hatches only the outermost area.
Ignore	When a closed hatch boundary contains other boundaries, BricsCAD hatches all interior areas, as if they were not present.

Options Section	**Description**

Annotative	Toggles the the Annotative property of the hatch: • **On** - applies the current annotative scale; see the CANNOSCALE (1.13.11)variable • **Off** - hatches use scale factor specified above When this option is turned on, the Associative option is unavailable
Associative	Toggles the associativity of hatch patterns:when the boundary is changed, the pattern automatically updates itself. • **On**- patterns is associative • **Off**- patterns i not associative: when the boundary is changed, the pattern remains as originally placed This option is not available for annotative hatches.
Create separate hatches	Toggles how multiple hatches are treated: • **On**- creates a separate hatch entity for each boundary in the selection set • **Off**-creates a single hatch entity of all areas
Draw order	Specifies where the pattern should be placed relative to overlapping entities: • **None**- draw order is ignored • **Send to Back**- pattern is behind (underneath) all other entities • **Send to Front**- pattern is in front (on top) of all other entities • **Send Behind Boundary**- pattern is placed behind the boundary • **Bring in Front of Boundary**- pattern is placed in front of the boundary
Layer	Specifies the layer on which the hatch should be placed: • **Use Current**- places the pattern on the current layer • **Other layer name** - **(choose a layer from the droplist onto which to place the pattern)**
Transparency	Set the transparency property of the hatch: • **Use current**- apply the current transparency value as defined by theCETRANSPARENCY (1.13.11)system variable • **By Layer**- apply the value of the transparency property of the layer on which the hatch is placed • **By Block**- the transparency value is controlled by block • **Specify value**- type a value or use the slider to set a value between 0 (fully opaque) and 90. The transparency level is limited to 90% to avoid confusion with layers that are frozen or turned off.The transparency value for new hatches is saved by theHPTRANSPARENCY (1.13.11)system variable.

Inherit properties	Copies the properties of another existing hatch pattern to use with this pattern: 1. Click the**Inherit Properties**button;the program temporarily dismisses the dialog box 2. **Select hatch entity to copy properties from-(use any selection method to choose a single hatch pattern already in the drawing.)** 3. **Select hatch entity to copy properties from-(select more hatches, or else pressEnterto return tot he dialog box.)** This button is useful for quickly setting options of a new hatch based on existing hatches.

Tutorial: Use the Quad to simplify working with Hatch

Click here (1.5.1) to watch

Grips Editing

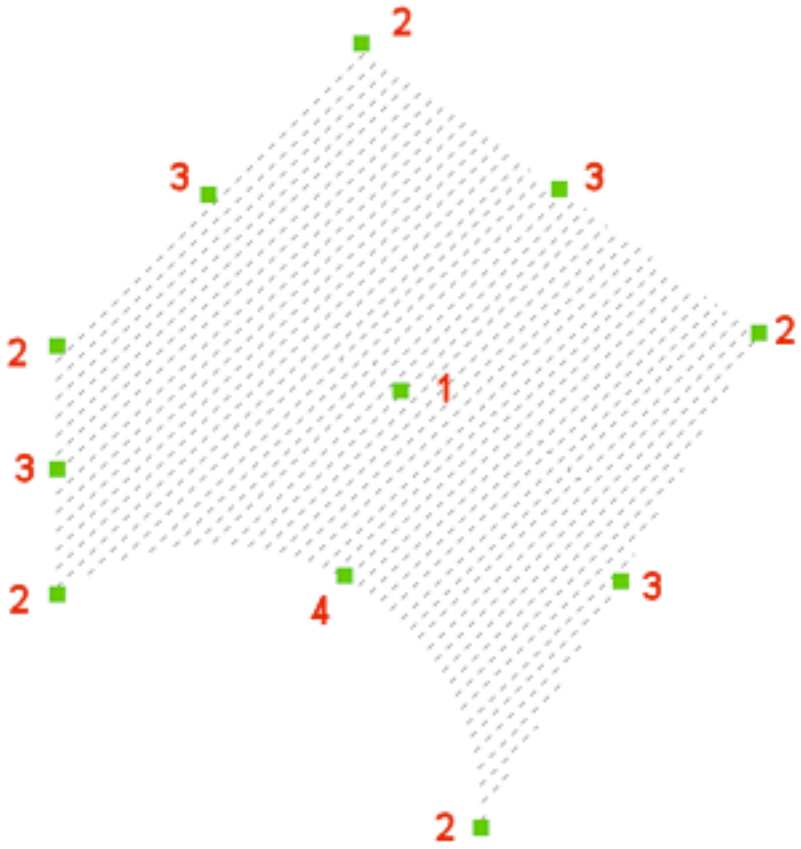
Non-associative hatches can be edited directly through grips:

1. Select the hatch. Notice that it has one central grip (1), grips on each boundary vertex (2) and grips in the midpoint of each boundary segment (3 and 4).
2. Drag the central grip (1) to move the hatch.
3. Drag a vertex grip (2) or a midpoint grip (3 and 4) to change the hatch boundary.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/HatchGripEditing.png})

1. Center grip: moves the hatch
2. Vertex grip: moves the vertex
3. Midpoint grip: moves the boundary segment
4. Arc midpoint grip: modifies the arc radius

Figure 6.1055



Procedures

- General procedure to create hatches and gradient fills (1.17.28)
- To define hatch properties (1.17.28)
- To define the boundary for a hatch or gradient fill (1.17.28)
- To edit a hatch or gradient fill (1.17.29)
- Defining the MAXHATCH setting (1.17.26)

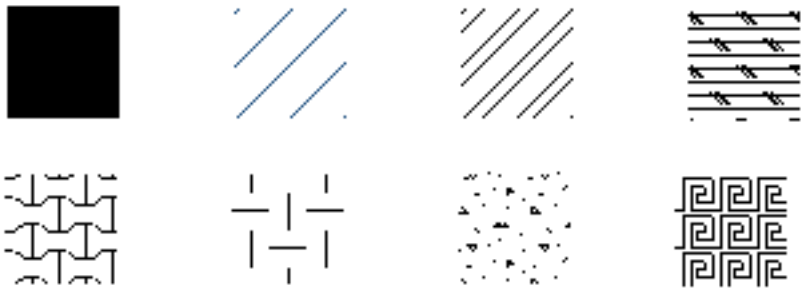
Related Commands

Gradient (6.15.17) - places a gradient fill through a dialog box. -Hatch (6.16.2) - places hatch patterns through the command bar. HatchEdit (6.16.3) - edits hatches and gradient fills in a dialog box. -HatchEdit (6.16.4) - edits the boundaries of hatch and gradient entities; displays prompts in the command bar. HatchGenerateBoundary (6.16.5) - generates a boundary around a hatch or gradient fill. Properties (6.24.49) - changes the properties of hatch patterns. Settings (6.27.23) - specifies the default parameters for hatch patterns. Solid (6.27.37) - draws 3- and 4-side solid-filled areas.

Figure 6.1056



Figure 6.1057



6.16.2 -Hatch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Fills closed 2D areas with repeating patterns; displays prompts at the command bar.Examples of hatch patterns: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/so

Note: 3D entities cannot be hatched.

Accessing the Command

command bar: **-hatch** aliases: **-h, -bh** Quad: **2D Editing** | ribbon:**Draw | Hatch | Hatch | Quick Hatch** : **-Hatch Prompts you in the command bar:** Current hatch pattern: ANSI31 Specify internal point or [hatch Properties/Select entities/Remove boundaries/ Advanced options/Draw order/Origin/ANnotative/Layer/Transparency/Undo]: **(Pick a point, or enter an option)** Selecting everything visible... (*) Specify internal point or [hatch Properties/Select entities/Remove boundaries/ Advanced options/Draw order/Origin/ANnotative/Layer/Transparency/Undo]: **(Pick another point, or enter an option, or press Enter to create the hatch)** * Entities which are not visible in the current viewport when the command is launched cannot be selected. Partly visible entities can, however, be selected.

Command Options

Option	**Description**
Current hatch pattern: ANSI31	Reports the name of the current hatch pattern; you change the pattern with the Properties option.
Specify internal point	Pick a point inside a closed area; BricsCAD creates an invisible boundary, and then hatches the area. You can pick more than one area to hatch at a time.

Properties Options	**Description**
Enter a pattern name	Specifies the name of the hatch pattern.
? to list patterns	Lists the names of all patterns; reminds you of their names.Press **F2** to see the list in the prompt history window.
Solid	Specifies a solid-color filled hatch.
User defined	Allows you to define simple hatch patterns based on angle, scale, and crosshatching.
Proceed	Skips the next two prompts, and returns to the original prompt.
Scale for pattern	Specifies the scale factor of the hatch pattern.The scale factor is usually the same as that of text and linetypes.
Angle for pattern	Specifies the angle at which the pattern is drawn.

User Defined Options	**Description**

Proceed	Skips the following prompts, and returns to the original prompt.
Space between standard pattern lines	Specifies the distance between hatch lines.
Angle for lines	Specifies the angle at which the pattern is drawn.
Cross-hatch area?	Applies the pattern a second time at 90 degrees to the first.
Select	Prompts you to select the entities making up the boundary: **Select entities** - **(select one or more entities;press Enter to return to original prompt.)**
Remove	Removes boundary entities; prompts you to select island (interior) entities, which will be hatched through.
Advanced	Accesses advanced hatching properties; prompts you: **Enter an option [Boundary set/Retain boundary/Island detection/Specify style/Set associativity(A)/Boundary tolerance(T)/Separatehatches(H)]** - **(enter an option; see below.)**
Boundary set	Specifies the entities to consider for creating boundaries; prompts you: **Specify candidate set for boundary [New selection/All objects(E)]<Everything>**- **(enter an option)**: • **Everything** - selects all entities in the current viewport • **New** - prompts you to select entities
Retain boundary	Determines whether the temporary boundary is retained after the command ends; prompts you: **Retain derived boundaries? [Yes-Retain boundariess(Y)/No-Discard boundary(N)]<No>- (enter an option)**: **Yes** - boundary is kept in the drawing. **No** - boundary is erased.
Island detection	Toggles whether islands are hatched over or not; prompts you: **Do you want island detection? [Yes-Detect islands(Y)/No-Set ray casting(N)]<Yes>**- **(enter an option)**: • **Yes** - islands (interior boundaries) are not hatched. • **No** - islands are hatched over.
Specify style	Specifies how islands are treated; prompts you: **Enter hatching style: [Nested/Outer/Ignore]<Nested>**- **(enter an option)**: • **Nested** - alternating islands are hatched, beginning with the outermost one. • **Outer** - only the outermost region is hatched; interior islands are not hatched. • **Ignore** - islands are ignored and hatched over.

set Associativity	Toggles whether hatches are associative; prompts you: **Associativity hatch? [Yes-Associative hatch(Y)/No-Nonassociative hatch(N)]<Yes>- (enter an option)**: • **Yes** - hatches are associative; when their boundary changes, they automatically update. • **No** - hatches are not associative; when the boundary changes, the hatch keeps its original shape.
boundary Tolerance	Specifies the largest gap that BricsCAD will close when hatching open areas: **Specify a boundary gap tolerance value <0>- (enter a value)**. A value of 0 means that no gaps in the boundary are tolerated by the program.
separate Hatches	**Create separate hatches? [Yes/No-Do not separate(No)]-(enter an option)**: • **Yes** - creates separate hatches if the hatch is composed of multiple closed boundaries • **No** - creates a single hatch entity.
Draw order	Specifies whether the hatch pattern appears visually above or below overlapping entities; prompts you: **Enter an option [None/Send to back(B)/Send to front(F)/Send behind boundary(H)/Bring in front of boundary(D)]- (enter an option)**: • **None** - ignores draw order • **send to Back** - places the hatch pattern behind (below) other entities; this prevents the hatch from obscuring other entities • **bring to Front** - places the pattern in front (on top) of other entities • **send beHind boundary** - places the pattern behind its boundary; this makes it easier to select the boundary • **bring in front of bounDary** - places the pattern in front of its boundary
Origin	Prompts for a new hatch origin point: **New value for the hatch origin <0.0000,0.0000>- (enter an option)**: • **x,y** - enter x,y coordinates • Pick a point in the drawing • **Enter** - **use the default value** Use entity snaps to pick the origin accurately.
Annotative	Sets the annotative property for the hatch and applies the current annotative scale (see the CANNOSCALEvariable); prompts you: **Create annotative hatch [Yes/No] <No>** - **(enter an option)**: • **Yes** - add annotative scale factor to hatch • **No** - uses the value specified by **Scale for Pattern** to create a non-annotative hatch

Layer	Creates the hatch on a specified layer, rather than on the current layer.Prompts you: **Specify layer or . for use current <.>- (enter an option)**: • Enter the name of the layer on which you want to create the hatch • Enter . or press **Enter** to create the hatch on the current layer.
Transparency	Sets the transparency property of the hatch.Prompts you: **Specify transparency [0/90/ByLayer(L)/ByBlock(B)/. for use current(.)] <Use current>- (enter an option)**: • **Use current** - applies the current transparency value as defined by the CETRANSPARENCY system variable. • **Specify transparency [0/90]** - specifies a value between 0 (fully opaque) and 90.The transparency level is limited to 90% to avoid confusion with layers that are frozen or turned off. • **By Layer** - applies the value of the transparency property of the layer on which the hatch resides • **By Block** - transparency value is controlled by block The transparency value for new hatches is saved by the HPTRANSPARENCYvariable.
Undo	Remove selected boundaries from the selection

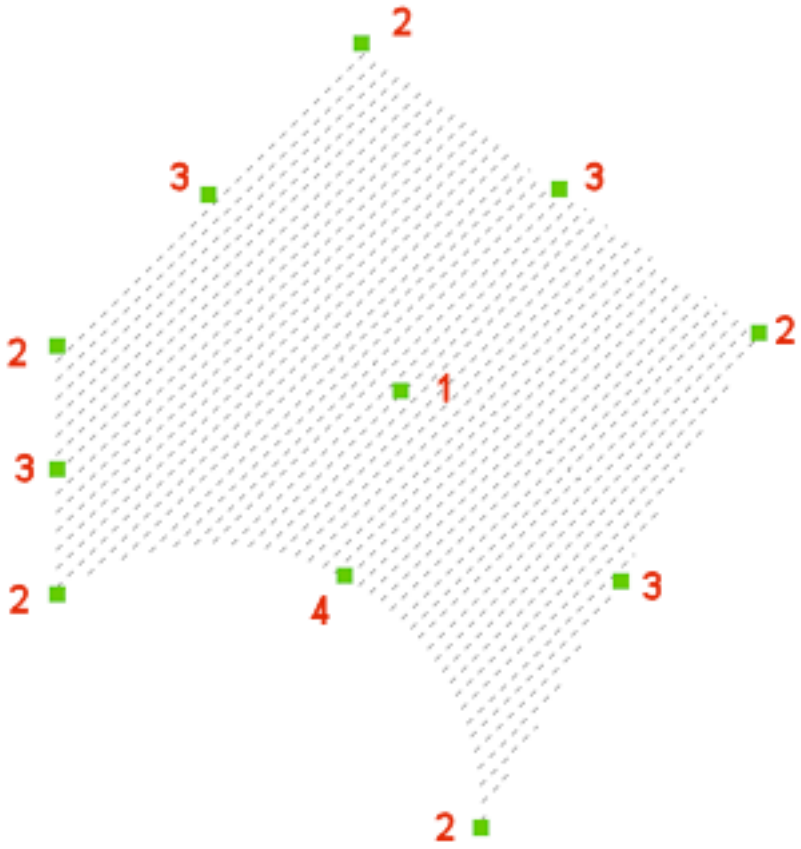
Grips Editing

Hatches can be edited directly through grips:

1. Select the hatch. Notice that it has one central grip (1), grips on each boundary vertex (2) and grips in the midpoint of each boundary segment (3 and 4).
2. Drag the central grip (1) to move the hatch.
3. Drag a vertex grip (2) or a midpoint grip (3 and 4) to change the hatch boundary.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/HatchGripEditing.png})
1. Center grip: moves the hatch 2. Vertex grip: moves the vertex 3. Midpoint grip: moves the boundary segment 4. Arc midpoint grip: modifies the arc radius

Figure 6.1058



Procedures

- General procedure to create hatches and gradient fills (1.17.28)
- To define hatch properties (1.17.28)
- To define the boundary for a hatch or gradient fill (1.17.28)
- To edit a hatch or gradient fill (1.17.29)
- Defining the MAXHATCH setting (1.17.26)

Related Commands

Hatch (6.16.1) - place hatch patterns through a dialog box. Boundary (6.6.15) and -Boundary (6.6.16) place a polyline boundary inside closed areas. HatchEdit (6.16.3) - edits hatches and gradient fills in a dialog box. -HatchEdit (6.16.4) - edits the boundaries of hatch and gradient entities; displays prompts in the command bar. HatchGenerateBoundary (6.16.5) - generates a boundary around a hatch or gradient fill. Properties (6.24.49) - changes the properties of hatch patterns. Settings (6.27.23) - specifies the default parameters for hatch patterns. Solid (6.27.37) - draws 3- and 4-side solid-filled areas.

Figure 6.1059



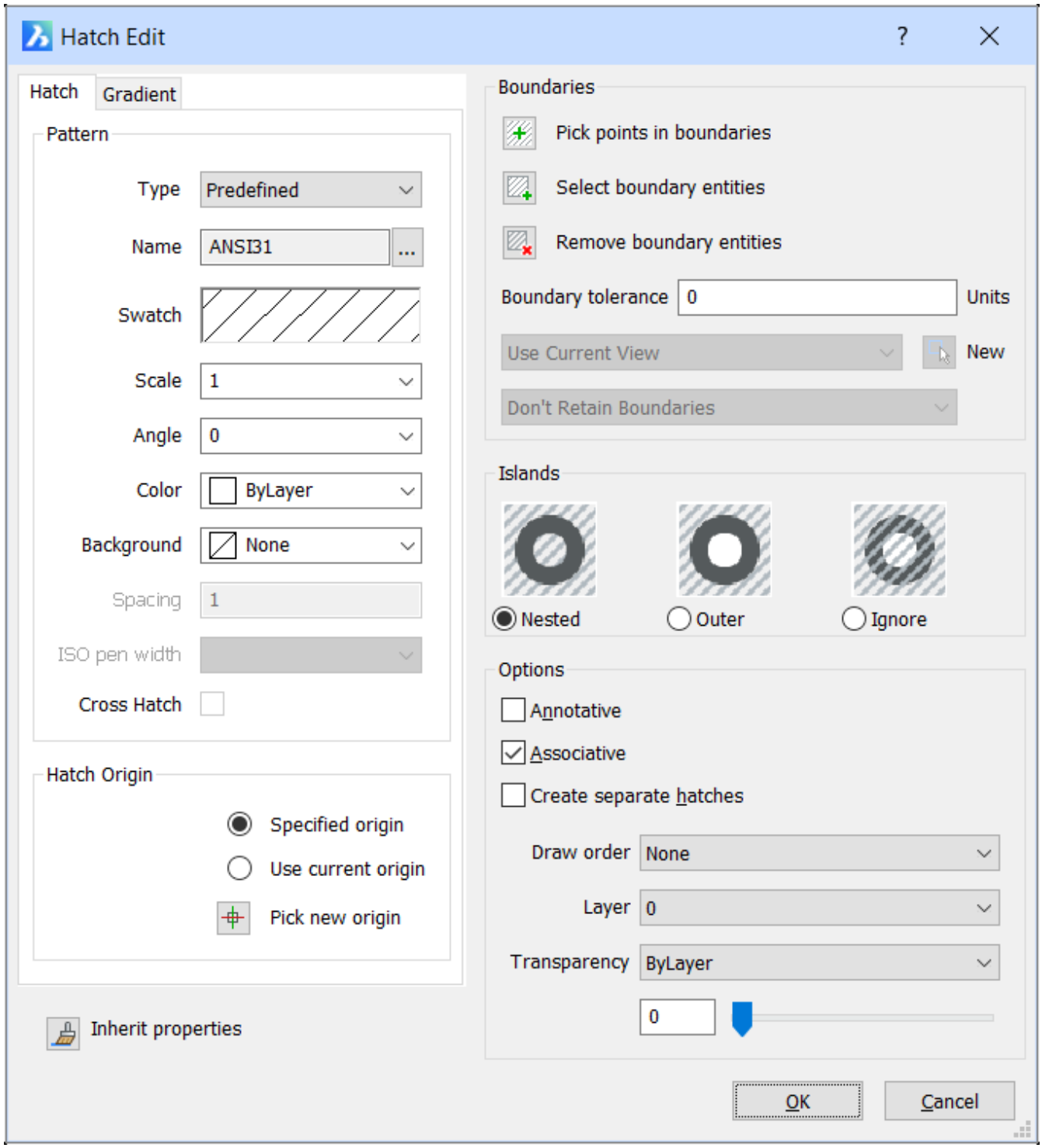
6.16.3 HatchEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits hatches and gradient fills through a dialog box.

Accessing the Command

 command bar: **hatchedit** transparent: **'hatchedit** alias: **he** mouse shortcut: **Double-click a pattern** menu bar: **Modify | Hatch edit** toolbar: **Modify |**  : **Hatchedit Prompts you in the command bar:** Select hatch entity: (**Select a hatch or gradient fill in the drawing)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he dlg.png}) Make changes, and then click the **OK** button to apply the changes. As an alternative, you can use the Properties (6.24.49) panel to edit hatch and gradient parameters.

Figure 6.1060



Command Options

Option	Description
Select hatch entity	Select an entity for editing: • Hatch pattern - see the Hatch (6.16.1) command for the meaning of the options in the dialog box • Gradient fill - see the Gradient (6.15.17) command • Any other entity - the program repeats the Select hatch entity prompt

Procedures

- To edit a hatch or gradient fill in the properties bar (1.17.28)
- To edit a hatch in a dialog box (1.17.29)
- To edit a gradient fill in a dialog box (1.17.29)

Related Commands

Gradient (6.15.17) - Fills closed areas with solid fills in nine different patterns and one or two colors; displays the Hatch and Gradient dialog box. Hatch (6.16.1) - fills closed areas with repeating patterns; displays the Hatch and Gradient dialog box. -Hatch (6.16.2) - fills closed areas with repeating patterns; displays prompts at the command bar. -HatchEdit (6.16.4) - Edits the boundaries of hatch and gradient entities; displays prompts in the command bar. HatchGenerateBoundary (6.16.5) - generates a boundary around a hatch or gradient fill.

Figure 6.1061



6.16.4 -HatchEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits the boundaries of hatch and gradient entities; displays prompts in the command bar.

Accessing the Command

command bar: **-hatchedit** transparent: **'-hatchedit** Quad menu: **2D Editing** | | | | ribbon:**Draw** | **Hatch | Hatch Add Loop** : **-HatchEdit Prompts you in the command bar:** Select hatch entity: **(Select a hatch or a gradient entity in the drawing)** Enter hatch option [Disassociate/ADd/Remove/ASsociate]: **(Enter an option, and then press Enter)**

Command Options

Option	Quad Icon	Description
Disassociate		Removes the associative property of the selected hatch or gradient entity; the programreports in the command bar: **Hatch boundary associativity removed**
Add		Adds boundary entities to the selected hatch or gradient entity; prompts you: **Specify internal point or [Select entities]** - **(do one of the following):** • Click inside a closed area; prompts you: **Selecting everything visible... Specify internal point or [Select entities]** - **(click inside another closed area, or press Enter to end.)** • Choose **Select entities**; prompts you: **Select entities** - **(select a closed entity.) Entities in set: 1 Select entities** - **(select another closed entity or else press Enter to end.)**

Remove		Removes boundary entities from the selected hatch or gradient entity. Boundaries of the selected hatch or gradient entity highlight; prompts you: 1. **Select entities** - **(select a boundary.)** 2. **Select entities** - **(select another entity, or press Enter to end.)**
Associate		Associates the selected hatch or gradient entity to a different boundary set; prompts you: **Specify internal point or [Select entities]** - **(do one of the following)**: • Click inside a closed area; prompts you: **Selecting everything visible... Specify internal point or [Select entities]** - **(click inside another closed area or press Enter to stop.)** • Choose **Select entities**; prompts you: **Select entities** - **(select a closed entity.) Entities in set: 1 Select entities** - **(select another closed entity or press Enter to stop.)**

Related Commands

HatchEdit (6.16.3) - edits hatches and gradient fills in a dialog box. -Hatch (6.16.2) - fills closed areas with repeating patterns; displays prompts at the command bar. Hatch (6.16.1) - fills closed areas with repeating patterns; displays the Hatch and Gradient dialog box. HatchGenerateBoundary (6.16.5) - generates a boundary around a hatch or gradient fill. Gradient (6.15.17) - fills closed areas with solid fills in nine different patterns and one or two colors; displays the Hatch and Gradient dialog box.

Figure 6.1062



6.16.5 HatchGenerateBoundary

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Generates a boundary around a hatch or gradient fill at the command line. (Use the Boundary command for the dialog box version of this command.)

Accessing the Command

 command bar: **hatchgenerateboundary** menu: **Draw | Boundaries | Hatch Generate Boundary** Quad: **2D Editing |**  ribbon:**Draw | Hatch | Hatch Add Loop** : **HatchGenerateBoundary Prompts you in the command bar:** Select hatches to generate boundary: **(Select a hatch or gradient fill)** Entities in set: 1 Select hatches to generate boundary: **(Select another hatch or gradient fill, or else press Enter to stop adding entities)** Generated boundaries for # hatch(es) The program creates a boundary made of a polyline around the selected hatches or gradient fill.

Command Options

Option	Description
Select hatches to generate boundary	Choose one or more patterns or filled areas, or press **Enter** to end the command

Related Commands

BHatch (6.6.5) - places hatch patterns through a dialog box. Boundary (6.6.15) and -Boundary (6.6.16) place a polyline boundary inside closed areas. GenerateBoundary (6.15.14) - Creates closed polylines from faces of 3D solids as well as the boundaries detected when the Enable Boundary Detection open of SELECTIONMODES is activated. Gradient (6.15.17) - Fills closed areas with solid fills in nine different patterns and one or two colors; displays the Hatch and Gradient dialog box. Hatch (6.16.1) - fills closed areas with repeating patterns; displays the Hatch and Gradient dialog box. -Hatch (6.16.2) - fills closed areas with repeating patterns; displays prompts at the command bar. HatchEdit (6.16.3) - edits hatches and gradient fills in a dialog box. -HatchEdit (6.16.4) - Edits the boundaries of hatch and gradient entities; displays prompts in the command bar.

Figure 6.1063



6.16.6 HatchToBack

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the draw order of all hatch entities in the drawing to display behind all other entities.

Accessing the Command

 command bar: **hatchtoback** menu bar: **Tools | Draw Order | Send Hatches to Back** ribbon:**Draw | Hatch | Send Hatches to Back** toolbar: **Draw Order |**  : **HatchToBack** 3 hatch entity(ies) sent to back

Command Options

No options. This command selects all hatch patterns in the current drawing automatically.

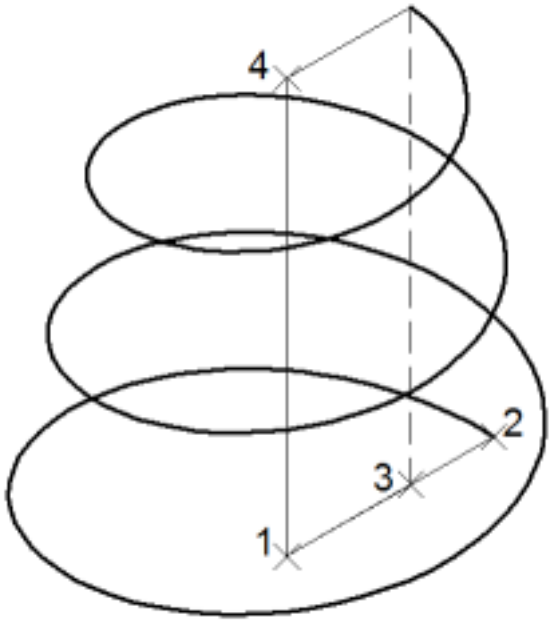
Related Commands

DrawOrder (6.10.38) - changes the display order of overlapping entities. DrawOrderByLayer (6.10.37) - sets draw order of overlapping objects through layer names. TextToFront (6.29.14) - sets the draw order of all texts and dimensions in the drawing to display in front of all other entities.

Figure 6.1064

Shape Classic Pro Platinum Mechanical BIM

Figure 6.1065



6.16.7 Helix

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates a 2D spiral or a 3D helix. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/H/HelixStep
The initial values for drawing a helix are

- Base Radius - 1
- Top Radius - equal to the base radius
- Base Diameter - 2
- Top Diameter - equal to the base diameter
- Height - 1
- Turns - 3
- Twist - CCW (counter clockwise)

In a drawing session, the previously used value becomes the new default value.

Accessing the Command

 command bar: **helix** transparent: **'helix** menu bar: **Draw| Helix** ribbon:**Draw | Draw | Polylines | Helix** toolbar: **Polylines |**  : **Helix Prompts you in the command bar:** Number of turns = 3.0 Twist = CCW Specify center point of base: **(Specify a point in the drawing; see #1 above)** Specify base radius or [Diameter] <default>: **(Specify the radius of the base, or else press Enter to accept the default; see #2)** Specify top radius or [Diameter] <default>: **(Specify the radius of the top, or else press Enter to the top radius equal to the base radius; see #3)** Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <default>: **(Specify the height of the helix, enter an option, or else enter 0 to create a 2D spiral; see #4)**

Command Options

Option	Description
Specify center point of base	Pick a point in the drawing for the center of the helix's base; see point #1 in the figure above

Specify base radius	Specifies theradius of the helix's base; see point #2: • Enter a value • Press **Enter** to accept the default value • Enter **D** to specify a diameter instead
Diameter	Specifies the diameter of the base of the helix; prompts you: **Specify diameter <default>** - **(enter a value or else press Enter to accept the default value.)**
Specify top radius	Specifies theradius of the helix's top; see point #3: • Enter a value • Press **Enter** to accept the default value • Enter **D** to specify a diameter instead
Specify helix height	Specifies the height of the helix: • Enter a value; enter **0** for a 2D spiral (a helix of no height) • Press **Enter** to accept the default value • Enter an option, as described below
Axis endpoint	Defines the height and orientation (angle) of the helix; prompts you: **Specify axis end point**- **(specify a point in the drawing.)** The first point of the axis was specified earlier with the**Specify center point of base** option.
Turns	Specifies the number of revolutions for the helix, between 1 and 500; prompts you: **Enter number of turns <default>**- **(enter a number, or else press Enter to accept the default.)**
turn Height	Specifies the height of one revolution of the spiral; prompts you: **Specify distance between turns <default>**- **(enter a value, pick two points, or else press Enter to accept the default.)** The Turn Height and the number of Turns define the height of the helix.
tWist	Determine which way the spiral/helix rotates; prompts you: **Enter twist direction of helix [CW/CCW] <default>**- **(enter an option)**: • **CW** - clockwise • **CCW** - counterclockwise

Grips Editing

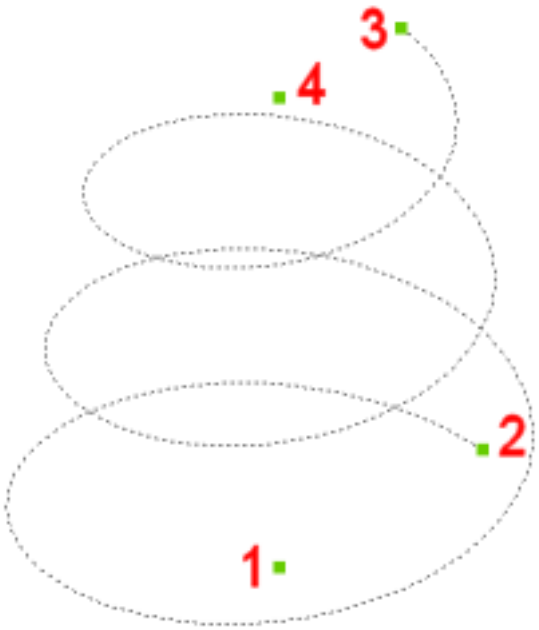
Helixes can be edited through grips:

1. Select the helix. Notice that it has four grips.
2. Drag the grips to perform the following actions:

- 1 - Moves the helix
- 2 - Modifies the base radius
- 3 - Modifies the top radius
- 4 - Modifies the height

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/H/HelixGripEditing.png})

Figure 6.1066



Related Commands

3dPoly (6.3.10) - draws 3D polylines. Pline (6.24.27) - draws polyline lines, arcs, and splines. Spline (6.27.65) - draws open and closed splines.

Figure 6.1067



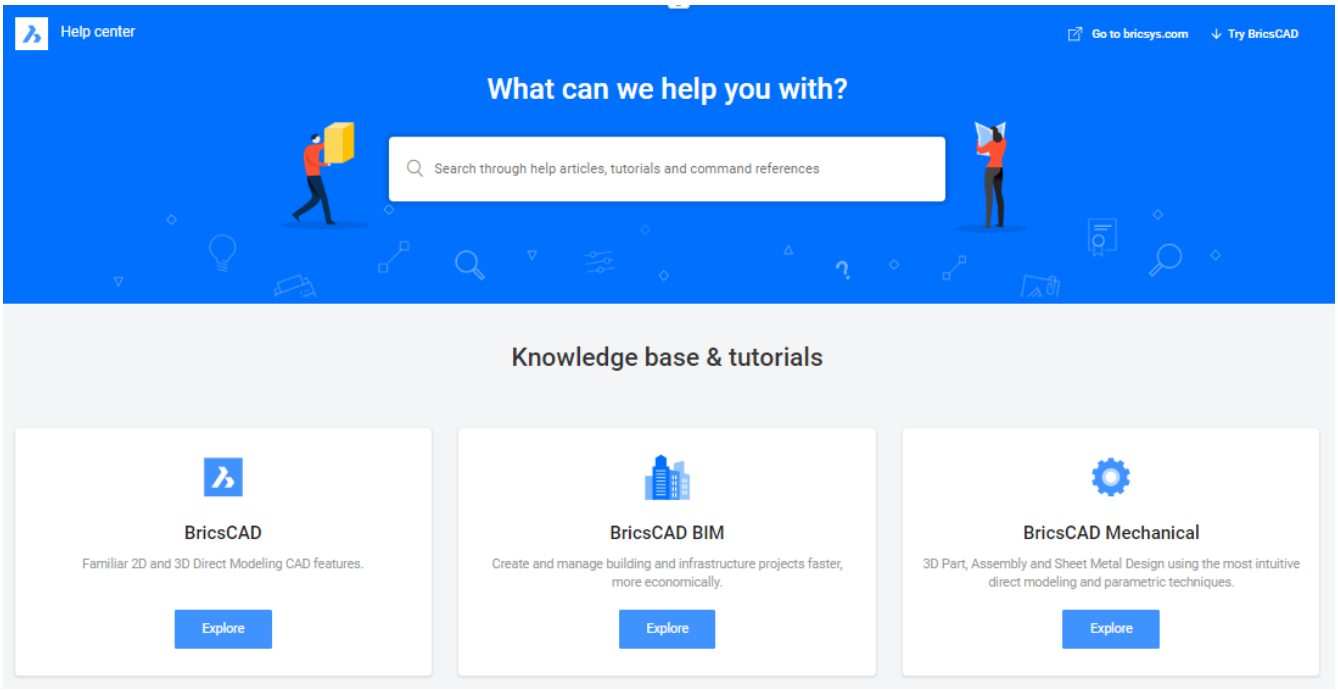
6.16.8 Help

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the BricsCAD online Help Center.

Accessing the Command


command bar: **help** transparent: **'help** alias: **?** **(6.36.1)**shortcut: **F1** menu bar: **Help | Help** toolbar: **Standard |**  ribbon: **Home | Help | Help** : **Help** Open a new tab in the computer's default Web browser: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/H/help-page.png})

Figure 6.1068



Command Options

None.

Related Commands

? (6.36.1) - opens the Help dialog box. HelpSearch (6.16.9) - searches the BricsCAD online help system and displays a list of matching topics in a tab of the Web browser.

Figure 6.1069



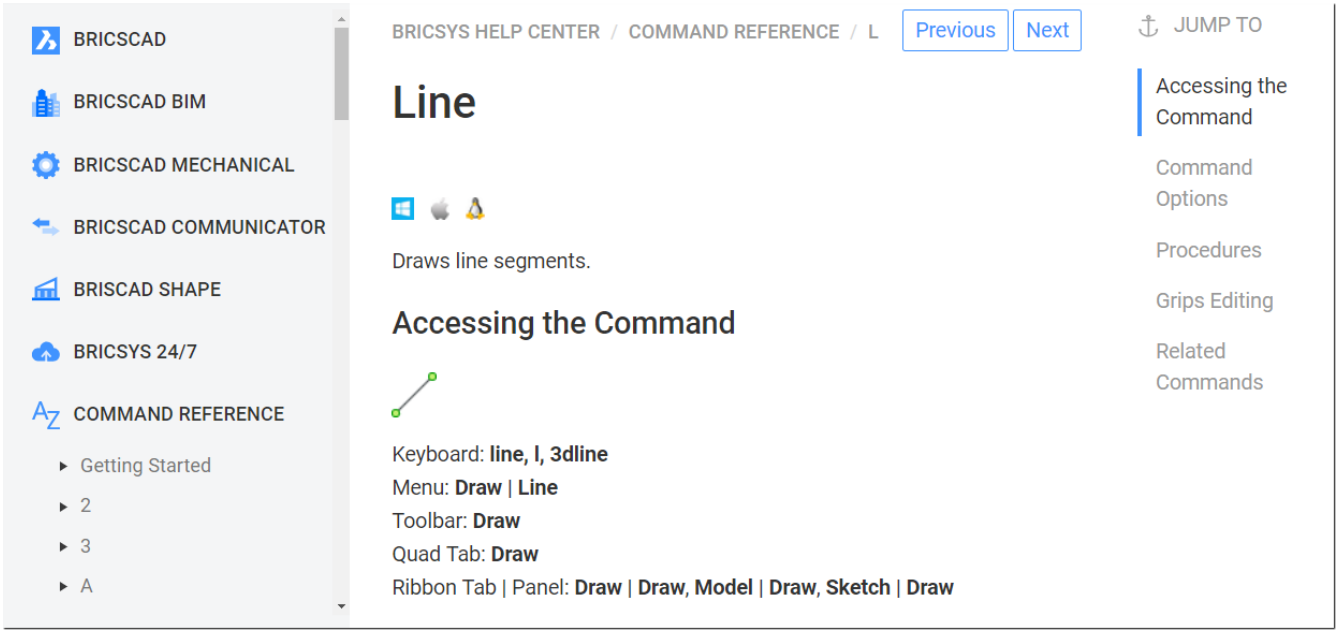
6.16.9 HelpSearch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Searches the BricsCAD online help and then displays the result in your computer's default Web browser.

Accessing the Command

command bar: **helpsearch** : **HelpSearch Prompts you in the command bar:** Search in Help: **(Type a search string)** Displays the result in your computer's default Web browser. For instance, enter **line**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/H/helpsearch-line.png})

Figure 6.1070



Command Options

Option	Description
Search in Help	Enter a term to be searched for, such as the name of a command or a drawing concept

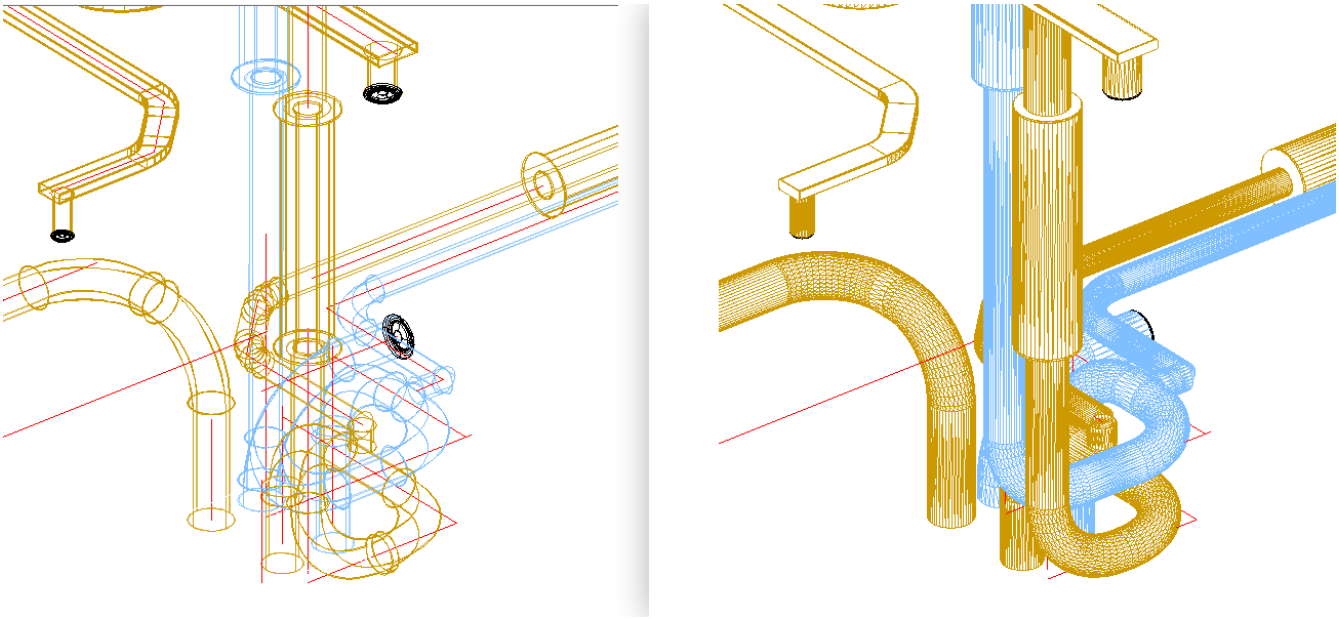
Related Commands

Help (6.16.8) - opens the BricsCAD Help window. ? (6.36.1) - displays the Help window.

Figure 6.1071



Figure 6.1072



6.16.10 Hide

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Removes hidden lines from 3D entities, as shown below. (This is an old command superseded by the Hidden visual style, which produces better results visually; see VisualStyles (6.31.13) command.) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/H/hide-1-eg.png})

Accessing the Command

command bar: **hide** alias: **hi** menu bar: **View | Rendering | Hide** ribbon:**View | Render Modes | Hide** toolbar: **Rendering |** : **Hide** Displays no prompt in the command bar; the program immediately removes hidden lines. To return to wireframe display, change the visual style to 2D Wireframe.

Command Options

No options.

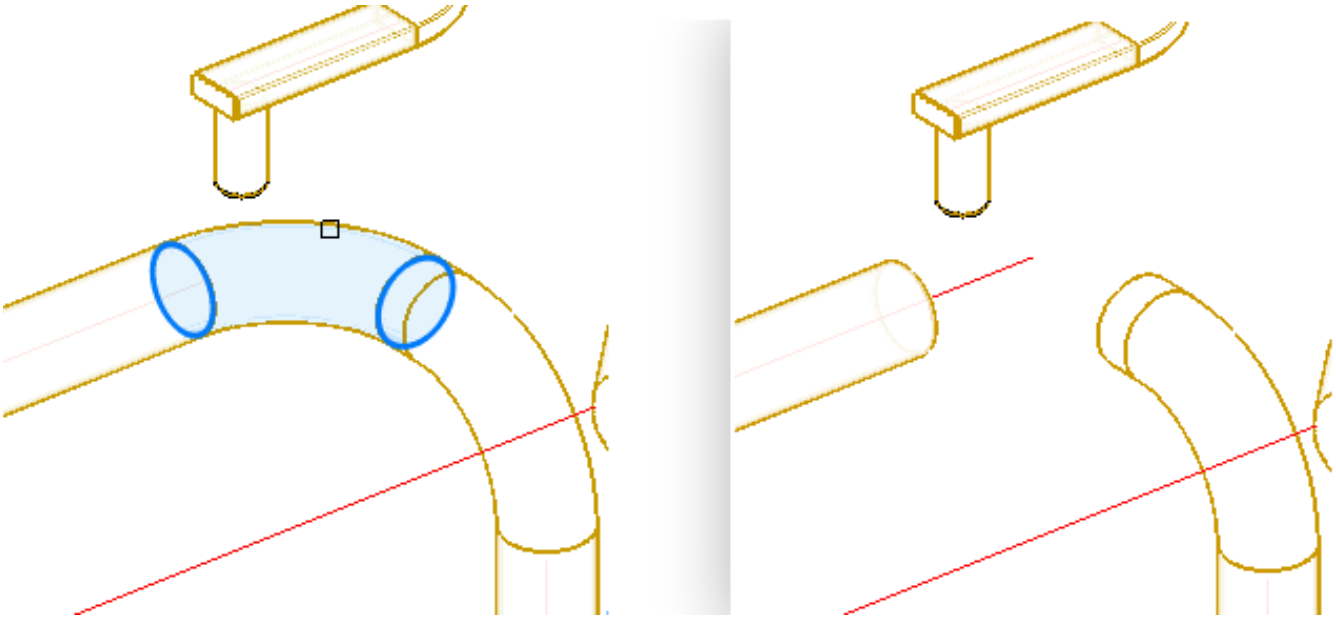
Related Commands

Render (6.26.22) - generates a photorealistic rendering using materials and lights Shade (6.27.27) - generates shaded images of 3D drawings. ShadeMode (6.27.29) - sets the shading style for 3D entities. Solprof (6.27.62) - creates a hidden line representation of 3D solids in a layout viewport. Flatshot (6.14.9) - creates a hidden line representation of all 3D solids in model space as a block or a new drawing. VisualStyles (6.31.13) - creates and controls visual style definitions in the Drawing Explorer. VsCurrent (6.31.21) - sets the visual style at the command line.

Figure 6.1073



Figure 6.1074



6.16.11 HideObjects

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Hides selected entities, like using the Freeze option of the Layer command but for individual entities. Hidden entities optionally stay hidden between drawing sessions. At left, an entity is selected; at right, it is hidden from view by this command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/H/hideobjects-1-eg.png})

Accessing the Command


command bar: **hideobjects** Quad: **Select** ribbon: **Model | View** toolbar: **Hide/Show Entities** : **HideObjects**
Prompts you in the command bar: Select Entities to Hide: **(Select one or more entities)** Entities in set: 1 Select Entities to Hide: **(Select more entities, or else press Enter to end the command.)** The program hides from view the entities you selected; to return them to view, use the UnisolateObjects (6.30.7) command.

Command Options

Option	Description
Select Entities to Hide	Select one or more entities, or else press **Enter** to end the command

The OBJECTISOLATIONMODE system variable controls whether the hidden state is saved or not: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/ObjectIsolationModes.png})

Figure 6.1075

[0] Objects are temporarily hidden for the current session. Interfered solids are selected together with the interference solids.
[1] Objects remain hidden between sessions. Interfered solids are selected together with the interference solids.
[2] Objects are temporarily hidden for the current session. Interfered solids are not selected together with the interference solids.
[3] Objects remain hidden between sessions. Interfered solids are not selected together with the interference solids.

Related Commands

LayFrz (6.20.8) - freezes the layers associated with entities selected in the drawing. LayOff (6.20.12) - turns off layers associated with entities selected in the drawing. IsolateObjects (6.17.21) - hides all but the selected entities. UnisolateObjects (6.30.7) - unhides entities that were hidden.

Figure 6.1076



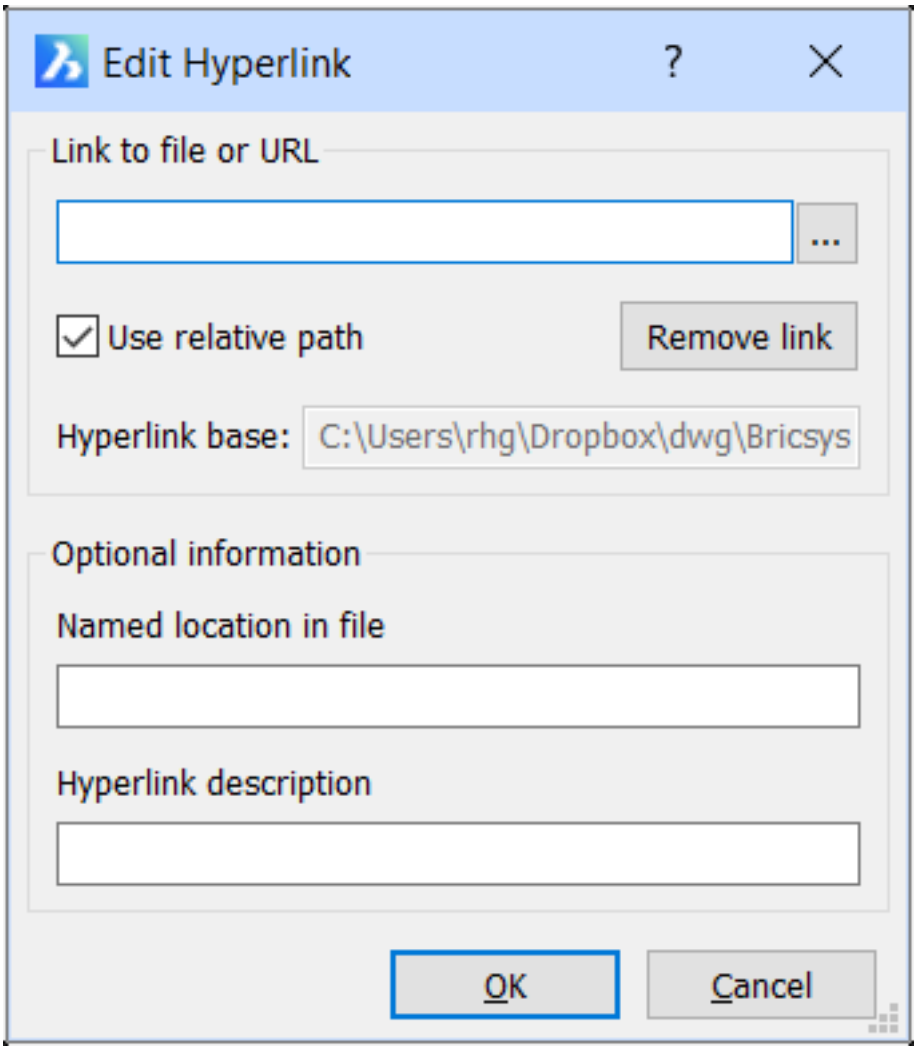
6.16.12 Hyperlink

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Links entities with files and Web pages through a dialog box.

Accessing the Command

command bar: **hyperlink** shortcut: **Ctrl+K** menu bar:**Insert | Hyperlink** : **Hyperlink Prompts you in the command bar:** Select entities: **(Choose one or more entities)** Entities in set: 1 Select entities: **(Choose more entities, or else press Enter to continue command)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V dlg.png}) Change hyperlink options, and then click **OK**.Each entity can contain just one hyperlink.

Figure 6.1077



Command Options

Option	Description
Link to file or URL	Specifies the hyperlink; can be the path and name of a file, networked file, or location on the Internet.Examples of hyperlink syntax • File - C:\foldername\filename.ext • Networked file - \\networkname\c\foldername\filename.ext • URL - https://www.bricsys.com or www.bricsys.com Locations on the Internet can include Web pages (http://), FTP sites (ftp://), and so on.
...	Displays the Select File dialog box: 1. Choose a file name 2. Click **Open** The program establishes a hyperlink between the entity and the file. Any type of file can be linked to.
Use relative path	Determines how the path is stored in the URL: • **Yes** - uses the path specified by **Hyperlink base**: hyperlink base + relative path = full path or URL • **No** - uses the actual path to the file on your computer
Remove link	Removes the link from the entity
Hyperlink base	Specifies the path to relative hyperlinks; available only when **Use Relative Path** is turned on
Named location in file	Specifies a bookmark name.The # character specifies locations (bookmarks) within files or Web pages, such as www.bricscad.sys#bookmark.
Hyperlink description	Describes the purpose of the URL or file; this description is displayed in a optional tooltip when the cursor is paused over the entity, as specified by the HyperlinkOptions (6.16.14) command.

Related Commands

-Hyperlink (6.16.13) - attaches hyperlinks through the command bar. HyperlinkOptions (6.16.14) - sets options for hyperlinks. Settings (6.27.23) - sets options for hyperlinks in the Files section of the Settings dialog box. Browser (6.6.19) and Url (6.30.11) - open the default Web browser. OnWeb (6.23.6) - opens the Bricsys home page in your computer's default Web browser. Mail (6.21.1) - attaches the current drawing to a message in the default email client.

Figure 6.1078



6.16.13 -Hyperlink

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Links entities with files and Web pages at the command prompt.This command is meant for use by macros and LISP routines.

Accessing the Command

command bar: **-hyperlink** transparent: **'-hyperlink** : **-Hyperlink Prompts you in the command bar:** Enter an option [Remove/Insert] <Insert>: **(Type R or I)** Entities can contain just one hyperlink; when you want to insert another hyperlink, BricsCAD asks: Overwrite existing hyperlinks? [Yes/No] <Yes>: **(Type Y or N)**

Command Options

Option	Description
Remove	Removes hyperlinks from entities; prompts you: 1. **Select entities** - **(choose one or more entities.)** 2. **Select entities -(choose more entities, or else press Enter to end the command.)**
Insert	Adds a hyperlink to one or more entities; prompts you: 1. **Select entities** - **(choose one or ore entities to which hyperlinks are to be added.)** 2. **Select entities -(choose more entities, or else press Enter to continue the command.)** 3. **Enter URL** - **(specify the path and name of the file, networked file, or location on the Internet.)** 4. **Enter named location** - **(specifies a bookmark name; press Enter to ignore this option.)** 5. **Enter description** - **(describes the hyperlink; press Enter to leave empty.)**

Overwrite Option	Description
Yes	Replaces the existing hyperlink with the new one
No	Leaves the existing hyperlink as it is; does not replace it

See the Hyperlink (6.16.12) command for example URLs.

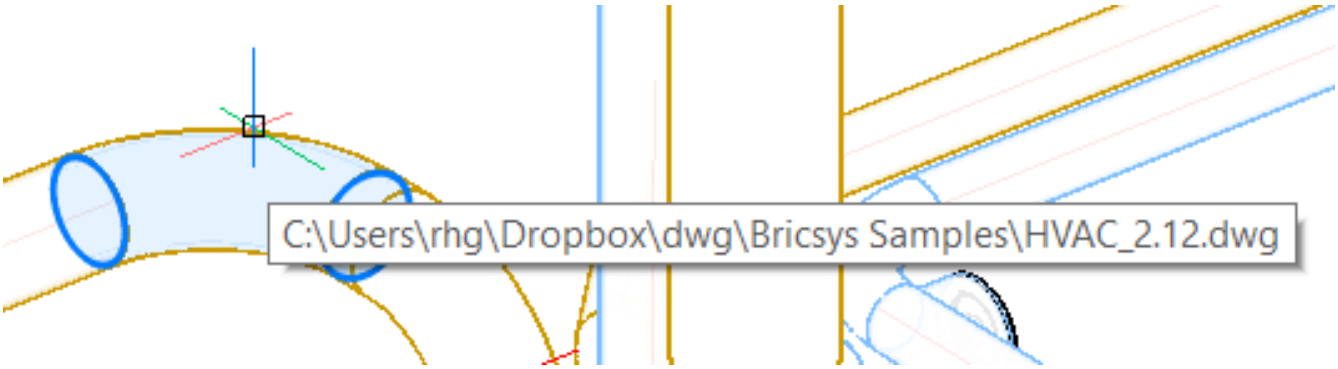
Related Commands

Hyperlink (6.16.12) - attaches hyperlinks through a dialog box. HyperlinkOptions (6.16.14) - sets options for hyperlinks. Settings (6.27.23) - sets options for hyperlinks in the Files section of the Settings dialog box. Browser (6.6.19) and Url (6.30.11) - open the default Web browser. OnWeb (6.23.6) - opens the Bricsys home page in your computer's default Web browser. Mail (6.21.1) - attaches the current drawing to a message in the default email client.

Figure 6.1079



Figure 6.1080



6.16.14 HyperlinkOptions

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the display of the hyperlink cursor and URL tooltip (shown below), and adds Hyperlink to the shortcut menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/H/hyperl

Accessing the Command

command bar: **hyperlinkoptions** transparent: **'hyperlinkoptions** : **HyperlinkOptions Prompts you in the command bar:** Display hyperlink cursor, tooltip, and shortcut menu? [Yes/No] <Yes>: **(Type Y or N)**

Command Options

Option	Description
Display hyperlink cursor, tooltip, and shortcut menu	Toggles the display of the ”hyperlink” icon, tooltip displaying the URL, and the Hyperlink sub-menu on shortcut menus: • **Yes** - turns on the display • **No** - turns off the display When the cursor passes over entities with hyperlinks, the ”world” icon is displayed. The text of the tooltip is specified by the Hyperlink (6.16.12) command’s Hyperlink Description option. When you right-click a selected entity that contains hyperlinks, the **Hyperlink** item is added to the menu; see below.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image163.jpg})

Hyperlink Cursor	Description
Open **url**	Opens the location specified by the URL: • Location on the Internet - BricsCAD opens the page in the default Web browser • File - BricsCAD opens the file with the associated application This option executes the URL command; see URL (6.30.11) command.
Copy Hyperlink	Copies the URL to the Clipboard; the URL can be pasted into the drawing or other documents with the Ctrl+V shortcut (PasteClipcommand)
Edit Hyperlink	Opens the Edit Hyperlink dialog box for editing the URL, description, and other parameters.See Hyperlink

Related Commands

-Hyperlink (6.16.13) - attaches hyperlinks through the command bar. Hyperlink (6.16.12) - attaches hyperlinks through a dialog box. Settings (6.27.23) - sets options for hyperlinks in the Files section of the Settings dialog box. Browser (6.6.19) and Url (6.30.11) - open the default Web browser. OnWeb (6.23.6) - opens the Bricsys home page in your computer's default Web browser. Mail (6.21.1) - attaches the current drawing to a message in the default email client.

6.17 I

6.17.1 Id

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Reports the x,y,z coordinates of points you pick using the current coordinate system.

Figure 6.1082



Accessing the Command

 command bar: **id** transparent: **'id** alias: **idpoint** menu bar: **Tools | Inquiry | ID Coordinates** ribbon: **Tools | Inquiry | ID Coordinates** toolbar: **Inquiry |**  : **Id Prompts you in the command bar:** Select point to identify coordinates: **(Pick a point in the drawing)** X=x.xxxx Y=y.yyyy Z=z.zzzz Use entity snaps (6.23.12) to pick points accurately on geometric features.

Command Options

Option	Description
Select point to identify coordinates	Specifies the point to be reported

Related Commands

Area (6.4.13) - reports the area, circumference, and perimeter of entities. Dist (6.10.31) - reports the 2D and 3D distances between two points. List (6.20.35) - lists the area and other parameters of entities in the command bar. MassProp (6.21.4) - reports the properties of solid models and regions. Properties (6.24.49) - lists the area and other parameters of entities in a palette.

Figure 6.1083



6.17.2 IfcExport

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Exports a BIM model to IFC, partially or entirely.

Accessing the Command

Keyboard:**ifcexport : IfcExport Prompts you:** Select entities to export [Entire model] <Entire model>: **(Select the entities to be exported or press Enter to export the entire model.)** The IFC File dialog box displays. Type a name in the File Name field, then click the Save button.

Command Options

No command options.

Settings

The following User Preferences (1.13.11) control the behavior of the command:

Name	Title	Description
IfcExportBaseQuantities	Export base quantities	Export the derived base quantities from BIM entities. If this setting is enabled , calculated quantities (e.g. length, thickness, height) for each BIM element can be exported to IFC.
IfcExportElementsOnOffAndFrozenLayers	Export elements on OFF and FROZEN layers	Also export entities on layers which are currently OFF or Frozen. By default, this setting is enabled.
IfcExplodeExternalReferences	Explode external references	Explode external references in the IFX spatial structure. If this setting is enabled , each external reference will be divided into the elements (e.g.**IfcSite, IfcBuilding)** based on the spatial project structure.
IfcExportMultiPlyElementsAsAggregated	Export mutli-ply elements as aggregated elements	Export mutli-ply elements as aggregated elements. If this setting is enabled , each ply material belonging to a building element (e.g. wall, column) is calculated and exported individually
IfcExportTessalation	Level of tesselation	Controls the level of tesselation of the exported geometry. The options are: • **0** or **Current**: no regeneration,the faceting is exported as set by the FACETRES system variable or the Modeler properties. • **Low**, **Medium** or **High**:the faceting is regenerated, which takes longer.

Related Commands

Export (6.17.5) - exports drawings in a variety of vector and raster formats. Import (6.17.10) - imports the geometry from external files into the current drawing.

Figure 6.1084



6.17.3 Image

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Attaches and adjusts images attached to drawings through the Drawing Explorer; supports the following formats: **BMP** - Windows bitmap **ECW** - Enhanced Compressed Wavelet format **GIF** - CompuServe graphic image format **JPG**, **JPEG**, and **JPEG2000** (JP2, j2k) - Joint Photographic Experts Group (common among digital cameras) **MTI** - Multi-resolution Tiled Image (Bricsys raster format for large images) **PCX** - PC Paintbrush **PNG** - Portable network graphics **SID** - MrSID is an acronym for Multi-resolution Seamless Image Database **TGA** - Targa

Accessing the Command

 command bar: **image** aliases: **expimages, im** menu bar: **Tools | Drawing Explorer | Images** ribbon:**Home | Settings | Drawing Explorer | Images Insert | Images | Image Management** toolbar: **Images |**  : **Image** Displays the Drawing Explorer, opened to the Images node.When no images are loaded into the current drawing, then the Images section is blank. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/image-dlg.png}) Drawing Explorer when one or more images are attached to the current drawing: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/images-2-dlg.png}) Press **F1** to access help. **Note:** You can control all attachments through the AttachmentsPanelOpen (6.4.26) command, and adjust the properties of images with the Properties (6.24.49) command.

Figure 6.1085

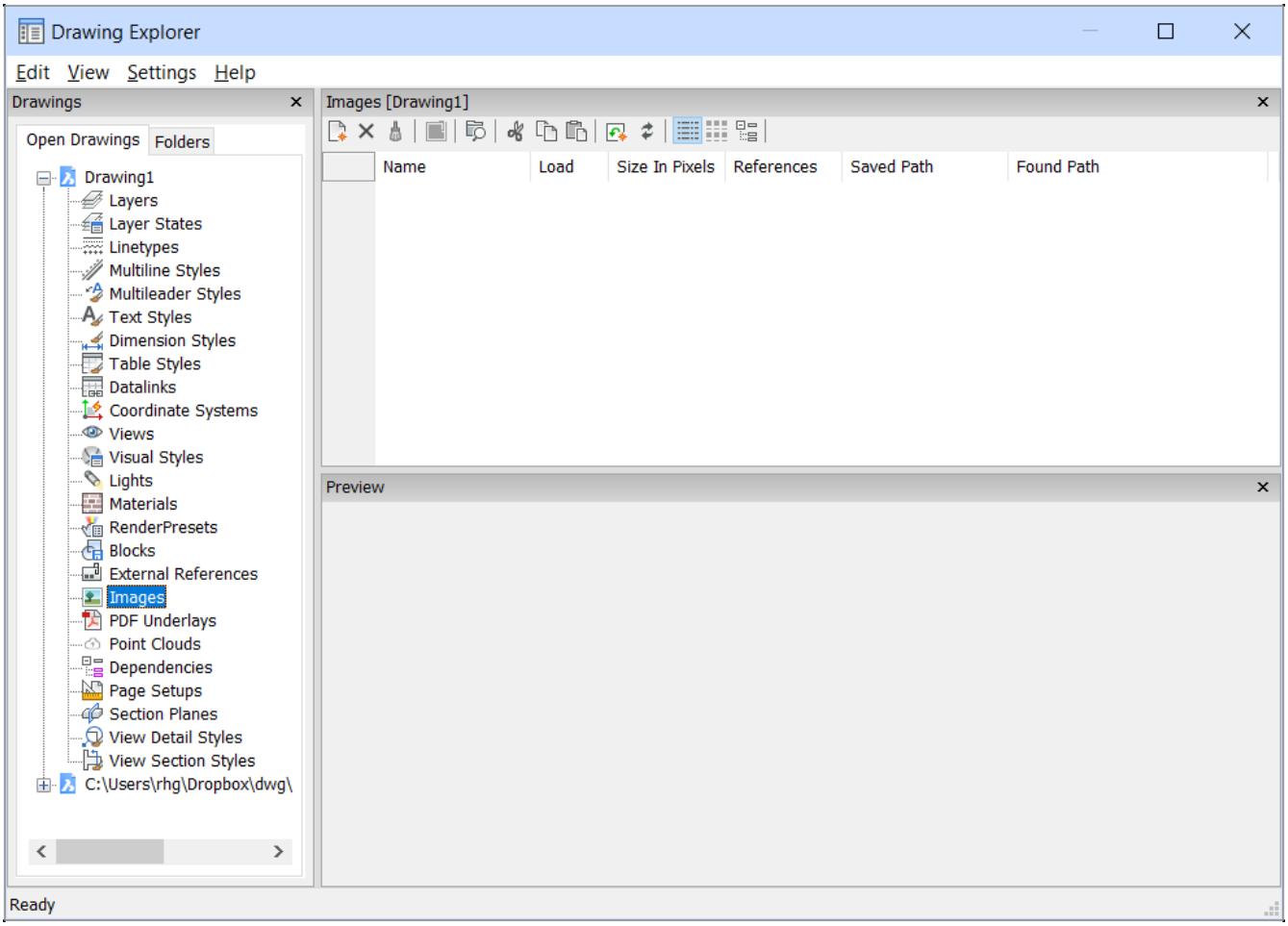
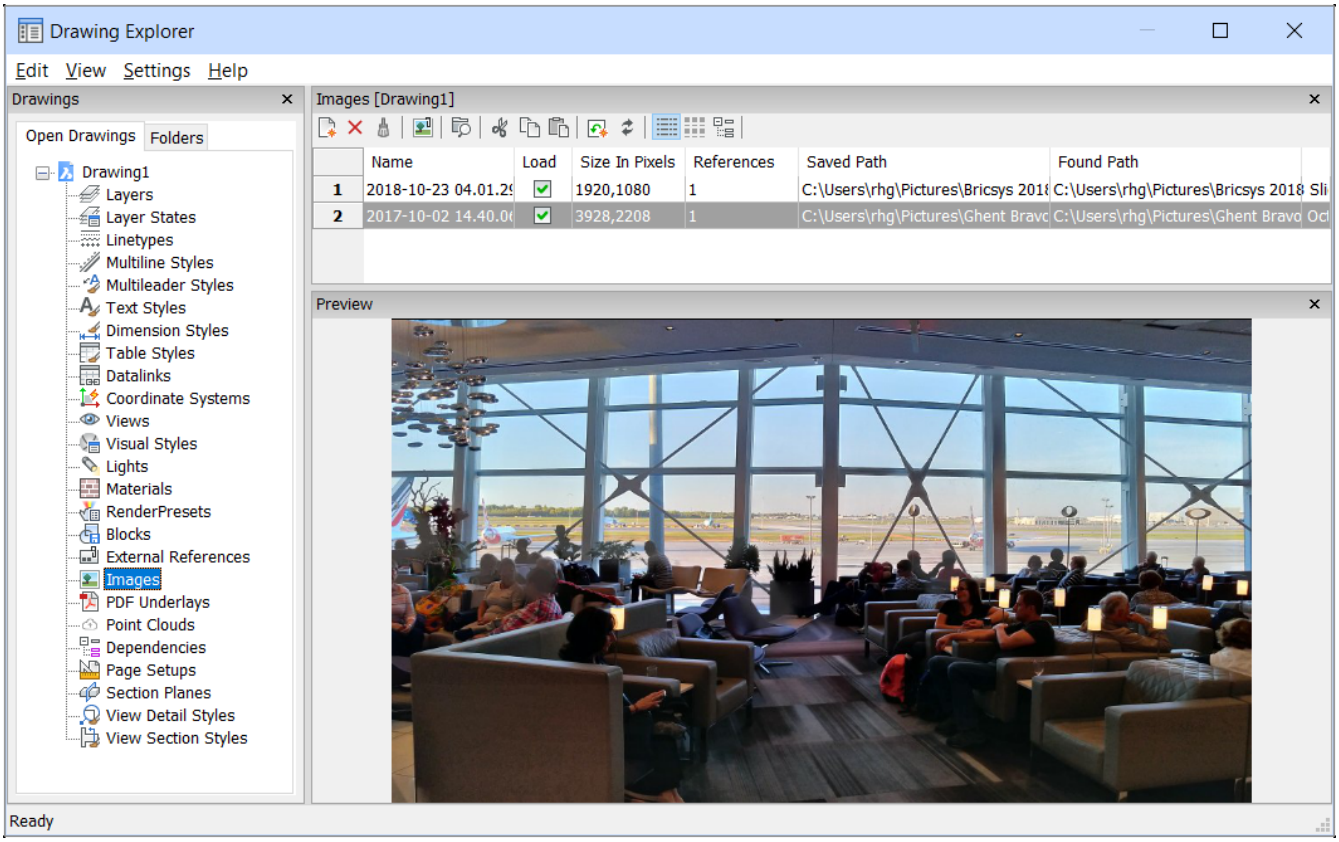


Figure 6.1086



Command Options

Toolbar Option	Description
New	Attaches raster image files to the drawing: 1. Click **New**. 2. In the Select Image File dialog box, choose an image file, and then click **Open**. 3. In the Attach Raster Image dialog box, modified the properties, and then click **OK**. 4. To close the Drawing Explorer, click **X**. See ImageAttach (6.17.5) command.
Delete	Removes the image from the drawing.You may need to use the Regen (6.26.15) command to update the screen.
Insert	Inserts more images in the drawing, like the New button; displays the Attach Raster Image dialog box. See ImageAttach (6.17.5) command.
Extended Search	Runs an extended search for missing attachments. (Not implemented.)
Cut	Cuts the image to the Clipboard, and removes it from the current drawing: 1. Choose the image file 2. Select **Cut** 3. Switch to the Image section of another drawing in the Drawing Explorer 4. Select **Paste**
Copy	Copies the image to the Clipboard. You can paste it in the Drawing Explorer's Image section of another drawing.
Paste	Pastes images and related data.
Regen	Updates the drawing to reflect changes made in this dialog box.
Regen Toggle	Toggles automatic drawing updates: • **On** - updates drawing automatically • **Off** - updates drawing only when the Redraw button is clicked
Details	Displays the details of each image. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Icons	Displays the image files as small icons, hiding their details. Not implemented.
Tree	Displays the relationship between files in an Explorer tree-like fashion. "Current" refers to the current drawing. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/

Detail View Options	Description
Name	Specifies the name of the image. This usually is the image's file name.
Load	Toggles whether the image is displayed in the drawing: • **On** - displays the image in the drawing • **Off** - displays only the image's frame, which is turned off by the ImageFrame (6.17.8) command
Size in Pixels	Reports the size of the image in pixels, horizontally and vertically.
References	Reports the number of times the image occurs in the drawing.
Saved Path	Specifies the path to the image file to attach a different version of the image or to find an image disconnected from the drawing: • ... button - changes the path
Found Path	Reports the path to the image file

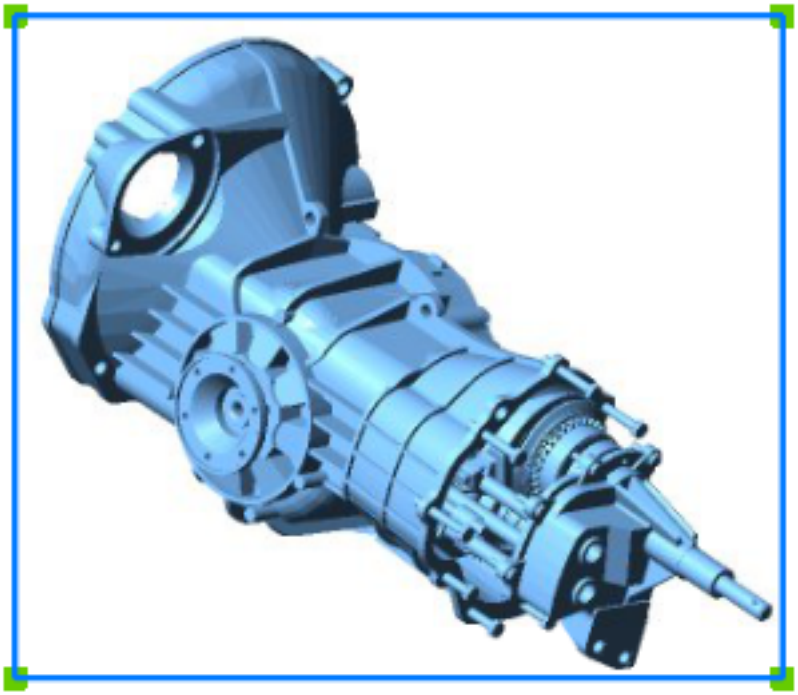
Grips Editing

Images can be edited directly through grips:

1. Select the image. Notice that its frame has four grips.
2. Drag a grip to stretch (resize) the image.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/ImageGrips.png})

Figure 6.1087



Procedures

- Open the Images Explorer (1.24.1)
- Images Explorer display options (1.24.1)
- Placing images in a drawing (1.24.1)
- To load / Unload images (1.24.1)
- Detaching images (1.24.1)

Related Commands

ImageAttach (6.17.5) - places raster images in drawings. -ImageAttach (6.17.6) - attaches raster images to the drawing; displays prompts in the command bar. ImageAdjust (6.17.4) - changes the properties of images through the Properties palette. ImageClip (6.17.7) - clips images. ImageFrame (6.17.8) - toggles the display of the frame around images. ImageQuality (6.17.9) - toggles the quality of the image. Transparency (1.17.8) - toggles the transparency of monotone images. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.1088

	Name	Loaded	Size In Pixels	References	Saved Path	Found Path
1	DEME	☑	1453,915	1	C:\BricsCAD Training\Bricscad\TrainingCD	C:\BricsCAD Training\Bricscad\TrainingCD\NL\Oefeningen\DEME.png
2	gearbox	☑	646,557	1	C:\BricsCAD Training\Bricscad\TrainingCD	C:\BricsCAD Training\Bricscad\TrainingCD\NL\Oefeningen\gearbox.png

Figure 6.1089

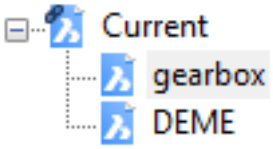


Figure 6.1090



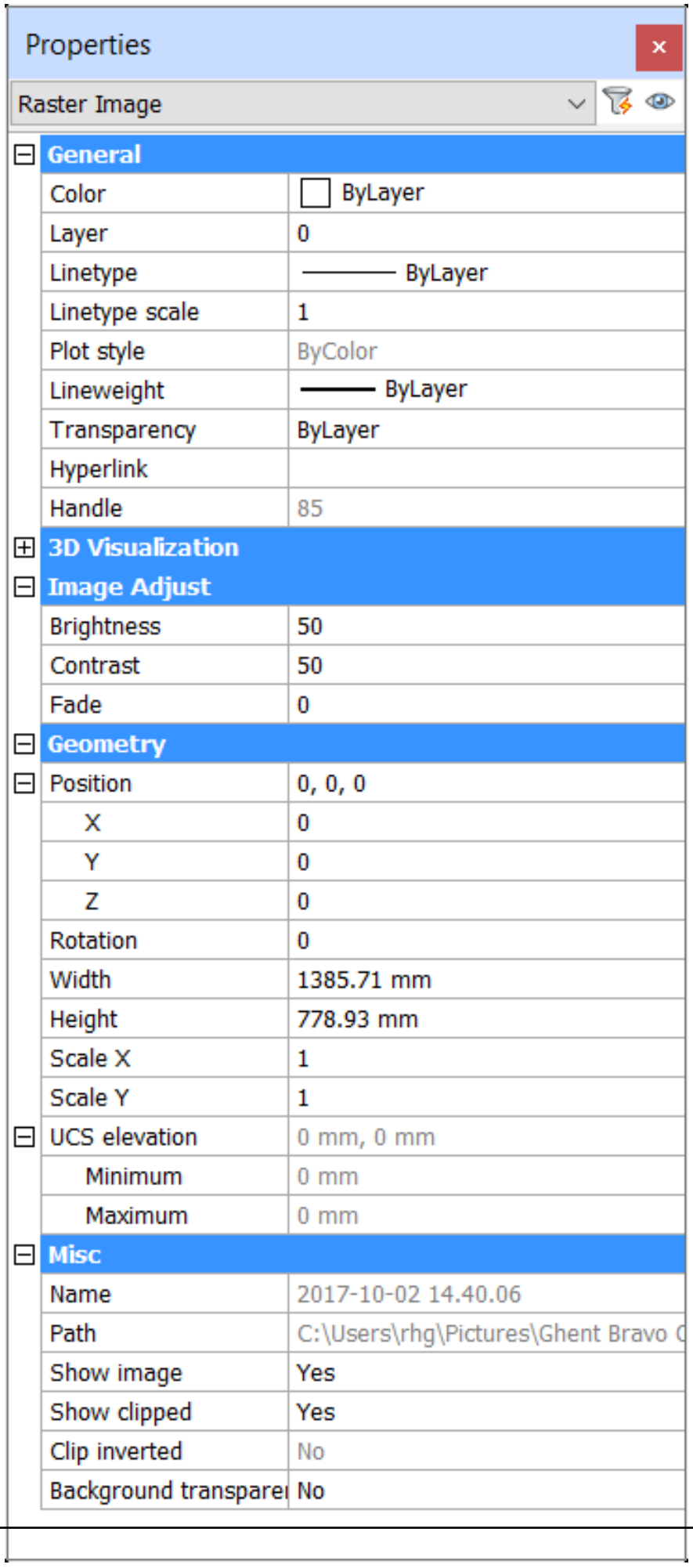
6.17.4 ImageAdjust

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adjusts the properties of images through the Properties palette.

Accessing the Command

 command bar: **imageadjust** transparent: **'imageadjust** shortcut: **double-click the image frame** alias: **iad** ribbon:**Insert | Images | Image Adjust** toolbar: **Images |**  : **ImageAdjust Prompts you in the command bar:** Select entities: **(Choose one or more image entities by their frames)** Select entities: **(Choose more entities, or else press Enter to continue)** Displays the Properties panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Com dlg.png}) As you change properties in this panel, the program applies them immediately to the selected image(s).

Figure 6.1091



Command Options

Option	Description
Select entities	Selects the images to be adjusted.
Properties Panel Options	**Description**
General Properties	**The following properties affect the frame that surrounds the image:**
Color	Specifies the color of the frame. See Color (6.9.33) command.
Layer	Specifies the layer of the frame.When the image is placed on its own layer, you can use layer visibility to toggle the display of the image and its frame.See Layer (6.20.2) command. Alternatively, use the ImageFrame (6.17.8) command to toggle the display of the frame independently of the image.
Linetype	Specifies the linetype of the frame. See Linetype (6.20.33) command.
Linetype Scale	Specifies the linetype scale of the frame. See LTSCALE (1.17.6) system variable.
Plot Style	Specifies the plot style of the frame.This option is available only when plot styles are enabled in the drawing, See PlotStyle (6.24.29) command.
Lineweight	Specifies the lineweight of the frame. See LWeight (6.20.41) command.
Hyperlink	Specifies a URL or path to a file. See Hyperlink (6.16.12) command.
Image Adjust	**The following properties affect the image:**
Brightness	Lightens and darkens the image: • **0** - very dark or black • **50** - normal • **100** - very light or white
Contrast	Increases and decreases the contrast: • **0** - very low contrast • **50** - normal • **100** - very high contrast
Fade	Fades the image: • **0** - no fade • **100** - fully faded
Geometry Options	**The following properties affect the image:**
Position	Specifies the x, y, and z coordinates of the image's lower right corner.When you change the coordinates, the lower right corner is marked with a large X. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Rotation	Specifies the rotation angle about the position point: • **Positive angles** - rotate the image counter-clockwise • **Negative angles** - rotate in the image clockwise
Width	Specifies the width of the image, in current unit

Height	Specifies the height of the image, in current units
Scale	Specifies the scale factor of the image: • **Less than 1** - makes the image smaller. • **More than 1** - makes the image larger.

Misc Options	**Description**
Name	Reports the name of the image; cannot be edited by the user
Path	Reports the path to the image file; cannot be edited by the user
Show Image	Toggles display of the image: • **Yes** - shows the image; this can be overridden by setting the layer to Frozen • **No** - hides the image
Show Clipped	Toggles clipping: • **Yes** - displays clipped image, if clipping has been previously applied by ImageClip (6.17.7) command; part of the image is hidden. • **No** - hides clipped portion(s); the full image is displayed
Clip Inverted	Inverts the clip to flip which part of the image is shown • **Yes** - inverts the clipping of the image • **No** - retains the original cklipping This option is available only when the image was previously clipped by ImageClip (6.17.7) command
Background Transparency	Toggles the background transparency: • **Yes** - makes the background color of the image • **No** - keeps the image nontransparent Transparency makes one color of the image see-through. See theTransparency (6.29.25) command.

Related Commands

Image (6.17.3) - controls the parameters of the image file. ImageAttach (6.17.5) - places raster images in drawings. -ImageAttach (6.17.6) - attaches raster images to the drawing; displays prompts in the command bar. ImageClip (6.17.7) - clips images. ImageFrame (6.17.8) - toggles the display of the frame around images. ImageQuality (6.17.9) - toggles the quality of the image. Transparency (1.17.8) - toggles the transparency of monotone images.

Figure 6.1092



Figure 6.1093



6.17.5 ImageAttach

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Attaches raster images of the following formats to the drawing through dialog boxes: **BMP** - Windows bitmap **ECW** - Enhanced Compressed Wavelet format **GIF** - CompuServe graphic image format **JPG**, **JPEG**, and **JPEG2000** (JP2, j2k) - Joint Photographic Experts Group (common among digital cameras) **MTI** - Multi-resolution Tiled Image (Bricsys raster format for large images) **PCX** - PC Paintbrush **PNG** - Portable network graphics **SID** - MrSID is an acronym for Multi-resolution Seamless Image Database **TGA** - Targa **TIF** and **TIFF** - Tagged Image File Format (commonly used by desktop publishing) **BIGTIFF**: TIFF variant file format which uses 64-bit offsets and supports much larger files.To allow fast display operations, BigTiff images are cached in the BricsCAD proprietary MTI format. Until this cache generation is completed, the image frame is displayed.

Accessing the Command

 command bar: **imageattach** transparent: **'imageattach** alias: **iat** menu:**Insert | Attach Raster Image** ribbon:**Insert | Images | Attach Raster Image** toolbar: **Images |**  : **ImageAttach** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/imageattach-1-dlg.png}) Choose an image, and then click **Open**. Displays a second dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
2-dlg.png}) Specify options, and then click **OK**.

Figure 6.1094

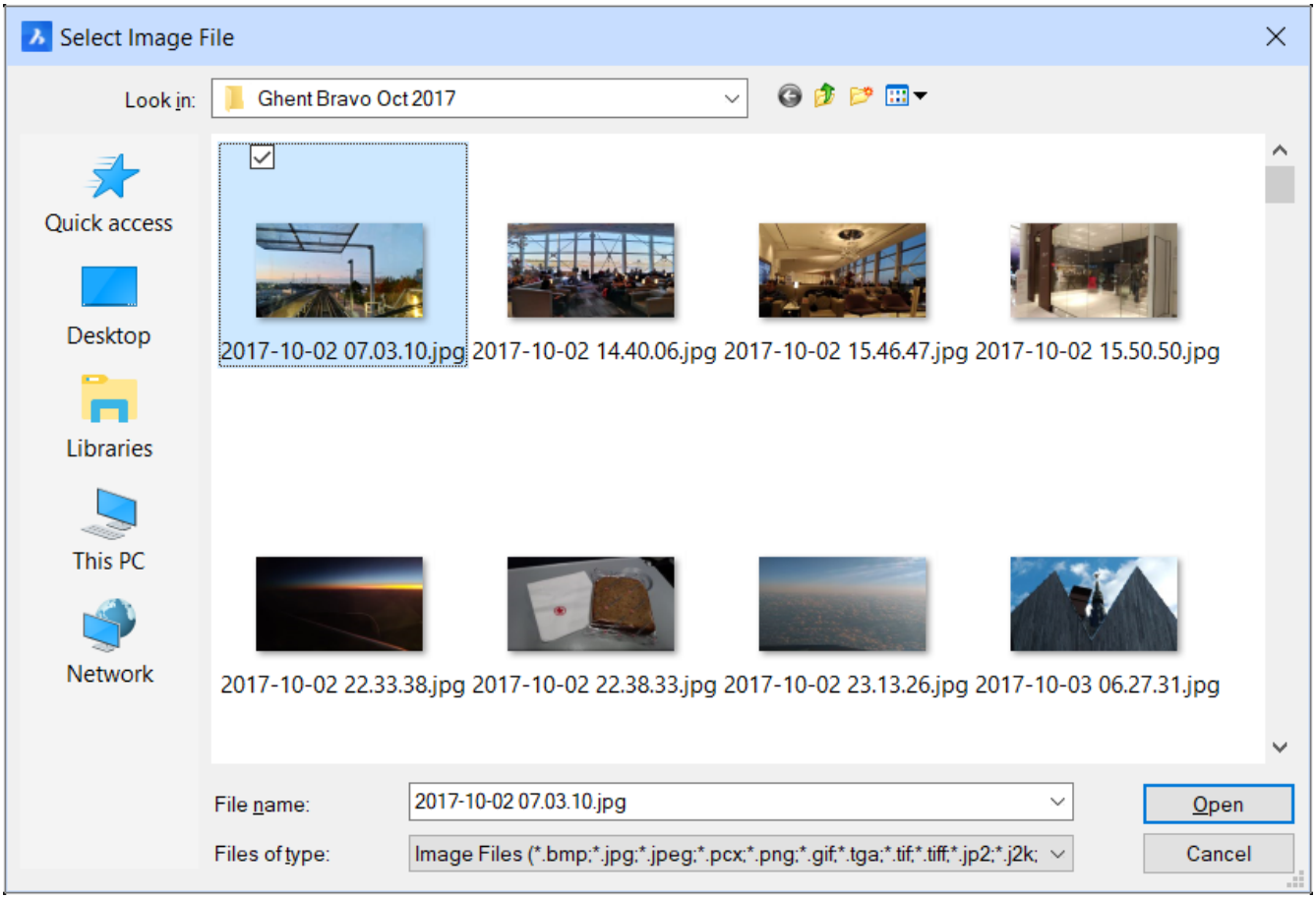
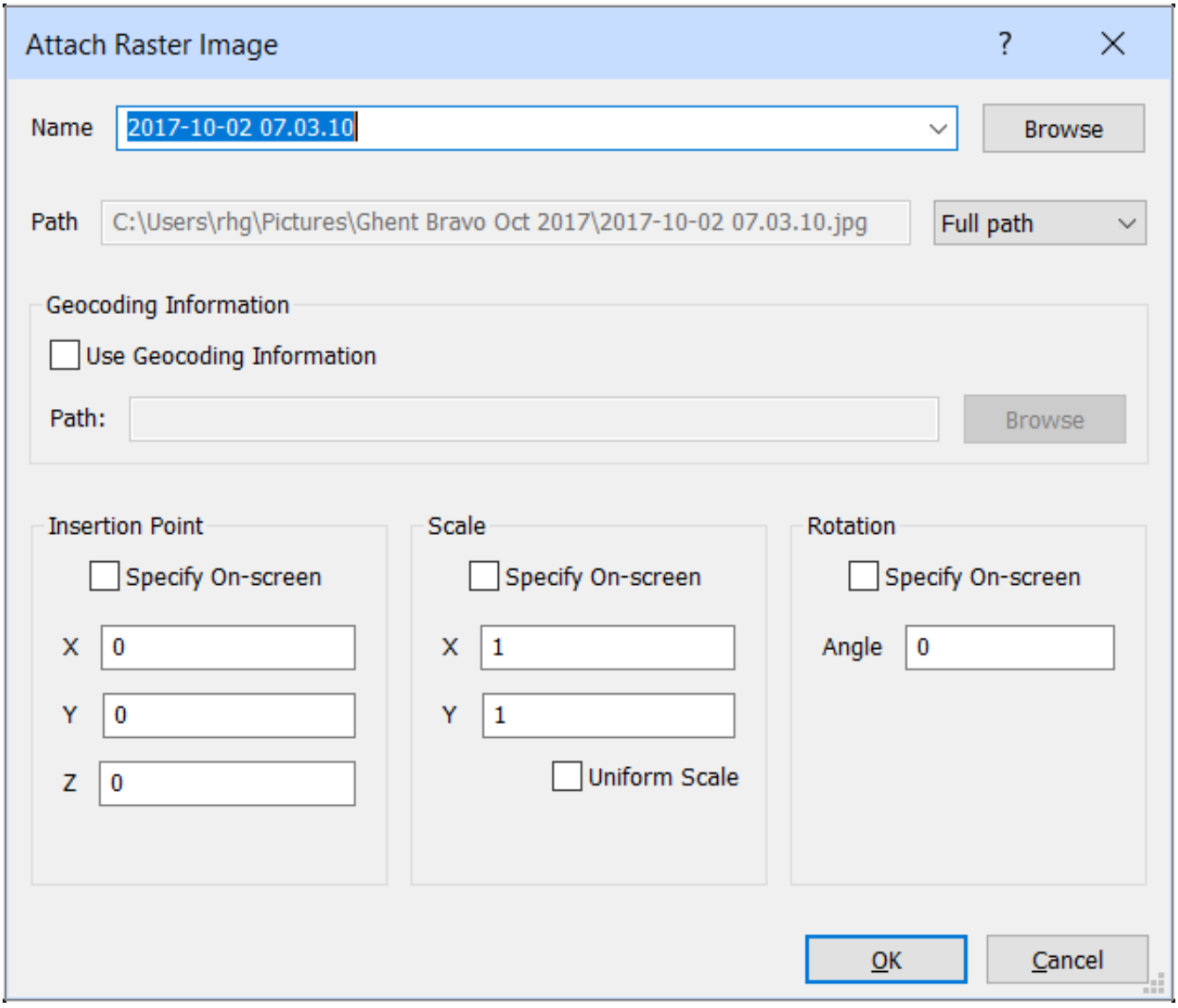


Figure 6.1095



Command Options

Options	Description
Name	Specifies the file name of the image.
Browse	Displays the Open dialog box; click this button to choose a different raster file. Choose a raster file, and then click Open .
Path	Reports the path to the image file; cannot be edited by the user.
Full Path	Determines how much of the path is stored: • Full Path - stores the full path to the file, including drive and folder names. For example: c:\foldername\imagename.ext • Relative Path - stores the path from the drawing's location to the image's location; the drawing must be saved before this option can be selected. For example: ..\foldername\imagename.ext where .. refers to the folder above the current one. • No path - the path is discarded.
Geocoding Information	Position files or world files have an extension that depends on what kind of image file they are associated with. See explanatory notes below: "About Geocoding Information".
Use Geocoding Information	Toggles use of a position file, which specifies the size, location, and rotation of the image file: • Yes - uses a position file. • No - does not use a position file.
Path	Reports the path to the position file.
Browse	Displays the Select Image Position File dialog box; choose a TWF file, and then click Open .
Insertion Point	Positions the lower left corner of the image in the drawing. This option is unavailable when a position file is used.
Specify On-screen	Toggles where you specify the insertion coordinates: • Off - Specifies the x,y,z coordinates in the dialog box. • On - Specifies the coordinates after you click OK, in the command bar: **Insertion point - (specify the x,y or x,y,z coordinates of the image's lower left corner.)**
XYZ	Specifies the x-, y- and z-coordinate of the image's lower left corner.
Scale	Sizes the image. This option is unavailable when a position file is used.
Specify On-screen	Toggles where you specify the scale factors: • Off - Specifies the x,y,z scales in the dialog box. • On - Specifies the scales in the command bar after you click OK : **Scale factor - (specify the scale factor.)**

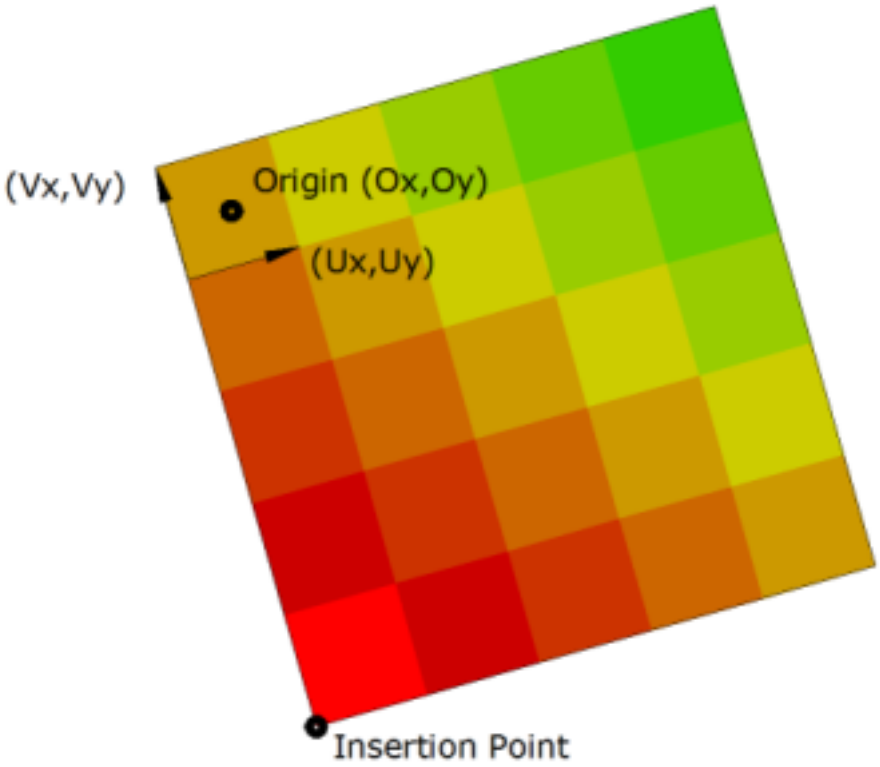
XY	Specifies the scale factor in the x and y direction: • Values larger than 1 make the image larger. • 1 inserts the image at actual size. • Values smaller than 1 make the image smaller.
Uniform Scale	Makes the Y scale factor equal to X.
Rotation Specify On-screen	Rotates the image about the insertion point. Toggles where you specify the rotation angle: • Off - Specifies the angle in the dialog box. • On - Specifies the angle in the command bar after you click OK :. Prompts you: **Rotation angle** - **(specify the angle.)**
Angle	Specifies the rotation angle about the insertion point. • Positive angles rotate the image counterclockwise. • Negative angles rotate the image clockwise.

About Geocoding Information

Position files (or world files) are plain ASCII text files that position raster files in the drawing. They are useful for positioning images according to geographic coordinates, such as latitude and longitude. They can be created using text editors such as **Notepad** . The format is a list of 6 numbers that fully describe location, scale and orientation of the top left pixel of the raster file to be attached. If **U** is the vector along the width, **V** is the vector along the height, and **O** is the center of the pixel (see image below): (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/C then the list consists of: Ux Uy Vx Vy Ox Oy Typically, the first and last letters of the image extension are used with a ”w” added at the end. Sometimes the full image extension with a ”w” at the end is used also. The following position file formats are supported:

- *.bpw (for BMP images)
- *.jgw (for JPG images)
- *.gfw (for GIF images)
- *.tfw (for TIFF images)
- *.j2w (for JPEG2000 images)
- *.sdw (for SID images)
- *.pgw (for PNG images)

Figure 6.1096



Procedures

- Placing images in a drawing (1.24.1)

Related Commands

Image (6.17.3) - controls the parameters of the image file. -ImageAttach (6.17.6) - attaches raster images to the drawing; displays prompts in the command bar. ImageAdjust (6.17.4) - changes the properties of images through the Properties palette. ImageClip (6.17.7) - clips images. ImageFrame (6.17.8) - toggles the display of the frame around images. ImageQuality (6.17.9) - toggles the quality of the image. Transparency (1.17.8) - toggles the transparency of monotone images.

Figure 6.1097



6.17.6 -ImageAttach

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Attaches raster images to the drawing; displays prompts in the command bar.

Accessing the Command

command bar: **-imageattach** : **-ImageAttach Prompts you in the command bar:** Image path will be stored as Full Path. Set image path mode [Full path/Relative path/No path] <Image path>: **(Enter the path and file name of the image file, or else choose a path option)** Use geocoding information [use Geocoding info] <Insertion point>: **(Pick an insertion point for the image, or else type G)** Scale [Size/XY scales] <1.0000>: **(Pick a point to specify the size of the image, enter a value for the scale factor, press Enter to accept the default scale, or else choose an option)** Rotation angle <0>: **(Pick a point to specify the rotation of the image, enter a value for the rotation angle, or else press Enter to accept the default value)**

Command Options

Option	Description
Set image path mode	Specifies how the program should remember the path to the image, which is saved in the drawing: • **Full path** - the full path of the image file is saved, such as D:\BricsCAD Training\EN\Exercises\Gearbox.png • **Relative path** - the path relativeto the drawing folder is saved, such as ..\Exercises\Gearbox.png • **No path** - the path is not saved, and so the program searches for the image in the drawing folder
Use geocoding information	Uses geocoding data to determine the image's insertion point, scale, and rotation angle;Prompts you: **Embedded geocoding information/Geocoding file path [Embedded]<Path\ImageName.pgw>** - **(enter an option)**: • **Embedded geocoding information** - use the geocoding information embedded in the image file • **Geocoding file path** - specify the name of a PGW file that contanis the geocoding data
Scale	Specifies the scale factor of the image; enter a number: • **Larger than 1.0** - increases the size of the image • **1.0** - image is inserted at its actual size • **Smaller than 1.0** - decreases the size of the image • Press **Enter** to accept the default Scale factor cannot be 0 or a negative value.This option maintains the image's aspect ratio.

XY scales	Specifies the scale factors for the X and Y directions independently; prompts you: **Specify X scale factor <1.0000>** - **(enter an option)**: • Press **Enter** to accept the default value • Pick a point in the drawing; the base point is the point you picked earlier for the insertion point • Type a positive value for the X scale factor **Specify Y scale factor <Equal to X scale factor <221.8098>>** - • Press **Enter** to make the Y scale factor the same as the X • Pick a point in the drawing; the base point is the point you picked earlier for the insertion point • Type a positive value for the Y scale factor
Size	Sizes the image interactively; the first point was the point you picked earlier for the insertion point. Prompts you: **Specify opposite corner** - **(pick a point in the drawing.)** This option maintains the image's aspect ratio.
Rotation Angle	Specifies the rotation angle; prompts you: **Rotation angle <0>** - **(press Enter to accept the default 0° rotation, or else pick a point to show the rotation, or type a value.)**

Related Commands

ImageAttach (6.17.5) - attaches raster images to the drawing; displays a dialog box. Image (6.17.3) - controls the parameters of the image file. ImageAdjust (6.17.4) - changes the properties of images through the Properties palette. ImageClip (6.17.7) - clips images. ImageFrame (6.17.8) - toggles the display of the frame around images. ImageQuality (6.17.9) - toggles the quality of the image. Transparency (1.17.8) - toggles the transparency of monotone images.

Figure 6.1098



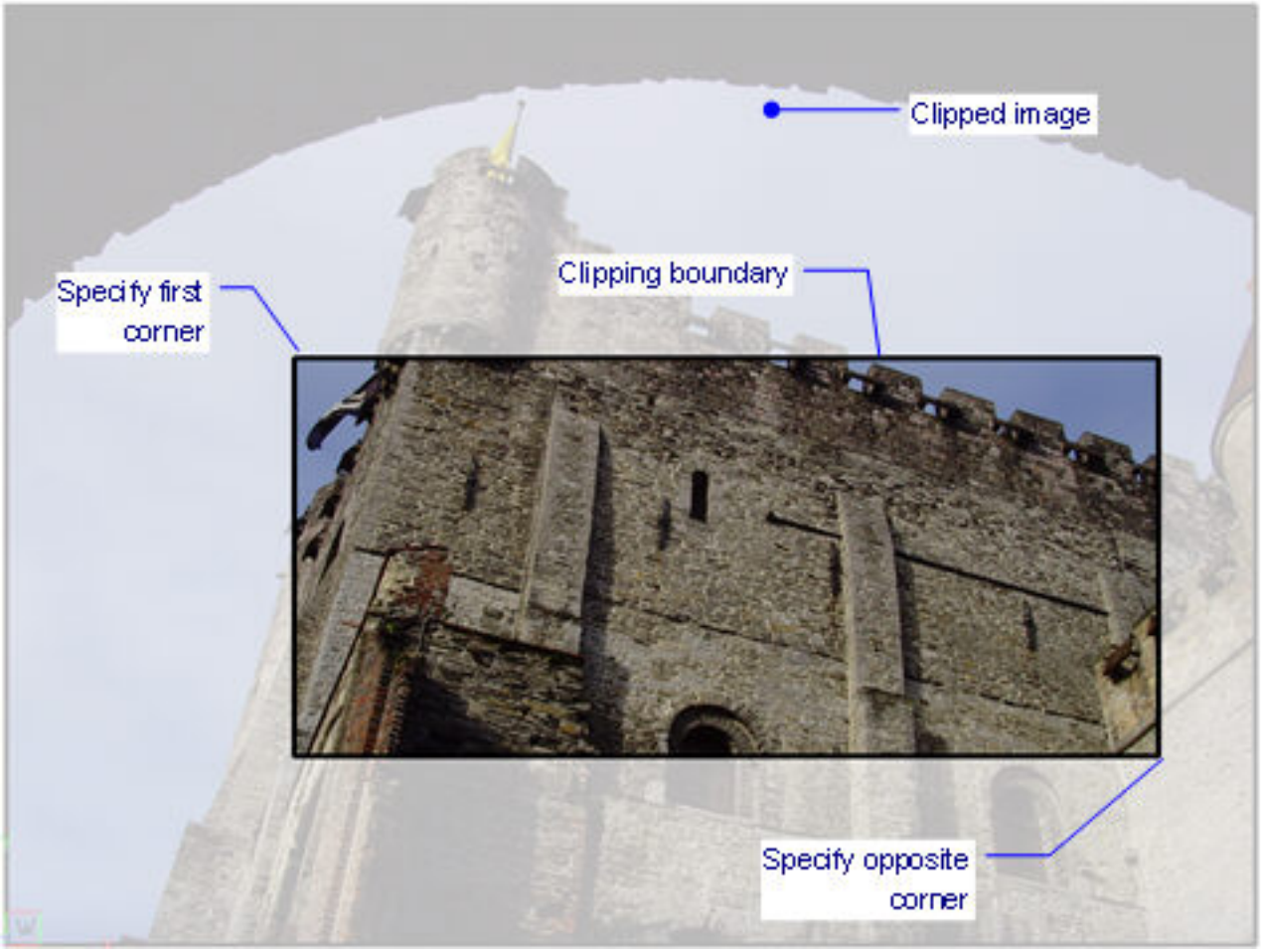
6.17.7 ImageClip

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Clips images with rectangular or polygonal shapes. **Note:**An image can have just one clipping boundary; creating a new one erases the old one.

Accessing the Command

 command bar: **imageclip** transparent: **'imageclip** alias: **icl** menu bar: **Modify | Raster Image | Clip Image** toolbar: **Images** |  : **ImageClip Prompts you in the command bar:** Select image: **(Select an image by clicking on its frame)** Enter image clipping option [ON/OFf/Invert/Delete boundary/New boundary] <New boundary>: **(Enter an option, or else press Enter to create a new clipping boundary)** Enter image clipping type [Polygonal/Rectangular] <Rectangular>: **(Press Enter to create a rectangular clipping boundary, or else choose Polygonal)** Specify first corner: **(Pick one corner of the clipping rectangle)** Specify opposite corner: **(Pick other corner of the clipping rectangle)** (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image166.jpg}) The clipped portion becomes invisible; the figure shows the clipped portion faded for illustrative purposes; in BricsCAD,

Figure 6.1099



Command Options

Option	Description
Enter clipping option	Determines the clipping action: • **ON** - turns on the clipping and displays the clipping boundary • **OFF** - turns off the clipping and hides the clipping boundary • **Invert** - inverts the clipping mode so that the image is clipped on the outside or the inside the clipping boundary • **Delete boundary** - removes the clipping boundary • **New boundary** - creates a new clipping boundary, and turns on clipping
Rectangular	Creates a rectangular clipping boundary; prompts you: 1. **Specify first corner** - specifies the first corner of the clipping rectangle 2. **Specify opposite corner** - specifies the other corner of the clipping rectangle
Polygonal	Creates a multi-sided, polygonal boundary; prompts you: 1. **Specify first point** - specifies the starting point of the clipping boundary 2. **Specify next point or [Undo]** - specifies the next point; as you pick points, the program previews the boundary 3. **Specify next point or [Undo]** - specifies the next point, or undoes the last one; this prompt repeats until you press **Enter** **Undo**- undraws the last polygonal segment (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/

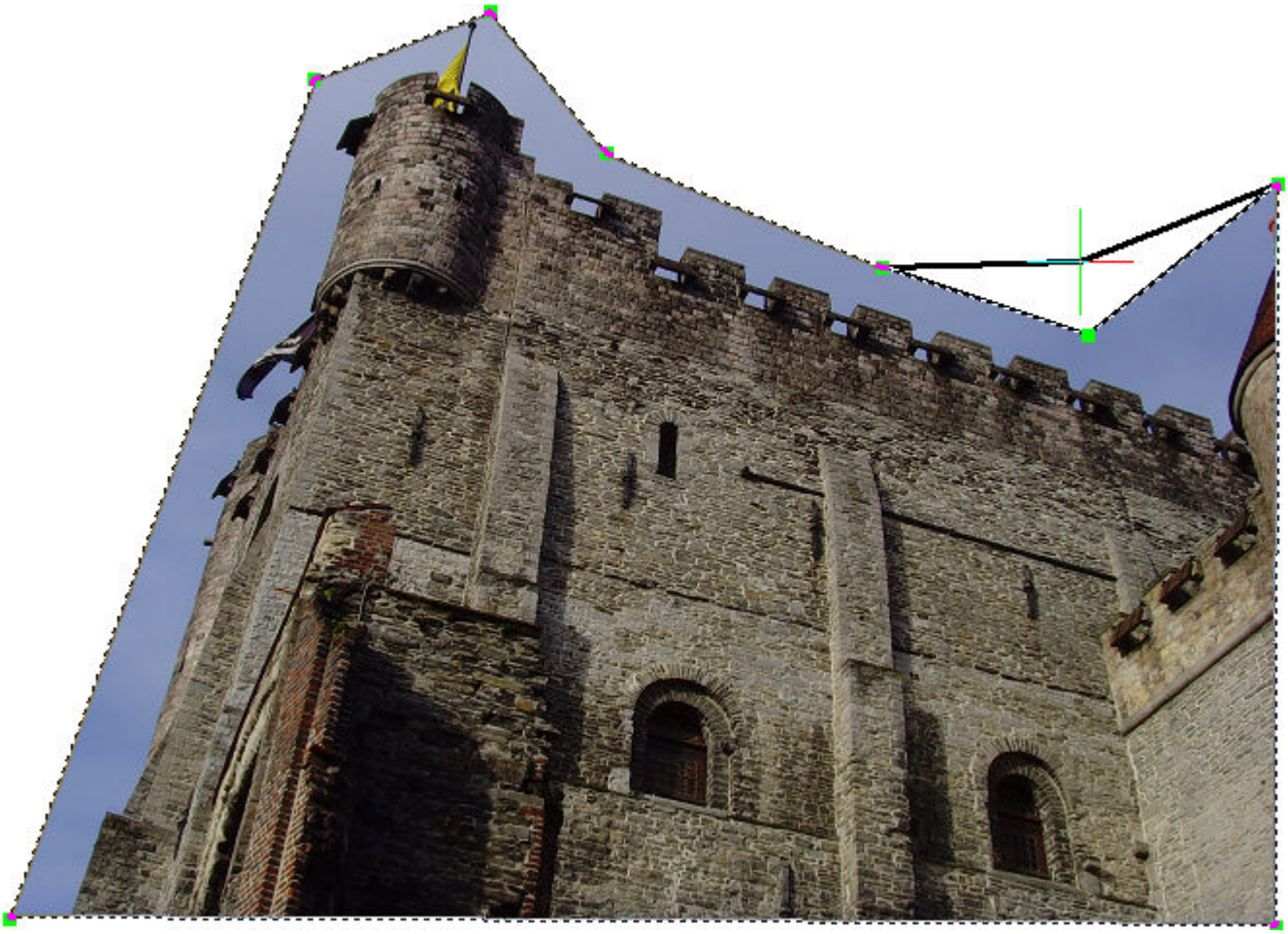
Grips Editing

Clipped images can be edited directly through grips:

1. Select the image frame. Notice that it has several grips.
2. Drag a grip to stretch the frame, which changes the path of the clipping boundary.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image169.jpg})

Figure 6.1100



Procedures

- To clip an image (1.24.2)
- To toggle the clipping boundary of an image (1.24.2)

Related Commands

Image (6.17.3) - controls the parameters of the image file. ImageAttach (6.17.5) - places raster images in drawings. -ImageAttach (6.17.6) - attaches raster images to the drawing; displays prompts in the command bar. ImageAdjust (6.17.4) - changes the properties of images through the Properties palette. ImageFrame (6.17.8) - toggles the display of the frame around images. ImageQuality (6.17.9) - toggles the quality of the image. Transparency (1.17.8) - toggles the transparency of monotone images.

Figure 6.1101



Figure 6.1102



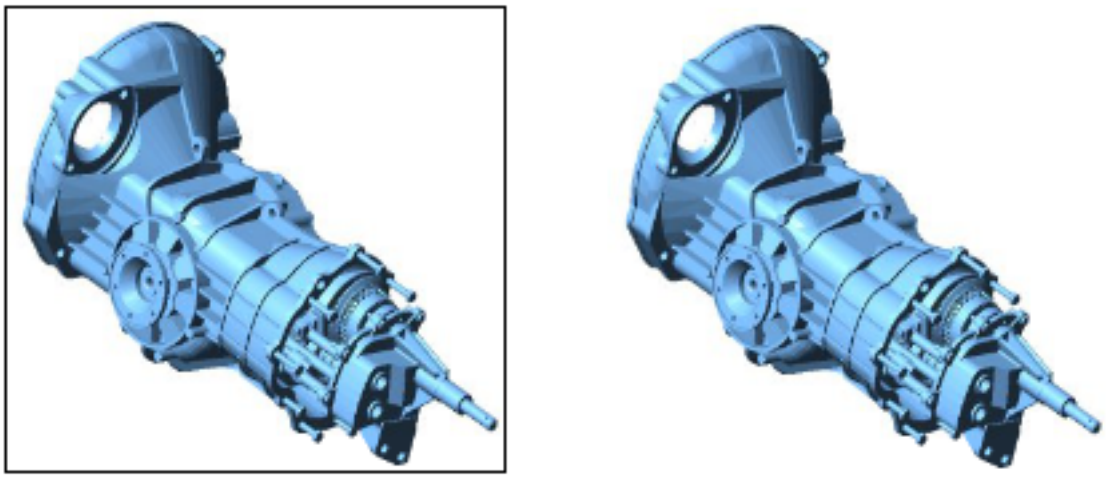
6.17.8 ImageFrame

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Toggles the display of the frame around all images in the drawing.

Accessing the Command

 command bar: **imageframe** transparent: **'imageframe** menu bar: **Modify | Raster Image | Display Image Frame** toolbar: **Images |**  : **ImageFrame Prompts you in the command bar:** Enter image frame setting <0>: **(Type 0 or 1)** Fame on (left) and off (right): (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef This command applies to all images in the drawing; frames cannot be toggled on individual images.It also applies to the frames made of rectangular and polygonal clippings. When frames are turned off, images can only be selected using window selection (1.23.6).

Figure 6.1103



Command Options

Option	Description
Enter image frame setting	Toggles the display of image frames: • **1** - turns on image frames. • **0** - turns off image frames.

Related Commands

Image (6.17.3) - controls the parameters of the image file. ImageAttach (6.17.5) - places raster images in drawings. -ImageAttach (6.17.6) - attaches raster images to the drawing; displays prompts in the command bar. ImageAdjust (6.17.4) - changes the properties of images through the Properties palette. ImageClip (6.17.7) - clips images. ImageQuality (6.17.9) - toggles the quality of the image. Transparency (1.17.8) - toggles the transparency of monotone images.

Figure 6.1104



6.17.9 ImageQuality

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Toggles the quality of images between draft and high.

Accessing the Command

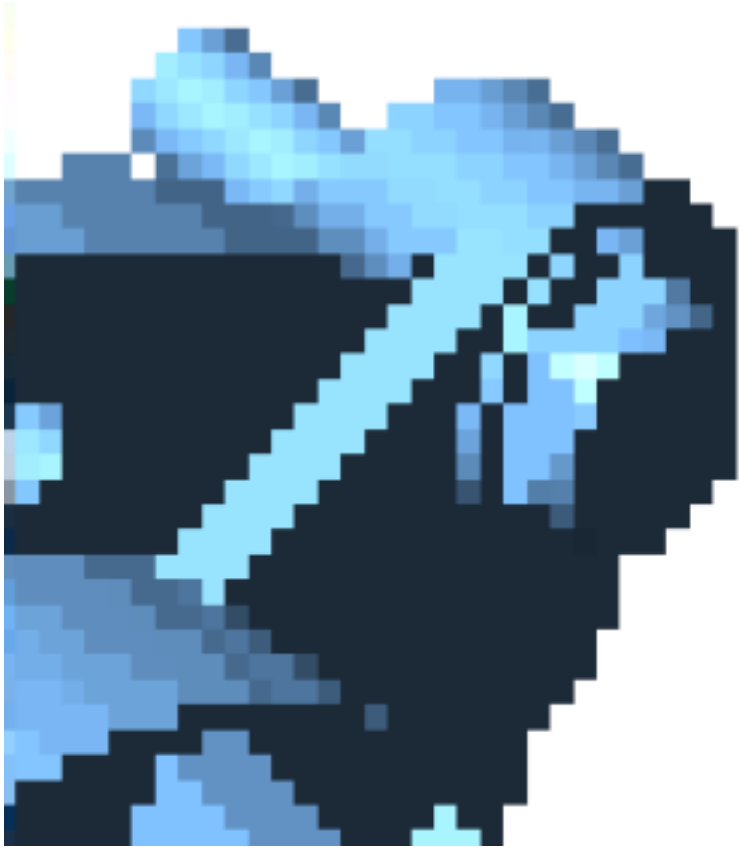
 command bar: **imagequality** transparent: **'imagequality** menu bar: **Modify | Raster Image | Image Quality** toolbar: **Images |**  : **ImageQuality Prompts you in the command bar:** Enter image quality setting [Draft/High] <High>: **(Type D or H)** Image quality high (left) and draft (right): (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/ImageQualityDraft.png})
Use this command to reduce the memory used by very large images.When set to Draft, this command:

- Reduces color resolution
- Scales images with large numbers of pixels to reduce resolution and memory footprint

Figure 6.1105



Figure 6.1106



Command Options

Option	Description
Enter image quality setting	Toggles the image display quality: • **Draft** - lowers quality, displays faster • **High** - raises quality, displays slower

Related Commands

Image (6.17.3) - controls the parameters of the image file. ImageAttach (6.17.5) - places raster images in drawings. -ImageAttach (6.17.6) - attaches raster images to the drawing; displays prompts in the command bar. ImageAdjust (6.17.4) - changes the properties of images through the Properties palette. ImageClip (6.17.7) - clips images. ImageFrame (6.17.8) - toggles the display of the frame around images. Transparency (1.17.8) - toggles the transparency of monotone images.

Figure 6.1107



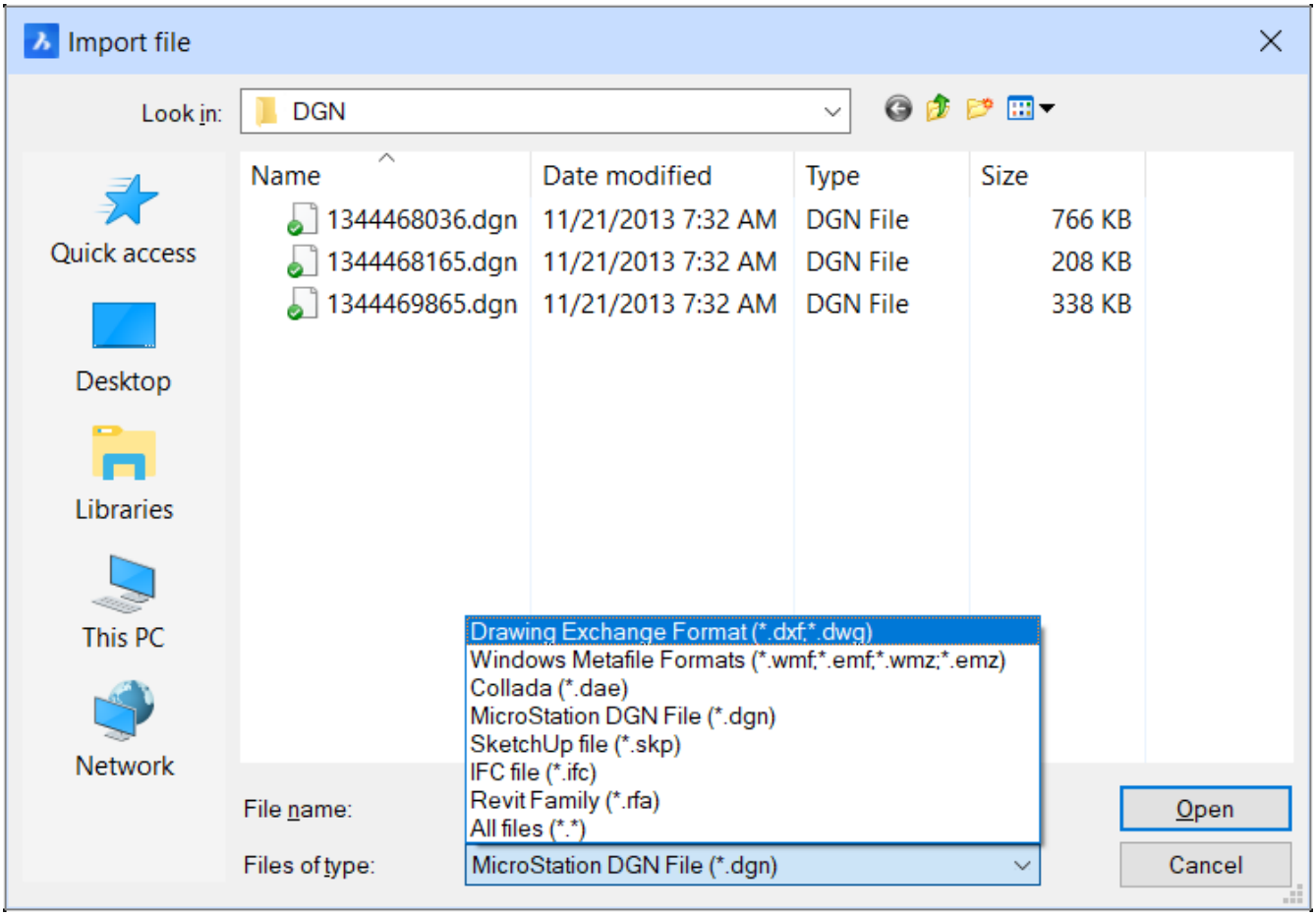
6.17.10 Import

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Imports the geometry from external files into the current drawing. **Note:** Some import formats have settings that can be changed through **Import** section of the Settings (6.27.23) command.

Accessing the Command


command bar: **import** alias: **imp** menu bar: **File | Import** ribbon:**Home | File | Import**: **import : Import** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/import-dlg.png}) Choose a file, and then click **Open**. The file is opened in the current drawing.

Figure 6.1108



Command Options

Option	Description
File name	Specifies the name of the file.
Files of type	Specifies the file type to show in the Import File dialog box: • **DAE** - Collada files • **DGN** - Microstation design files • **DWG** - drawing files • **DXF** - drawing interchange files • **EMF**, **WMF**, **EMZ**, **WMZ** - Windows metafiles Formats available with the BIM add-on: • **IFC** - Industry Foundation Classes (IFC) data model that describes building and construction industry data • **RFA** - Revit family files of symbols • **SKP** - Sketchup model files
Open	Inserts the geometry in the current drawing.

The following 3D formats are available through a separate extra-cost module, BricsCAD Communicator; download itfrom the Bricsys website.

Format	Extension	Description
Standard for Exchange of Product data	STE, STP, STEP	STEP (Standard for the Exchange of Product Data) is an ISO standard exchange format used for representing three-dimensional data in a format that can be recognized by mul-

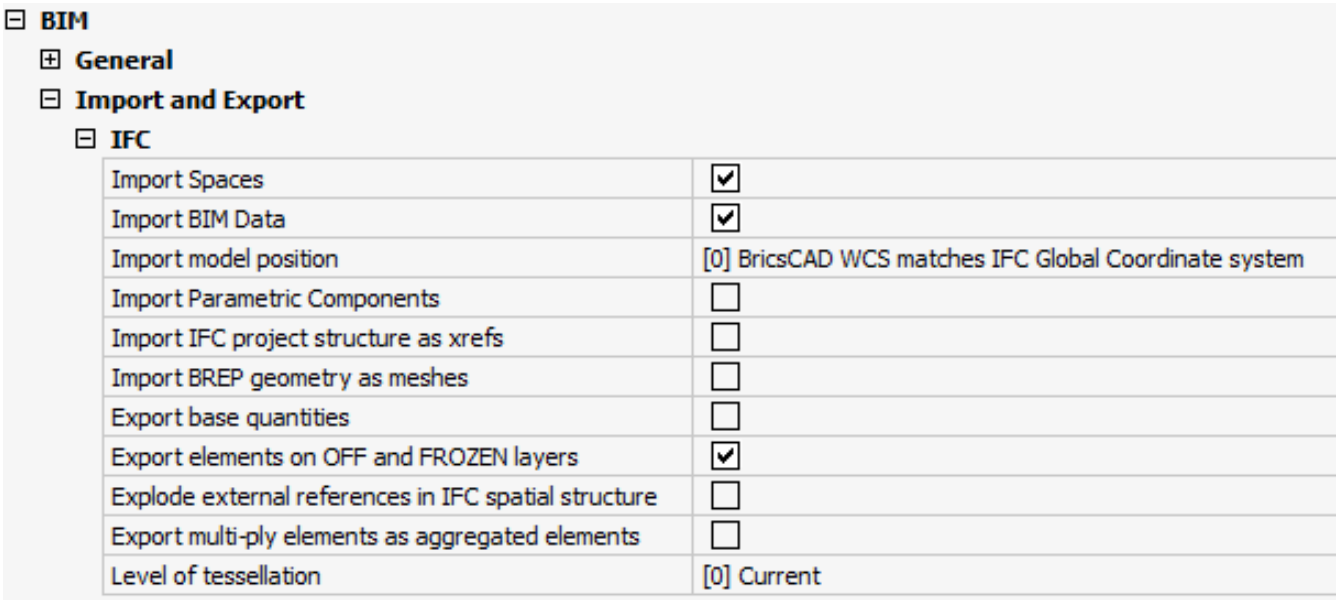
JT	JT	JT is a 3D data format developed by Siemens PLM Software (formerly UGS Corp.) and is used for product visualization, collaboration, and CAD data exchange. It can contain any combination of approximate (faceted) data, boundary representation surfaces (NURBS), Product and Manufacturing Information (PMI), and Metadata (textual attributes) either exported from the native CAD system or inserted by a product data management (PDM) system.
CATIA V4	model	CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systèmes.
CATIA V5	CATPart, CATProduct	V5 of the Catia CAD program from Dassault.
PARASOLID	XMT_TXT, XMT_BIN X_T, X_B	Parasolid is a geometric modeling kernel originally developed by ShapeData, now owned by Siemens PLM Software (formerly UGS Corp.), that can be licensed by other companies for use in their 3D computer graphics software products.
Solid Edge	PAR, PSM	Solid Edge is a 3D CAD parametric feature solid modeling software. which provides solid modeling, assembly modelling and drafting functionality.
NX	PRT	PRT is a part or model file format used by PTC Pro/Engineer, which is a CAD program that creates design parts and assembly processes.
Creo Elements Pro Engineer	ASM, PRT	Parametric, integrated 3D CAD/CAM/CAE solution created by Parametric Technology Corporation (PTC).
Solidworks	SLDASM, SLDPRT	SolidWorks is a 3D mechanical CAD program that runs on Microsoft Windows and is being developed by Dassault Systèmes. SLDPRT: part file, SLDASM: assembly file The ImportSwRotateYZ (1.13.11) user preference controls if geometry imported from SolidWorks files should be automatically rotated to map Y axis to Z axis.
VDA-FS	VDA	VDA-FS is a CAD data exchange format for the transfer of surface models from one CAD system to another. Its name is an abbreviation of ”Verband der Automobilindustrie - Flächenschnittstelle”, which translates to the ”automotive industry association - surface data interface”. Standard was specified by the German organization VDA.

Physical materials (2.6.10) If the imported assembly has physical materials assigned to its parts, the materials can be imported together with the assembly structure. For each material in the imported file, a corresponding material will be created in the target document material library, and its name, density, specific heat and thermal conductivity will be copied from the source material. If the imported file contains several materials with the same name, the first one will be used. If the document already has a material with the same name in its material library, this material will be used instead. Materials copied to the document material library will not be erased in Undo operations.

Settings

The import and export procedures using **Communicator** are controlled through a series of user preferences. See the **Communicator** section on the Settings (6.27.23) dialog. The **IFC** import and export is controlled through a series of settings in the **BIM/Import and Export** section on the Settings (6.27.23) dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/BIM/ImportExportIFCSettings.png})

Figure 6.1109



Related Commands

CommunicatorInfo (6.9.39) - checks whether the Communicator product is correctly installed and prints a diagnostic report in the Prompt History window. dmSimplify (6.12.27) - simplifies the geometry and topology of 3d solid entities. DmStitch (6.12.28)- converts a set of region and surface entities that bound a watertight area to a 3d solid. DmAudit (6.12.2)- fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces). Export (6.13.15) - exports the current drawing in a variety of raster and vector formats. IfcEcport (6.17.2) - exports a BIM model to IFC, partially or entirely. Insert (6.17.13) - inserts DWG, DXF, and DWT files. InsertObj (6.17.17) - inserts objects from other applications. Load (6.20.36) - loads SHX shape files. VSlide (6.31.22) - displays SLD, WMF, and EMF files.

Figure 6.1110



6.17.11 -Import

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Imports geometry from external files at the command prompts. **Note:** This command opens all file formats supported by the Import (6.17.10) command.

Accessing the Command

command bar: **-import** : **-Import Prompts you at the command line:** Enter import file name: **(Enter the path and file name of an importable file, such asC:\dwg\DXF\bc.dxf)** The geometry in the file is added to the current drawing.

Command Option

Option	Description
Enter import file name	Specifies the full path path and file name of an importable file: • Enter a path and name, such as**C:\dwg\DXF\bc.dxf** • Press **~** (tilde) to display theImport File dialog box; see the Insert (6.17.10) command • Press **Enter** to exit the command; the program reports, "No file name entered"

Related Command

Import (6.17.10) - imports geometry from external files into the current drawing via a dialog box.

Figure 6.1111



6.17.12 Imprint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Imprints 2D entities onto planar faces of 3D solids and surfaces to create additional edges. **Note:** Use dynamic UCS (DUCS) to draw on the faces of 3D objects.

Accessing the Command

 command bar: **imprint** menu bar: **Model | 3D Solid Editing | Imprint** toolbar: **3D Solid Editing**|  : **Imprint Prompts you in the command bar:** Select 3d solid, surface or region: **(Select a 3D solid, a surface, or a region entity)** Select an entity to imprint: **(Select a 2D entity lying on or intersecting a planar face of the selected entity; the figure shows a circle, line, and rectangle polyline on the faces of a 3D solid)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/imprint-11-eg.png}) Delete the source object Yes/<No>: **(Type Y or N)** Select an entity to imprint: **(Select another 2D entity to imprint on the same sold/surface/region, or else press Enter to end the command; the figure shows the imprints on the 3D solid, with source entities deleted)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/imprint-2-eg.png}) With the entities imprinted on the solid, you can use commands like Extrude (6.13.21) or dmPushPull (6.12.20) to manipulate the newly created faces, as shown below. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands 3-eg.png})

Figure 6.1112

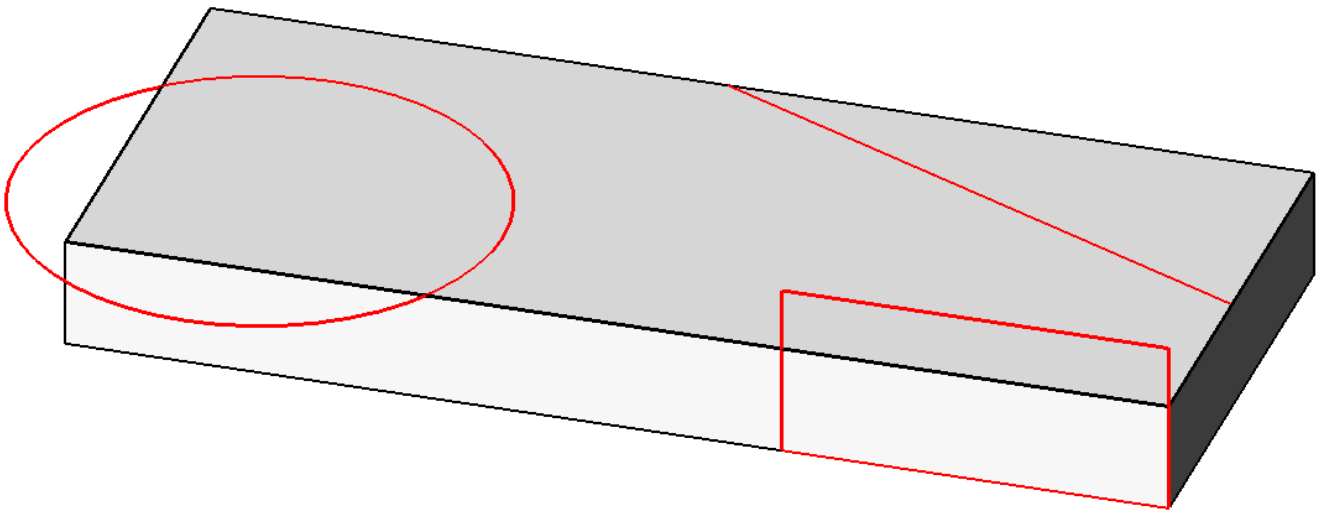


Figure 6.1113

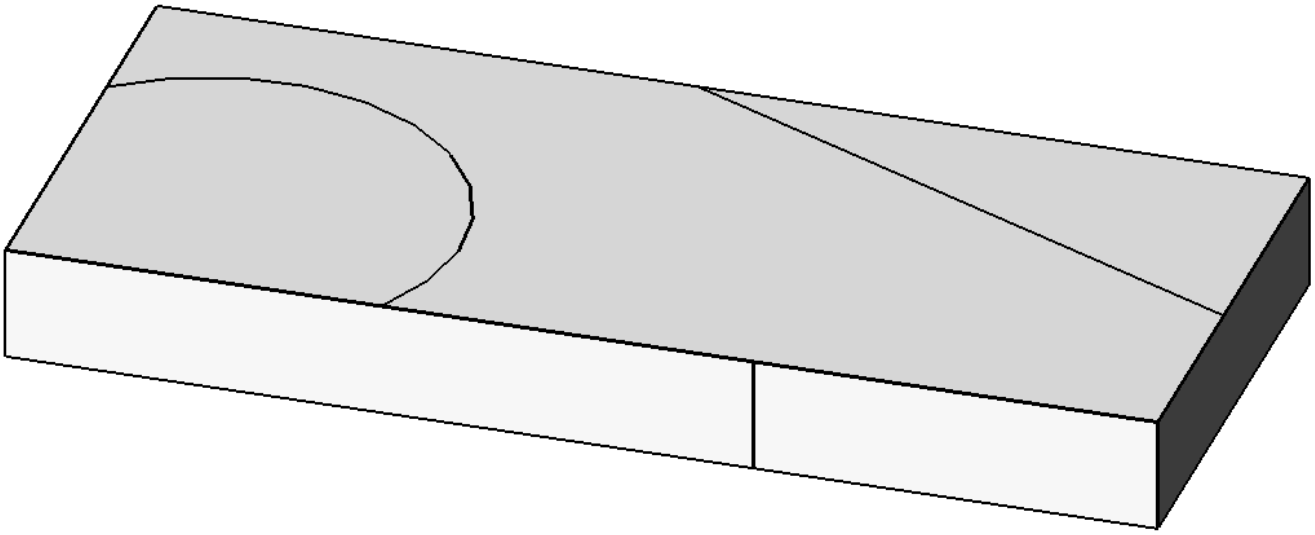
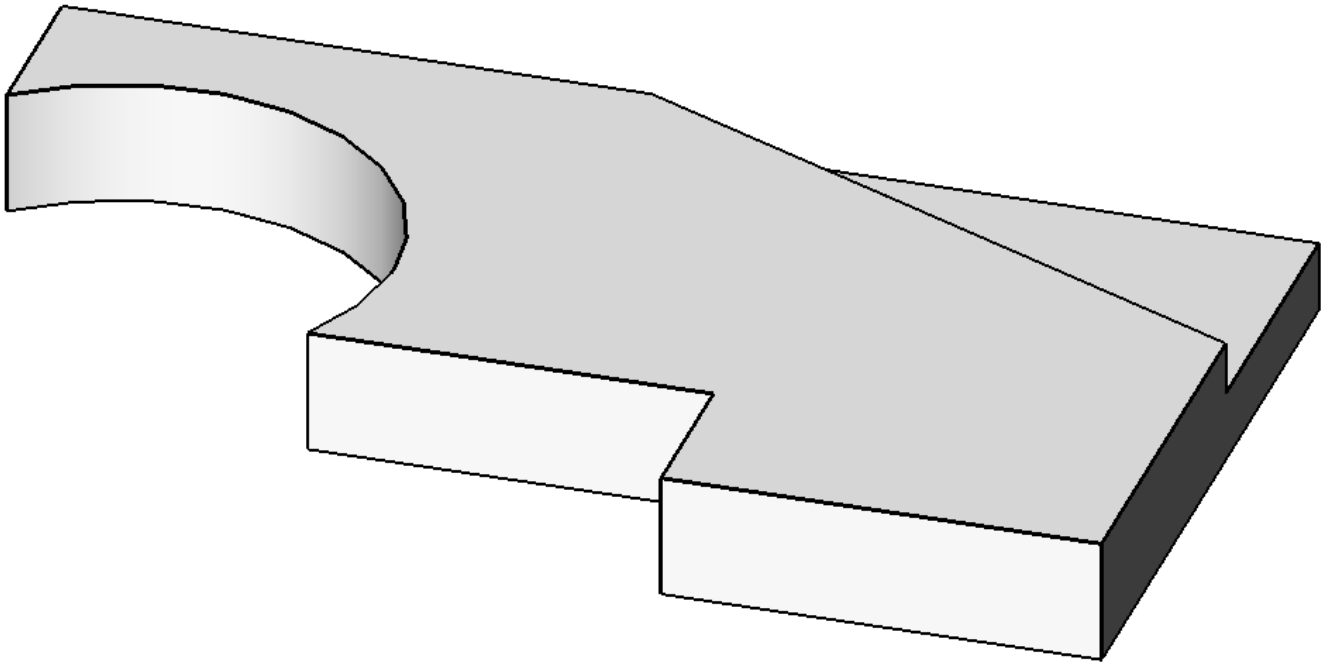


Figure 6.1114



Command Options

Option	Description
Select 3d solid, surface or region	Select one of the following entities: • 3D planar solid, which is a solid that has at least one flat face, so no spheres or tori • Surface entity • Region entity; see the Region command
Select an entity to imprint	Select any 2D entity laying on or intersecting the planar face; this is the ”source object”
Delete the source object	Determines the fate of the source entities used to make the imprints: • **Yes** - deletes the source entity after the imprint is completed • **No** - keeps the source entity around The**Select an entity to imprint** prompt repeats so that you can imprint more source entities on the solid/surface/region.

Related Commands

ProjectGeometry (6.24.48) - projects 2D geometry onto a region, a surface or a 3D solid; allows to create additional linework on the faces of solids and surfaces. SolidEdit (6.27.39) - edits 3D solids and 2D regions. XEdges (6.34.3) - creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, surfaces, etc. and regions.

Figure 6.1115



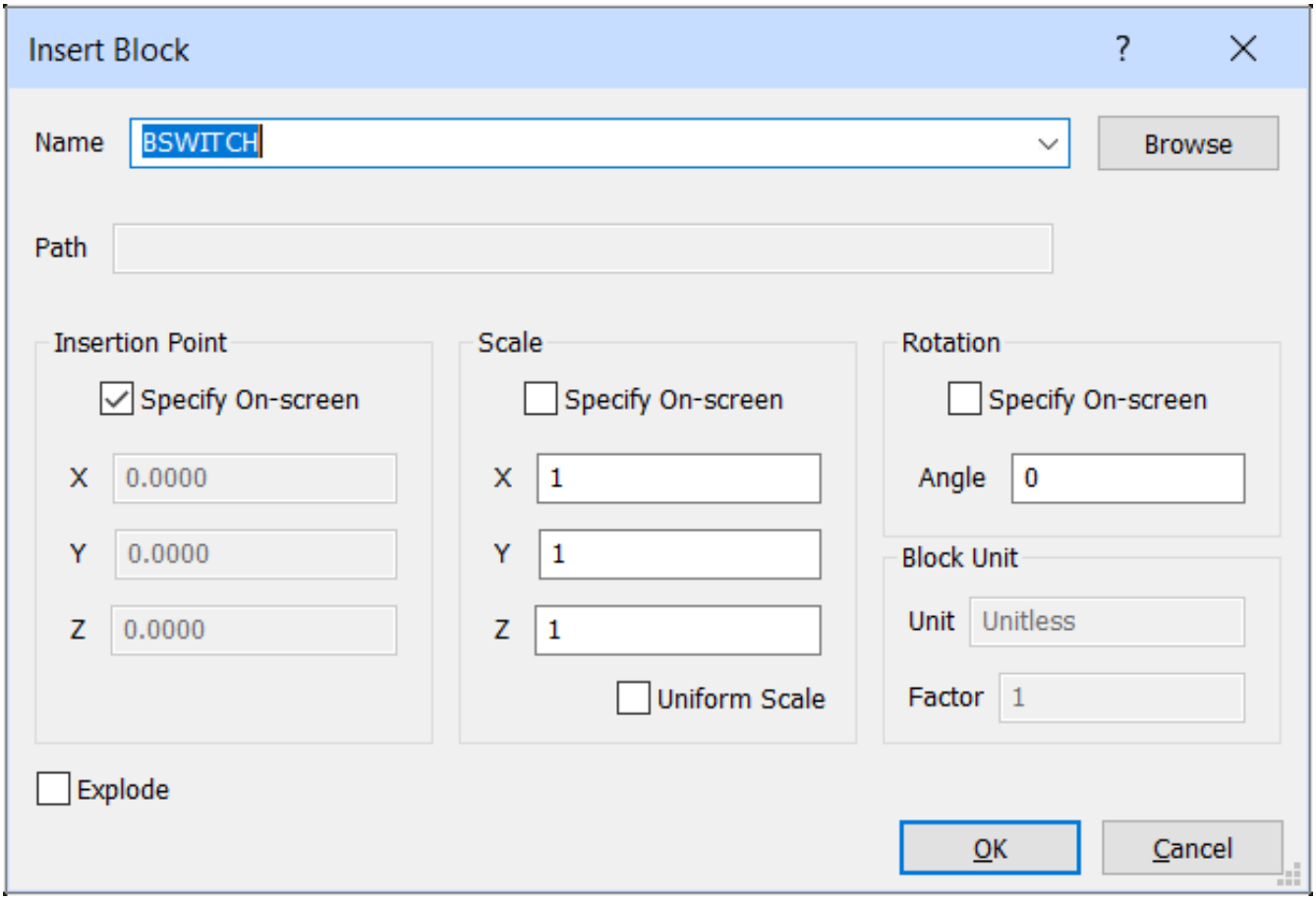
6.17.13 Insert

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Inserts blocks into the current drawing through a dialog box, and then prompts for attribute values, if any; also inserts entire DWG and DXF files as blocks into the drawing.

Accessing the Command


command bar: **insert** aliases: **i, ddinsert** menu bar: **Insert | Insert Block** ribbon: **Insert | Blocks** toolbar: **Insert** | Quad:**Insert** : **Insert** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Cor dlg.png}) Enter options, and then click **OK**. If the block contains attributes, then you are prompted to enter values for the attributes in the command bar or through a dialog box -- depending on the value of the the value of the ATTDIA (attribute dialog) variable.

Figure 6.1116



Command Options

Option	Description
Name	Specifies the name of the block, whose definitionexists in the drawing, or is a DWG or DFX file on the computer or network: To choose a block in the drawing: • Enter the name • Click the **Name** droplist to choose from the names of blocks defined in the drawing Blocks are created with the Block (6.6.11) command.
Browse	Selects a DWG or DXF file from your computer or network: 1. Click the **Browse** button 2. Select a DWG or DXF file from the Select Drawing File dialog box 3. Click **Open**
Path	Reports the path to the block, if the block was opened from a DWG or DXF file.

Insertion Point	Specifies the insertion point of the block in the drawing. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ Insertion points are typically chosen at the lower-left corner of the block, in the center of the block, or at entity in the block that represents a connection point.
Specify On-screen	Toggles where you specify the insertion coordinates: • **Off** - you specify the x,y,z coordinates in the dialog box • **On** - you specify the coordinates in the command bar after you click **OK**when the program prompts you: **Insertion point** - **(specify the x,y or x,y,z coordinates of the block's lower left corner)**
X Y Z	Specify the X-, Y- and Z- coordinates of the block's insertion point. The Z coordinate is typically left at 0 for 2D blocks.
Scale	Scales the block: • Values larger than 1 - make the block larger • 1 - inserts the block at actual size • Values smaller than 1 - make the block smaller • Values less than zero - flip the block, like mirroring it (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Specify On-screen	Toggles where you specify the scale factors: • **Off** - specifies the x,y,z scales in the dialog box. • **On** - Specifies the scale factor in thecommand bar after you click **OK**when the program prompts you: **Scale factor** - **(specify the scale factor.)**
X Y Z	Defines the scaling of the block: • **X**- specifies the scale factor along the x axis; enter a negative value to mirror the block about the y axis • **Y**- specifies the scale factor along the y axis; enter a negative value to mirror the block about the x axis • **Z**- specifies the scale factor along the z axis
Uniform Scale	Toggles whether or not the same scale factor is used for all axes: • **On**- use the x scale factor also for the y and z scale factors • **Off**- allows the x, y, and z scale factors to be different from one other

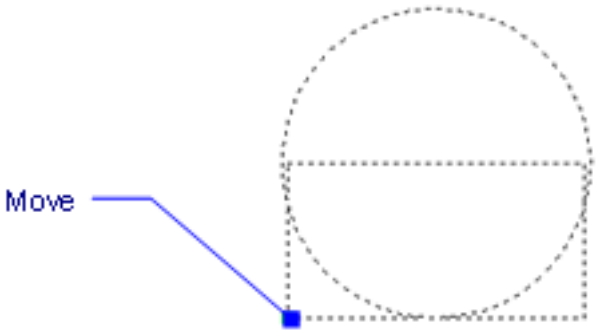
Rotation Angle	Specifies the rotation angle of the block about its insertion point, starting with the x-axis as 0 degrees: • **Positive angles** - rotate the block counter-clockwise • **Negative angles** - rotate the block clockwise (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Specify On-screen	Toggles where you specify the rotation angle: • **Off** - specifies the angle in the dialog box • **On** - specifies the anglein thecommand bar after you click **OK**when the program prompts you: **Rotation angle** - **(specify the angle.)**
Explode	Toggles whether the block is inserted exploded: • **Yes** - insert the block and then explode it to its constituent parts; see the Explode command • **No** - does not explode block; it is inserted as a block
Block Unit	Controls the automatic scaling of the block with respect to the INSUNITS system variable of the current drawing.
Unit	Shows the INSUNITS setting of the inserted block or drawing.
Factor	Shows the calculated scaling factor with respect to the INSUNITS setting of the inserted block (or drawing) and the current drawing.

Attribute Options	**Description**
ATTDIA = on	When the block contains attributes, the program prompts you to enter values for each attribute. Whether it displays a dialog box or prompts you at the command line depends on the AttDia variable. When AttDia=on, displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/ins attdia-dlg.png}) 1. Select an attribute in the list 2. Specify the value in the **Value** field; when the value is mtext,click the **...** button to access the mtext editor 3. Repeat for each attribute 4. Click **OK** To accept the default value for all attributes, simply click **OK**.
ATTDIA = OFF	When AttDia=off (default), the program prompts you in the command bar: **Attribute Prompt <default value>** - **(Type a new value at the command prompt, or else press Enter to accept the default value.)**

Grips Editing

Blocks can be edited directly through grips:

Figure 6.1117



1. Select the block. Notice that it has one grip located at the insertion point.
2. Drag the grip to move the block.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image760.png})

Procedures

- General procedure to insert a block (1.22.6)
- To insert a block (1.22.6)
- To insert a block using the Drawing Explorer (1.22.6)
- To insert a block aligned with an entity (1.22.6)
- Placing blocks with attributes (1.22.14)

Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Field (6.14.3) - inserts a field in a multiline text, table cell or attribute. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. InsertAligned (6.17.15) - inserts blocks repeatedly, and inserts mirrored blocks. MInsert (6.21.17) - inserts a block as a rectangular array PasteBlock (6.24.9) - pastes entities from the Clipboard as blocks. Pdfattach (6.24.14) - inserts a PDF file as an underlay into the current drawing; displays dialog boxes. -PdfAttach (6.24.15) - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar. PdfImport (6.24.17) - imports a pdf andconverts the content to dwg entities. Properties (6.24.49) - edits attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place. ResetBlock (6.26.27) - resets dynamic blocks to default values. WBlock (6.33.1) - writes blocks and other drawing parts to files on disk. XAttach (6.34.1) - attaches externally-referenced drawings through a dialog box.

Figure 6.1118

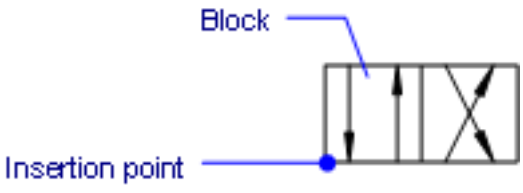


Figure 6.1119

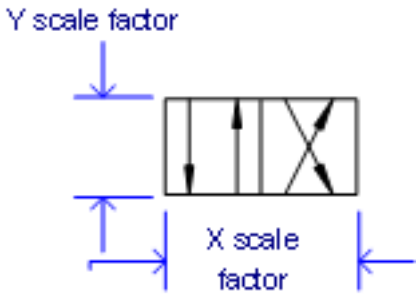


Figure 6.1120

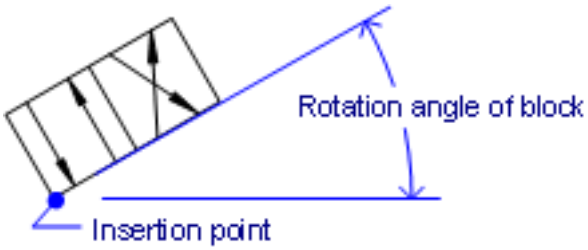


Figure 6.1121

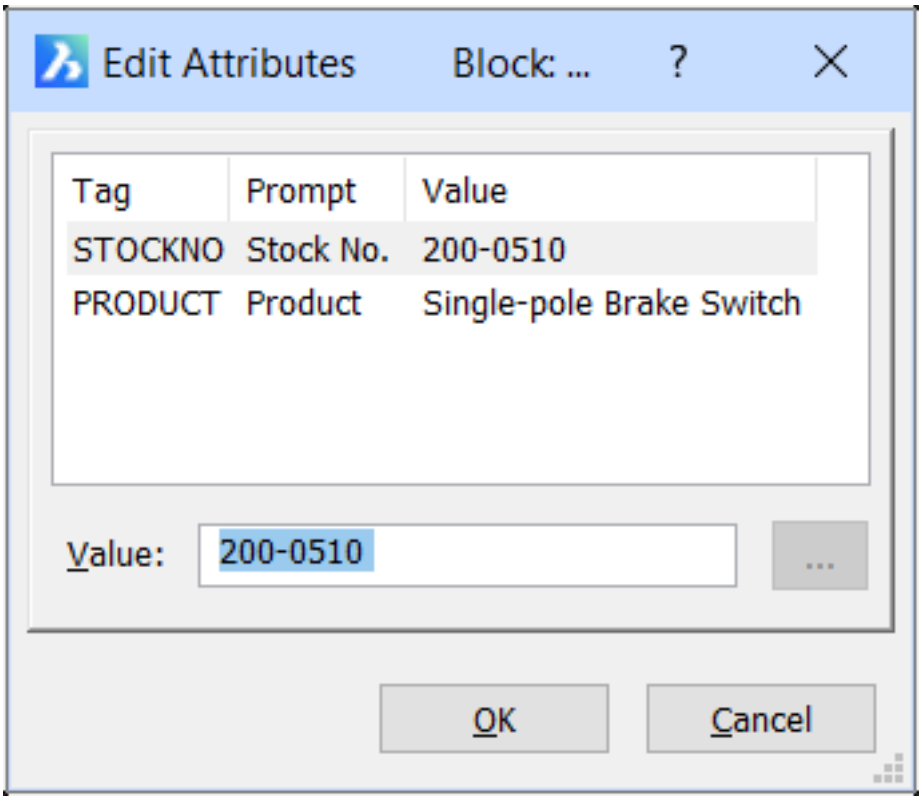


Figure 6.1122



6.17.14 -Insert

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Inserts blocks through prompts in the command bar; also prompts for attribute values, if any are part of the block definition.

Accessing the Command

command bar: **-insert** alias: **-i** : **-Insert Prompts you in the command bar:** ? to list blocks in drawing/~ to open the file dialog/<Block to insert> <BlockName>: **(Type the name of the block to insert, press Enter to accept the name of the previously inserted block, or else choose an option)** Insertion point for block or [Multiple blocks/Scale]: **(Specify the insertion point, or else choose an option)** [Corner] <1.000000>: **(Specify a point to show the size of the block, type a value to specify the X scale factor, press Enter to accept the default, or else enter C)** Y scale factor: < Equal to X scale (1.000000)>: **(Press Enter to set the Y scale factor equal to the X scale factor, or else type a value to specify the Y scale factor)** Rotation angle for block <0>: **(Press Enter to accept the default rotation angle, pick a point to show the rotation, or else or type a value)**

Command Options

Option	Description
? to list blocks in drawing	Lists the names of all blocks in the current drawing. Prompts you: **Block(s) to list <* for all>** - **(enter an option)**: • Press **Enter** to list all blocks • Enter part of a name to view the names of certain blocks. You can use wildcards like ? (for an individual character) and * for all characters. Generates a report similar to the following one: **Defined blocks. "_DYN_DIM_3D" "_None" User Unnamed Blocks Blocks 2 0** If necessary, press **F2** to view the full list; see TextScr (6.29.13) command. You must restart the -Insert command to insert the block.
~ to open the file dialog	Displays the Insert Block dialog box.Choose a DWG or DXF file, and then click **Open** .

BlockName=DrawingName

Redefines an existing block, replacing the block by a specified drawing. At the prompt: **? to list blocks in drawing/~ to open the file dialog/<Block to insert> <BlockName>:(Specify the block to be replaced and the replacement drawing.)**
BlockName=DrawingName * **The block BlockName already exists. Do you want to redefine it? [Yes-redefine block/No-do not redefine] <No-do not redefine>: (Type Y to redefine the block.)**

- Unless a path is specified,BricsCAD searches for the drawing in the folder of the current drawing and the paths defined by the SRCHPATH (1.13.11) system variable. When the replacement drawing is not found BricsCAD responds: Could not find file DrawingName
- Else, BricsCAD responds: The block ”BlockName” redefined: Units: Millimeters Conversion: 1 Insertion point for block or [Multiple blocks/Scale]: X scale factor or [Corner/scale X Y Z] <1.000000>: Y scale factor: <Equal to X scale (1.000000)>: Rotation angle for block <0>:

* If the EXPERT (1.13.11) system variable is 2 or higher, the final prompt is suppressed and the block is redefined without confirmation.

Block to insert

Specifies the name of the block to be inserted; prompts you: **Block(s) to list [* for all] -**

- The program first looks the block in the block definitions of the current drawing
- If the block is not found in the current drawing, then the program searches the paths defined by the SRCHPATH (1.13.11) system variable
- When the block is not found there either, the command responds ’Could not find file <blockname>’

Multiple blocks

Inserts the same block multiple times by repeating the following prompts until you press **Enter** or **Esc**.

Scale

Sets the overall scale factor for the block. Prompts you: **Scale factor for block:- (enter a value)**:

- Values larger than 1 - make the block larger
- 1 - inserts the block at actual size
- Values smaller than 1 - make the block smaller
- Values less than zero - flip the block, like mirroring it

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/

Insertion point for block

Specifies the insertion point of the block in the drawing. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Insertion points are typically chosen at the lower-left corner of the block, in the center of the block, or at entity in the block that represents a connection point.

Corner	Indicates the size of the block by specifying a second corner of a rectangle. The insertion point is the first corner. Prompts you: **Select other corner for scale or enter X scale factor** - **(pick a second corner, or enter a number for the x scale factor.You can pick points or enter x,y coordinates.)** (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Y scale factor	Specifies the scale factor of the block in the y direction. Enter a negative number to mirror the block.Press Enter to make the y scale = x scale.
Rotation angle for block	Specifies the rotation angle of the block about its insertion point. Enter an angle or pick a point to show the rotation .(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
ATTDIA = ON	If the block contains attributes, the value of the ATTDIA (Attribute Dialog) system variable determines whether you are prompted in the command bar to set the attributes or through a dialog box. A dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/ins attdia-dlg.png}) Select an attribute in the list, then define the attribute in the **Value** field.
ATTDIA = OFF	You are prompted in the command bar: **Attribute Prompt <Default Text>**. **(Type a new value at the command prompt or press Enter to accept the default value.)**

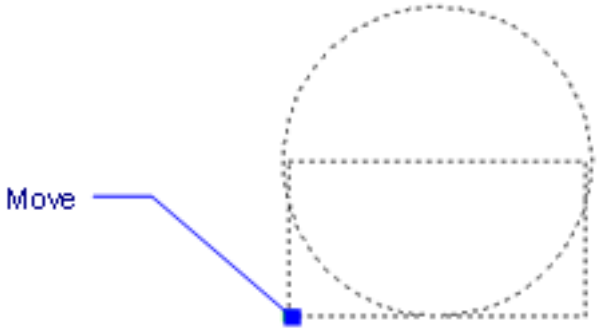
Grips Editing

Blocks can be edited directly through grips:

1. Select the block. Notice that it has one grip located at the insertion point.
2. Drag the grip to move the block.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image760.png})

Figure 6.1123



Related Commands

InsertAligned (6.17.15) - inserts blocks repeatedly, and inserts mirrored blocks. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. PasteBlock (6.24.9) - pastes entities from the Clipboard as blocks. AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces blocks to constituent entities and attributes to tags. Pdfattach (6.24.14) - inserts a PDF file as an underlay into the current drawing; displays dialog boxes. -PdfAttach (6.24.15) - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar. Properties (6.24.49) - edits block and attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place. ResetBlock (6.26.27) - resets dynamic blocks to default values.

Figure 6.1124

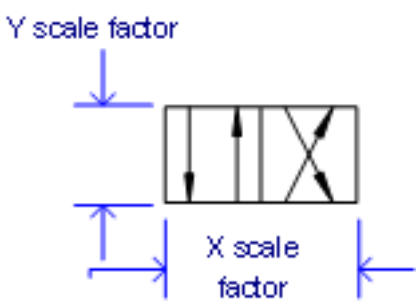


Figure 6.1125

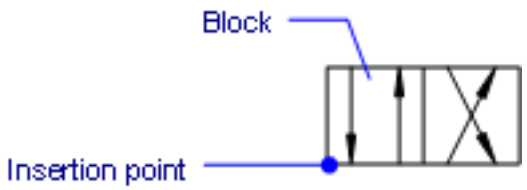


Figure 6.1126

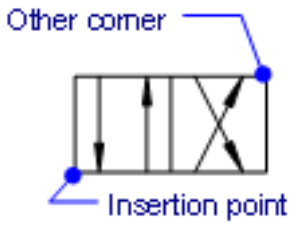


Figure 6.1127

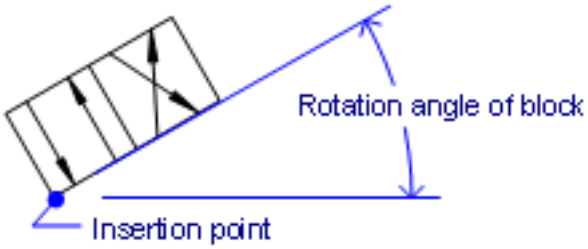


Figure 6.1128

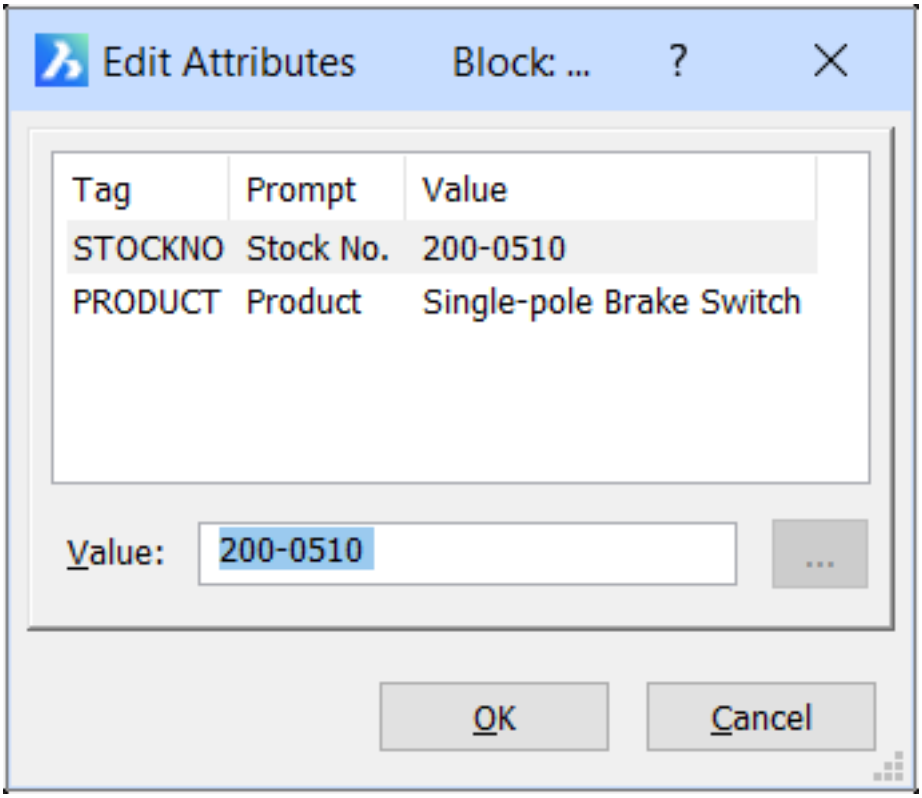
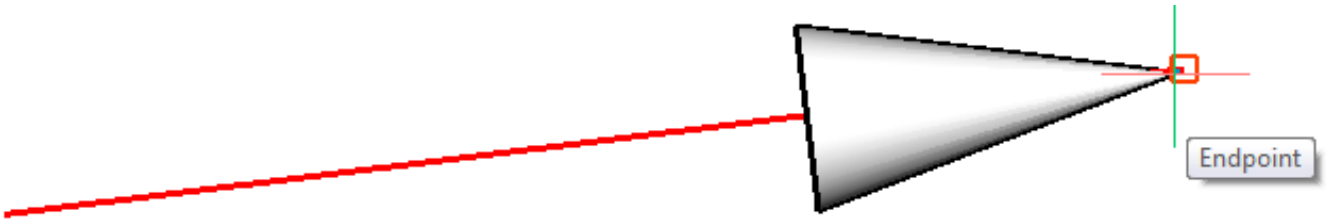


Figure 6.1129



Figure 6.1130



6.17.15 InsertAligned

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Inserts blocks aligned with entities, and optionally mirrors them. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/eneg.png}) **Note:** This command must be used with an entity snap turned on.

Accessing the Command

command bar: **insertaligned** alias: **insal** menu: **Insert | Insert Aligned Block** : **InsertAligned Prompts you in the command bar:** ? to list blocks in drawing/~ to open the file dialog/<Block to insert> <defaultname >: **(Type ? for a list of block names, enter ~ for a file dialog box, or else enter the name of a block)** Insertion point for block or [Multiple blocks/Scale]: **(With an entity snap turned on move the cursor over an entity in the drawing, , or else type Mor S)** Enter block mirroring control point: **(Move the cursor to flip the block, and then pick a point to place the block)**

Command Options

Option	Description
? to list blocks in drawing	Lists the names of all blocks in the current drawing. Prompts you: **Block(s) to list <* for all>** - **(press Enter to list all blocks, or enter part of a name to view some blocks.If necessary, press F2 to view the full list. See TextScr command.)**
~ to open the file dialog	Displays the Insert Block (6.17.13) dialog box.Choose a DWG or DXF file, and then click **Open**.
Block to insert	Specifies the name of the block to be inserted: • Press **Enter** to accept the name of the previously inserted block • Type the name of the block to insert
Multiple blocks	Repeats the Insertion point for next block: prompt until you press **Enter** or **Esc**.This allows you to insert the same block many times quickly.
Scale	Scale the block; prompts you: **Scale factor for block** - **(type a value.)**

Insertion point for block	Specifies the insertion point of the block in the drawing; follow these steps: 1. Turn on an entity snap mode, such as **end** or **nea** 2. Move the cursor over an entity, such as a line or circle 3. Click to insert the block 4. At the 'Enter block mirroring control point' prompt, move the cursor to flip the direction of the block 5. Click to locate the block
Enter block mirroring control point	Flips the block by 180 degrees: 1. Move the cursor to flip the block back and forth; the block is mirrored about its insertion point 2. Click to place the block (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/insc mirror-eg.png})

Related Commands

AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces attributes to tags. Insert (6.17.13) - inserts blocks into drawings. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. Properties (6.24.49) - edits attribute values and properties. RefEdit (6.26.12) - edits blocks and attributes in-place.

Figure 6.1131



Figure 6.1132



Figure 6.1133



6.17.16 Insertion

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the insertion entity snap, which snaps to the insertion points of text, mtext, tables, attribute text, blocks, xrefs, and attachments, like images and PDF files. (\vref {fig:C:/Dev/Task-2448/images/wwwbrics

Accessing the Command

 command bar: **insertion** transparent: **insertion** alias: **ins** toolbar: **Entity Snaps** |  : **Insertion** Displays no prompts in the command bar. When insertion snap is on, the cursor displays the insertion icon when it comes near to likely candidates.

Command Options

Option	Description
Insertion	Entering this command toggles Insertion entity snap on and off. It snaps to the insertion points of • Text, mtext, tables, and attribute text • Blocks, xrefs, and attachments, like images and PDF files This command can be entered transparently during commands, but must be entered without the apostrophe prefix ('), such as **int**.

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.1134



6.17.17 InsertObj

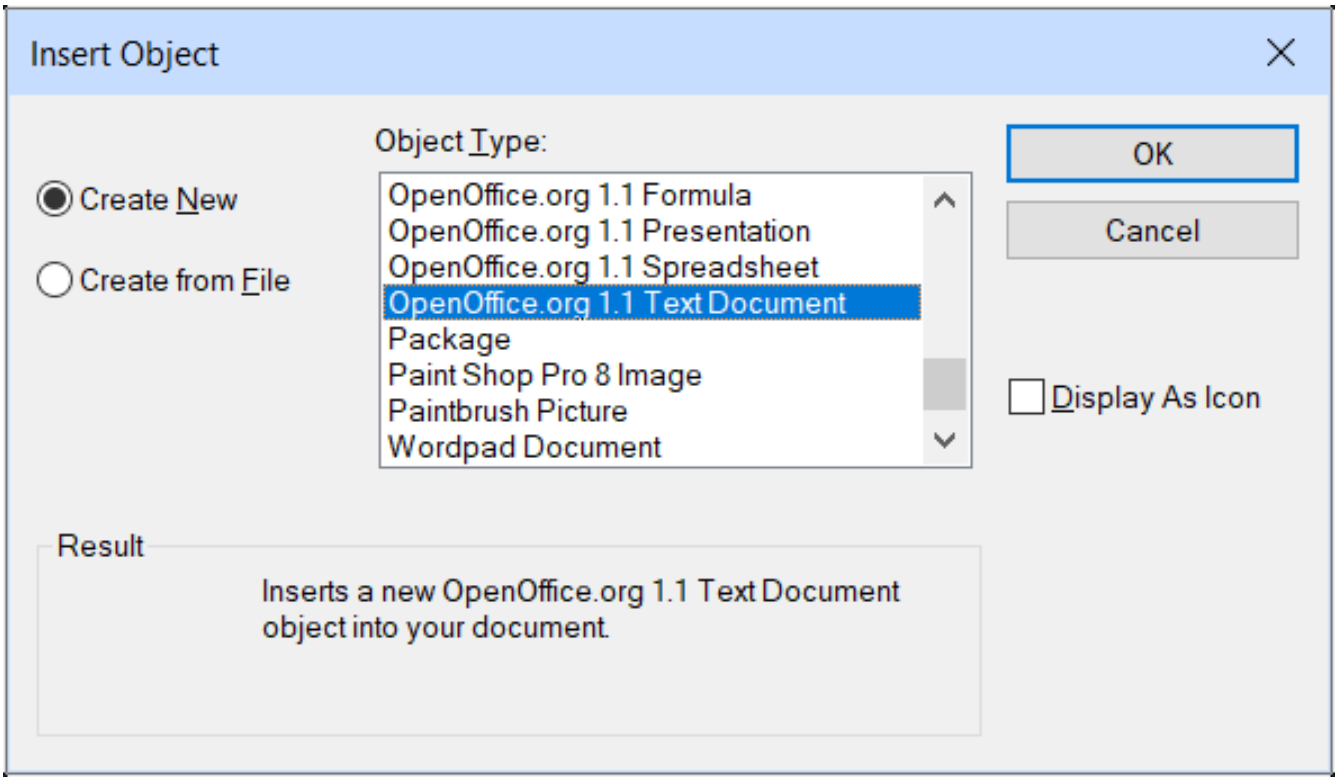
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Inserts non-BricsCAD entities objects in the current drawing.

Accessing the Command

command bar: **insertobj** alias: **io** menu bar: **Insert | Insert Object** : **InsertObj** Displays a dialog box; the objects types listed varies according the software installed on your computer. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e dlg.png})

1. Choose an object type, and then click **OK**.
2. BricsCAD opens the related application, in which you create an object (such as a spreadsheet or diagram) or use the application's **File | Open** command to open a file.
3. When done, choose **File | Close and Return to Drawing**in the application. The object appears in the upper left corner of the drawing, sometimes only as a blank rectangle.

Figure 6.1135



Command Options

Option	Description
Create New	Creates a new object by opening an application.These objects are also known as ”OLE objects”, short for Object Linking and Embedding.
Create From File	Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/ins 2-dlg.png})
File	Specifies the file name of the file to be inserted.
Browse	Displays the Browse dialog box; choose a file, and then click **Open**.
Link	Toggles linkage to the source document: • **On** - links the object in the drawing to the source file, so thatwhen the source file is changed, the linked object also changes • **Off** - does link not the object
Display As Icon	Displays the object as an icon. The icon is related to the document type and is defined by the source applications.

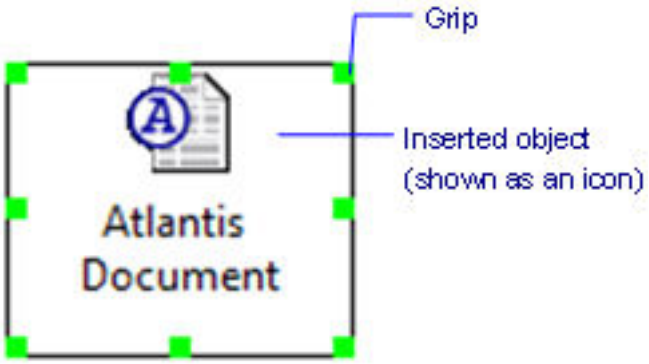
Change Icon	Displays the Change Icon dialog box: (\vref {fig:C:/Dev/Task-2448/images/storagegoogleapiscom/bricsys-static/help/enUS/V19/Commands/I/insertobj-changeicon-dlg.png}) 1. Choose another icon 2. Optionally edit the label text 3. Click **OK**to return to the previous dialog box.

Grips Editing

Select an OLE object by clicking on its frame. Eight grips appear: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricsc

Grips Editing	**Description**
Resize	Resizes the object by dragging any of the four corner grips. The aspect ratio is maintained.
Stretch	Stretches the object by dragging any of the four midpoint grips. The aspect ratio is not maintained.
Erase	Erases the selected object when the Del button is pressed.

Figure 6.1136



Related Commands

OleLinks (6.23.4) - adjusts links of OLE entities embedded in or linked to drawings. OleOpen (6.23.5) - opens linked and embedded objects in the appropriate application. PasteSpec (6.24.11) - pastes entities in several formats. Insert (6.17.13) - inserts DWG files in drawings. ImageAttach (6.17.5) - attaches raster images to drawings. XAttach (6.34.1) - attaches DWG files to drawings.

Figure 6.1137

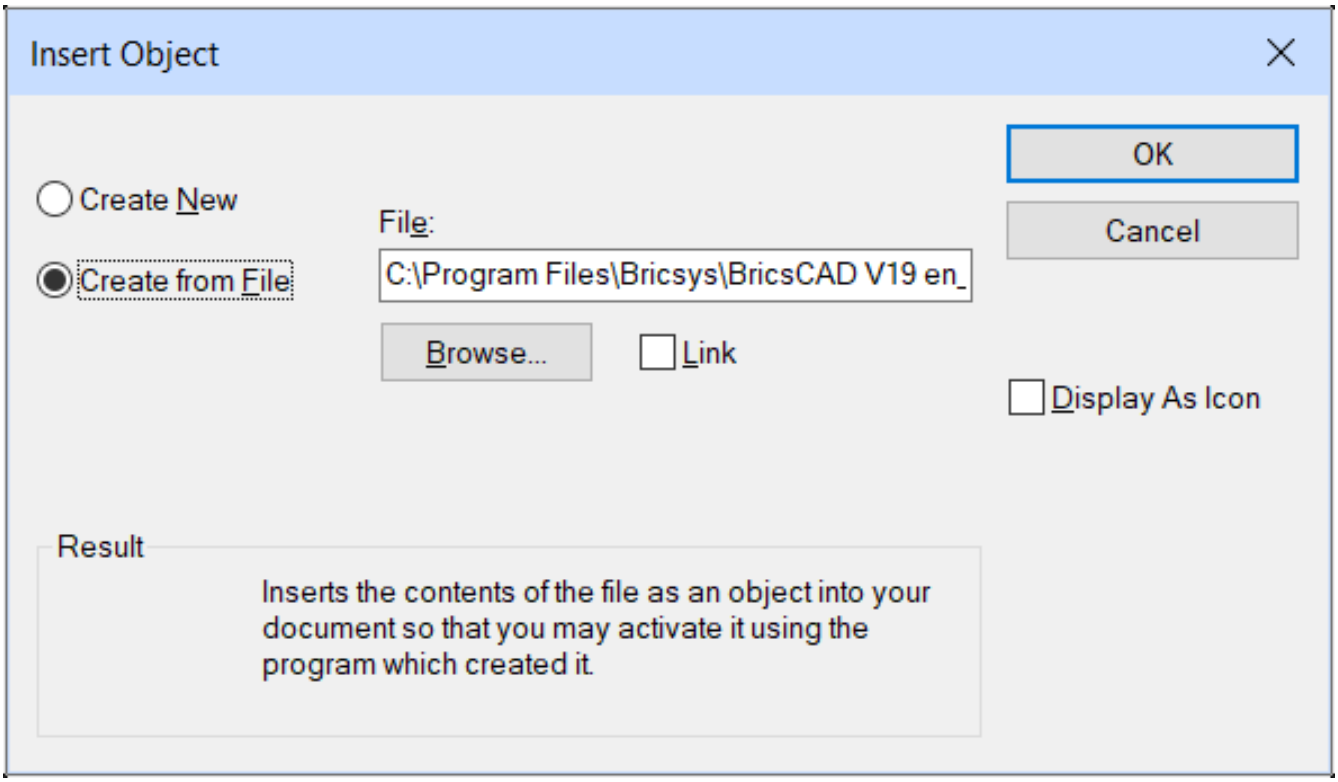


Figure 6.1138

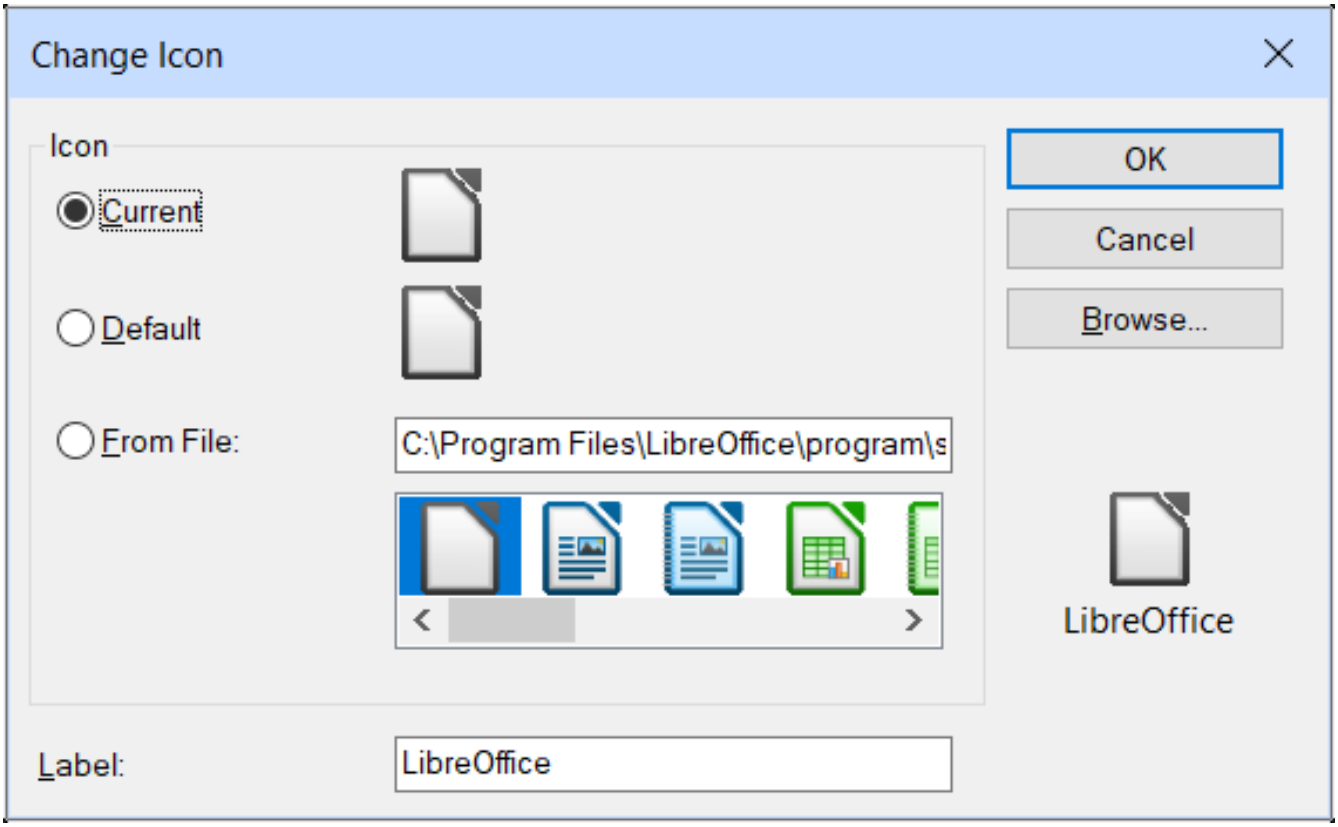
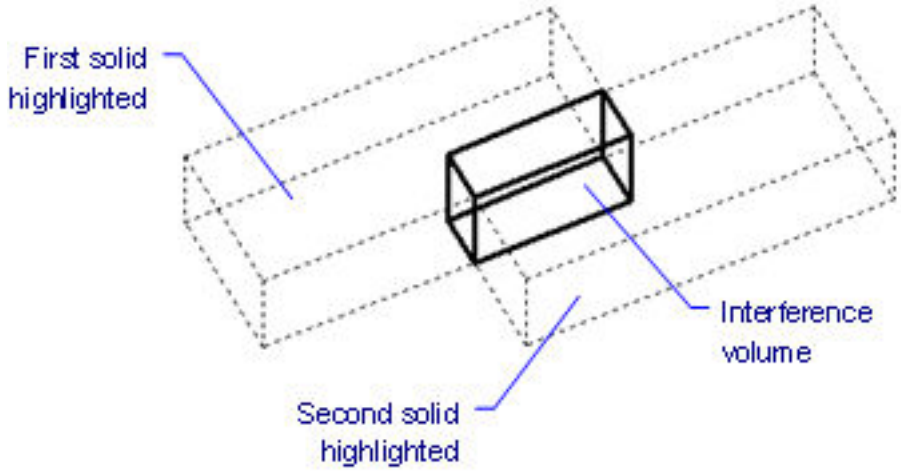


Figure 6.1139



Figure 6.1140



6.17.18 Interfere

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Shows volumes and areas of interference between two sets of ACIS entities. ThePlatinum edition additionally shows volumes of interference between two sets of ACIS solids, and then optionally creates new ACIS solids from the common portions of pairs of intersecting solids, placing them on layer "Interferences." (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image410.jpg})

Accessing the Command



command bar: **interfere** transparent: **'interfere** alias: **inf** menu bar: **Model | 3D Solids| Interfere** toolbar: **3D Solids |**  : **Interfere Prompts you in the command bar:** Select first set of Acis objects or [Nested selection/Settings] <Nested selection>: **(Choose one or more 3D solids or 2D regions to make up the first set, or else type N or S)** Entities in set: 1 Select first set of Acis objects or [Nested selection/Settings] <Nested selection>: **(Choose more ACIS entities, or else press Enter to continue)** Select second set of entities or [Nested selection/Settings/checK first set] <checK first set>:**(Choose one or more 3D solids or 2D regions to form the second set, or choose an option)** Entities in set: 1 Select second set of entities or [Nested selection/Settings/checK first set] <checK first set>: **(Choose more ACIS entities, or else press Enter to continue)** Comparing 1 solid against 1 solid. Interfering solids (first set): 1 (second set): 1 Interfering pairs: 1 Created 1 interference solid on layerInterferences The interference solid remains in the drawing when the command ends.

Command Options

Option	Description
Select first set of Acis objects	Specifies the first set of ACIS entities: choose one or more 3D solids and/or 2D regions. You can respond with **all** to place all ACIS entities into a set, and then later use the **checK first set** option to check them against each other; this obviates the need to create two sets of entities.

Nested selection	Selects ACIS entities inside blocks or xrefs; prompts you: 1. **Select nested Acis objects for first set [eXit] <eXit>** - **(select a nested ACIS entity.) Entities in set: 1** 2. **Select nested Acis objects for first set [eXit] <eXit>** - **(select more nested ACIS entities, or else press Enter to proceed.)**
Settings	Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/inte settings-dlg.png}) • **Create interference solids** - when on, makes new solids from the interference area or volume • **Layer** - specifies the layer on which to place the new interference entities • **Color** - specifies the color of the interference entities; the color of the layer is changed to match is color • **OK** - exit the dialog box and returns to the command
Select second set of Acis objects	Specifies the second set of ACIS entities: choose one or more 3D solids and/or 2D regions. The entities in the first set are compared with the entities in the second set.
checK first set	Checks the interference of entities in the first set, when it contains two or more entities, and then creates interference enty(ies) of them.

Related Commands

Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Region (6.26.18) - creates 2D regions from 2D entities. Revolve (6.26.30) - revolves 2D entities into 3D solid models. Section (6.27.11) - creates 2D regions from 3D models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Subtract (6.27.56) - removes one solid model from a second. Union (6.30.6) - joins two or more solid models into a single entity.

Figure 6.1141

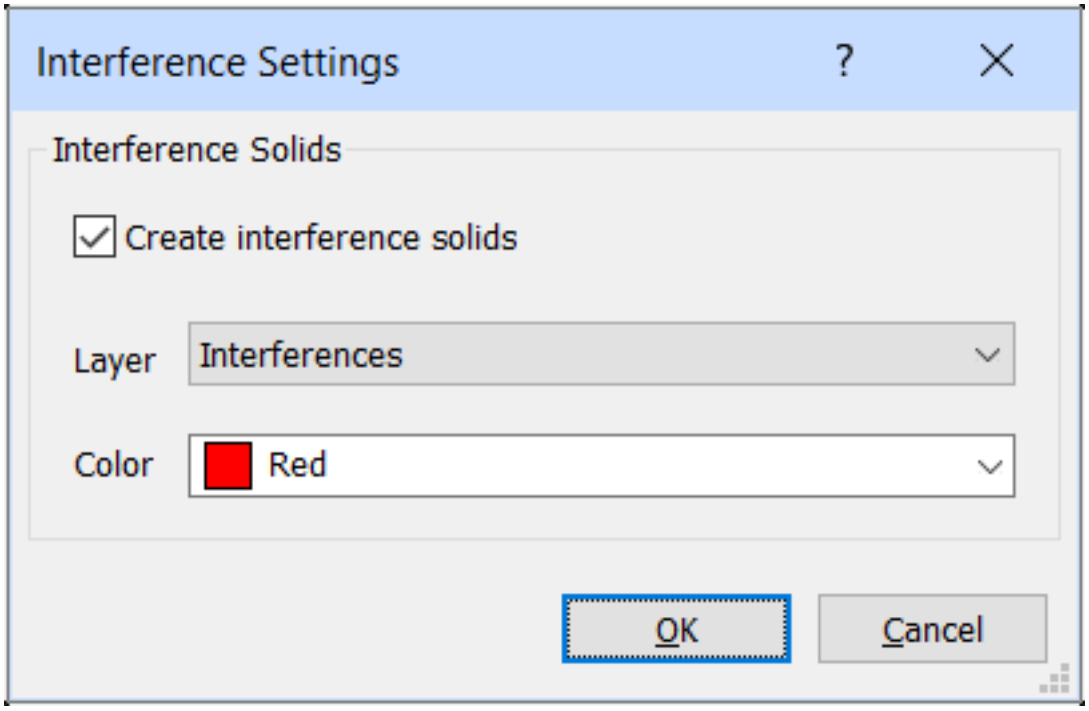


Figure 6.1142



6.17.19 Intersect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Performs Boolean intersection operations on 3D solids and 2D regions by removing all but portions in common.

Accessing the Command



command bar: **intersect** alias: **in** menu bar: **Model | 3D Solid Editing | Intersect** ribbon:**Solid | Edit | Boolean | Intersect** toolbar: **3D Solid Editing |**  : **Intersect Prompts you in the command bar:** Select ACIS object to intersect: **(Choose one 3D solid and/or 2D region)** Entities in set: 1 Select ACIS object to intersect: **(Choose another 3D solid and/or 2D region; you must choose at least two entities)** Entities in set: 2 Select ACIS object to intersect: **(Choose more ACIS entities, or else press Enter to perform the intersection operation)** Caution: if solids and regions do not intersect, BricsCAD erases them. The program removes all but the volumes and areas in common from the selected entities. Left: the entities selected; right: intersection result. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/IntersectResult.png}) This command also applies to regions: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/Intersec

Figure 6.1143

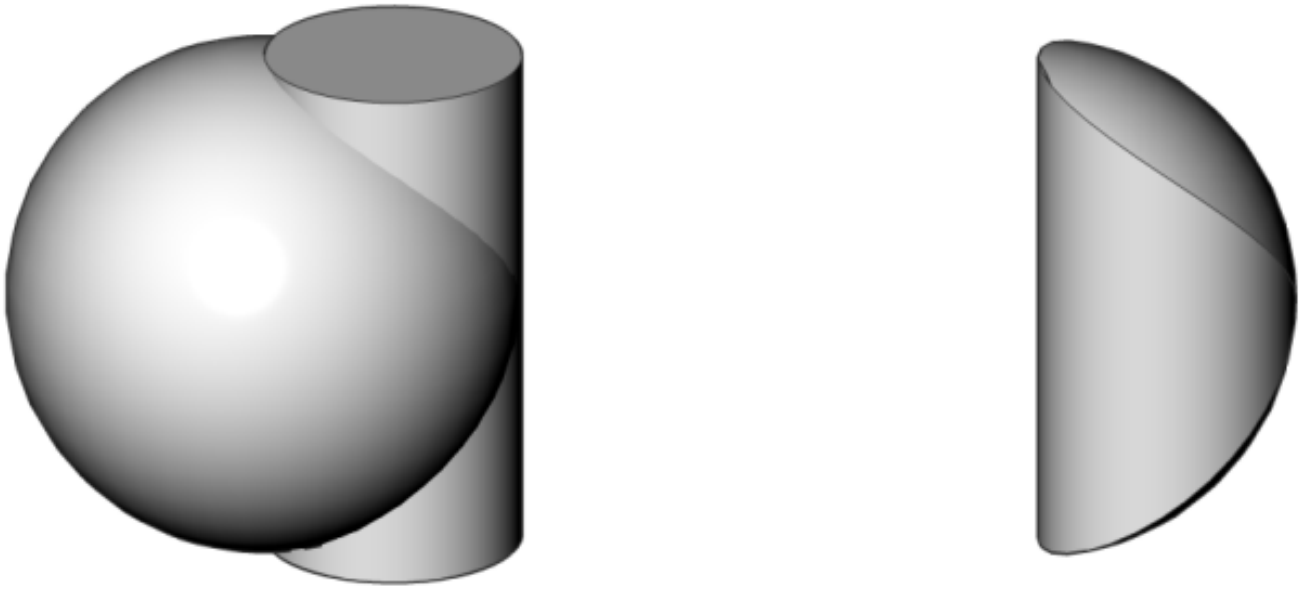
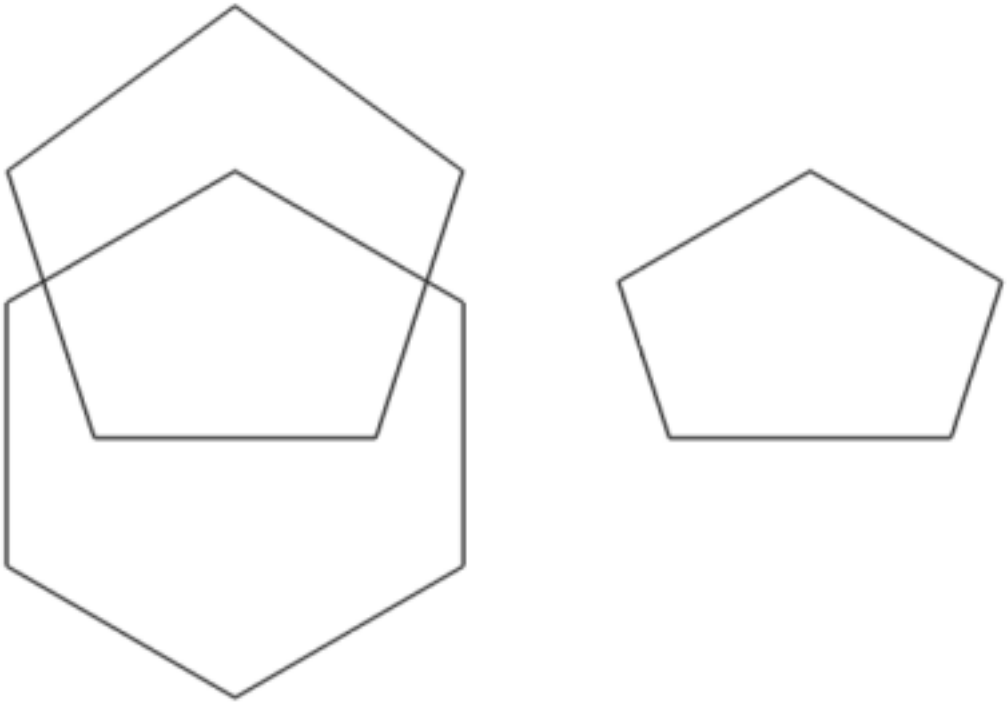


Figure 6.1144



Command Options

Option	Description
Select ACIS object to intersect	Specifies the 3D solids and/or 2D regions to be intersected.Choose two or more entities; non-ACIS entities are ignored. You may enter **all** to select all entities in the drawing. Caution: if solids and regions do not intersect, BricsCAD erases them. When this occurs, use the **U (6.30.1)** command to bring them back to life.

Related Commands

Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Region (6.26.18) - creates 2D regions from 2D entities. Revolve (6.26.30) - revolves 2D entities into 3D solid models. Section (6.27.11) - creates 2D regions from 3D models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Subtract (6.27.56) - removes one solid model from a second. Union (6.30.6) - joins two or more solid models into a single entity.

Figure 6.1145



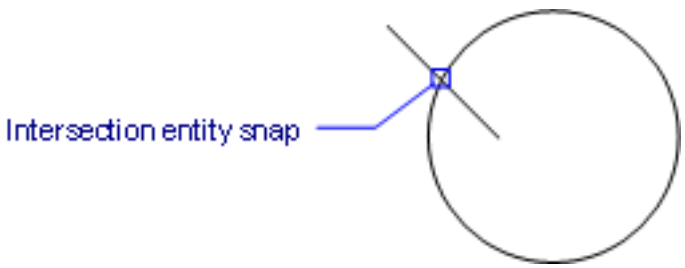
6.17.20 Intersection

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles Intersection entity snap; snaps to the intersections of entities.

Accessing the Command

command bar: **intersection** transparent: **int** alias: 3dintersection (6.3.6) toolbar: **Entity Snaps |** : **Intersection**
Displays no prompts in the command bar. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/Cr
When intersection entity snap is on, the cursor snaps to the intersections of crossing entities; it does not snap to apparent intersections in 3D space.

Figure 6.1146



Command Options

Option	Description
Intersection	Entering this command toggles Intersection entity snap. It snaps to the intersections of entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.1147



6.17.21 IsolateObjects

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Hides all but the entity(ies) selected.

Accessing the Command

 command bar: **isolateobjects** aliases: **isolate** toolbar: **Hide/Show Entities** |  Quad: **Custom** |  : **IsolateObjects Prompts you in the command bar:** Select entities to isolate: **(Select one or more entities)** Entities in set: 1 Select entities to isolate: **(Select more entities, or else press Enter)** Selected entities display only. Other entities are hidden. Use the UnisolateObjects (6.30.7) command to make the hidden entities visible again.

Command Options

Options	Description
Select entities to isolate:	Select one or more entities to remain visible; after you press Enter, all other entities become invisible.

The OBJECTISOLATIONMODE variable controls what happens to the the hidden state of the entities that weren't selected to be hidden: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/ObjectI

Figure 6.1148

[0] Objects are temporarily hidden for the current session. Interfered solids are selected together with the interference solids.
[1] Objects remain hidden between sessions. Interfered solids are selected together with the interference solids.
[2] Objects are temporarily hidden for the current session. Interfered solids are not selected together with the interference solids.
[3] Objects remain hidden between sessions. Interfered solids are not selected together with the interference solids.

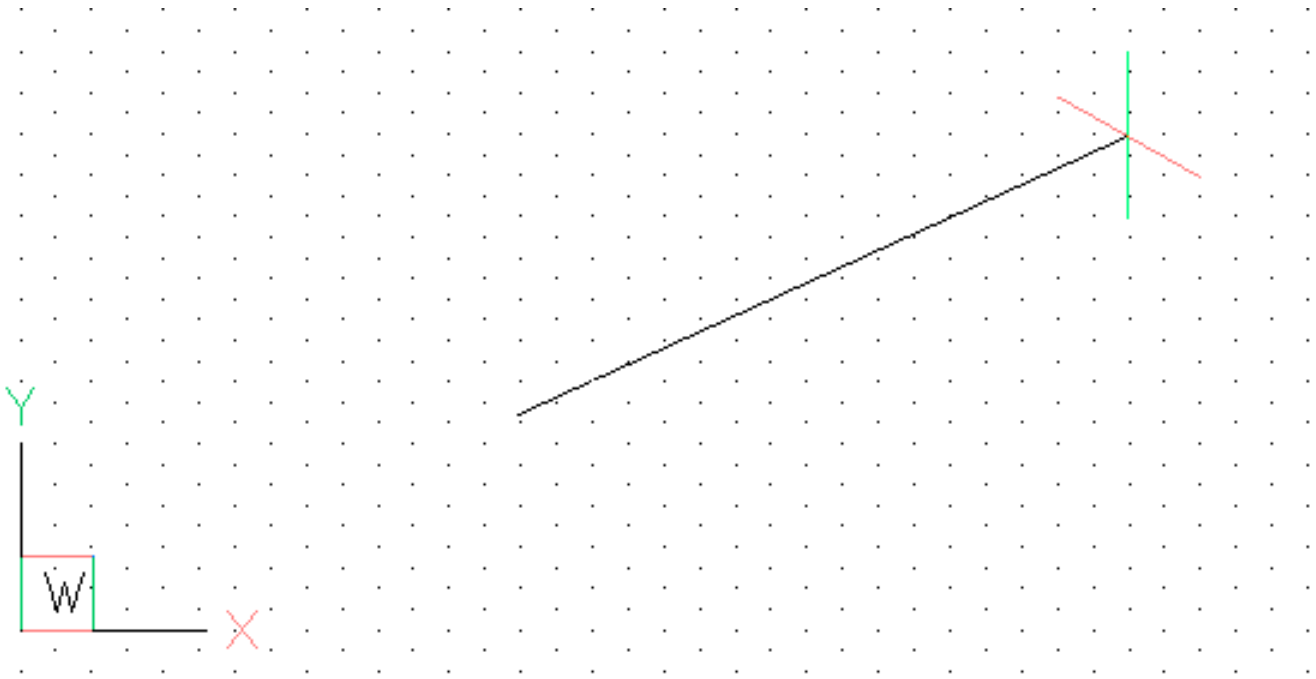
Related Commands

LayIso (6.20.9) - isolates layers associated with entities selected in the drawing; locks or turns off all other layers. HideObjects (6.16.11) - hides the selected entities. UnisolateObjects (6.30.7) - unhides entities that were hidden.

Figure 6.1149



Figure 6.1150



6.17.22 Isoplane

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Switches the isometric drafting plane and grid from left to top, to right. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/I/isometric-grid-eg.png})

Accessing the Command

command bar: **isoplane** transparent: **'isoplane** alias:is shortcut: **Ctrl+E F5** : **Isoplane Prompts you in the command bar:** Isoplane (Left) [TOGgle/keep using Left/switch to Right/switch to Top] <Switch to Top>: **(Enter an option, or else press Enter to accept the default)**

- To turn on isometric drafting mode: set the Snap (6.27.38) command's Style option to Isometric
- To turn on the isometric grid: set the Grid (6.15.21) command to Onand GridStyle to On.
- To draw isometric circles: use the Ellipse (6.13.5) command's Isocircle option.

The grid does not display in isometric mode when GridStyle is off (grid displayed as lines). The crosshair cursor automatically takes on the isometric slant.

Command Options

Option	Description
Left	Switches the isoplane to left.
Right	Switches the isoplane to right.
Top	Switches the isoplane to top.
TOGgle	Toggles the isoplane from left to top to right.
Keep using ...	Keep using the current isoplane orientation.

Related Commands

Snap (6.27.38) - toggles isometric drawing mode. Ellipse (6.13.5) - draws isometric circles.

6.18 J

6.18.1 Join

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Joins lines, lightweight and old-style 2D polylines, 3D polylines, arcs, elliptical arcs, polyarcs, splines, and helices at their common endpoints. The entity that results from the join depends on the input entities; see Rules, below.

Figure 6.1151



Accessing the Command

 command bar: **join** menu bar: **Modify | Join** ribbon: **Change | Modify | Break | Join** toolbar: **Modify |** : **Join Prompts you in the command bar:** Select source entity or multiple entities to join at once: **(Select one or more entities, or enter all to select all entities in the drawing)** Select entities to join: **(Select more entities, or else press Enter to join the selected entities)** 2 entities joined into 1 polyline, 1 entity discarded from the operation

Rules

(Collinear means the entities lie in the same imaginary line; coplanar means the entities lie in the same plane.) Two 2D polylines are joined into a single

- **2D polyline** - when they are coplanar.
- **3D polyline** - when they are not coplanar and if they are composed only of straight segments
- **Spline** - when they are not coplanar and if at least one of the polylines has a bulged (arc) segment

A 2D polyline and a 3D polyline are joined into a single

- **2D polyline** - when they are coplanar
- **3D polyline** - when they are not coplanar and if the 2D polyline only has straight segments
- **Spline** - when they are not coplanar and if the 2D polyline has at least one arc segment

A line and a circular arc (or a polyline with arcs) are joined into a single

- **2D polyline** - when they are coplanar
- **Spline** - when they are not coplanar

The following pairs of entities are joined into a spline

- Line and an elliptical arc
- Spline and another open entity, such as a elliptical arc or polyline
- Helix and another open entity, such as a line or arc

Even when there are gaps between these entities, the command joins them into a single entity:

- **Collinear lines** - joined into a single line
- **Coplaner arcs** (with same radii and center points) - joined into an arc or a circle
- **Coplaner elliptical arcs** (with same major and minor axes) - joined into an elliptical arc or ellipse,

Circular and elliptical arcs are joined in a counterclockwise direction starting from the source arc. To join (close) an arc, use the Circle (6.9.20) command's **turn Arc into circle** option.

Command Options

Options	Description
Select source entity or multiple entities to join at once	Specifies the entities to be joined; you may enter **all** to select all entities in the drawing, and the program will filter out invalid ones.
Select entities to join	Select more entities, or else press **Enter** to finish the command and start the joining.

Procedures

- To join colinear lines (1.23.25)
- To join two arcs (1.23.25)

Related Commands

Break (6.6.18) - breaks entities. Explode (6.13.13) - reduces complex entities to simpler ones, such as blocks into constituent parts, and polylines into lines and arcs. Overkill (6.23.14) - deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones. PEdit (6.24.20) - joins entities into a polyline.

6.19 K

6.19.1 KeepMe

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adds changed entities to the source drawing, when two somewhat different drawings are being compared visually. **Note:** The command works only during a session begun by the DWGcompare (6.10.41) command.

Figure 6.1152



Accessing the Command

 command bar: **keepme** toolbar: **Compare** |  : **KeepMe** Select entities: **(Select one or more entities, or else enter all to select all entities in the drawing)** Select entities: **(Select more entities, or else pressEnter to end the command)** 1266 entities were kept

Command Options

Option	Description
Select entities	Selects the entities to compare: • Select one or more entities • **All** - selects all entities in the drawing • **Enter** ends the command The entities you select are added to the source drawing.

Procedures

Merging the compared drawings (1.14.8)

Related Commands

DwgCompare (6.10.41) - compares a selected drawing with the current drawing during a comparison session. EndCompare (6.13.6) - exits a drawing comparison setting, started by the DwgCompare command.

6.20 L

6.20.1 LayCur

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Moves selected entities to the current layer.

Figure 6.1153



Accessing the Command


command bar: **laycur** menu bar: **Settings | Layers | Move to current layer** ribbon: **Home | Layers** toolbar: **Entity Properties** : **LayCur Prompts you in the command bar:** Select entities to be moved to the current layer: **(Select one or more entities)** Entities in set: xx Select entities to be moved to the current layer**: (Select more entities, or else press Enter to move the selection set to the current layer)** xx entities were moved to the current layer ("<Layer Name>")

Command Options

Option	Description
Select entities to be moved to the current layer	Select one or more entities, and then press **Enter**. The program changes the layers assigned to the entities to the current layer.

Related Commands

Change (6.9.10) - changes the properties of entities, through the command bar. ChProp (6.9.18) - changes the properties of entities, with fewer options than the Change command

Figure 6.1154



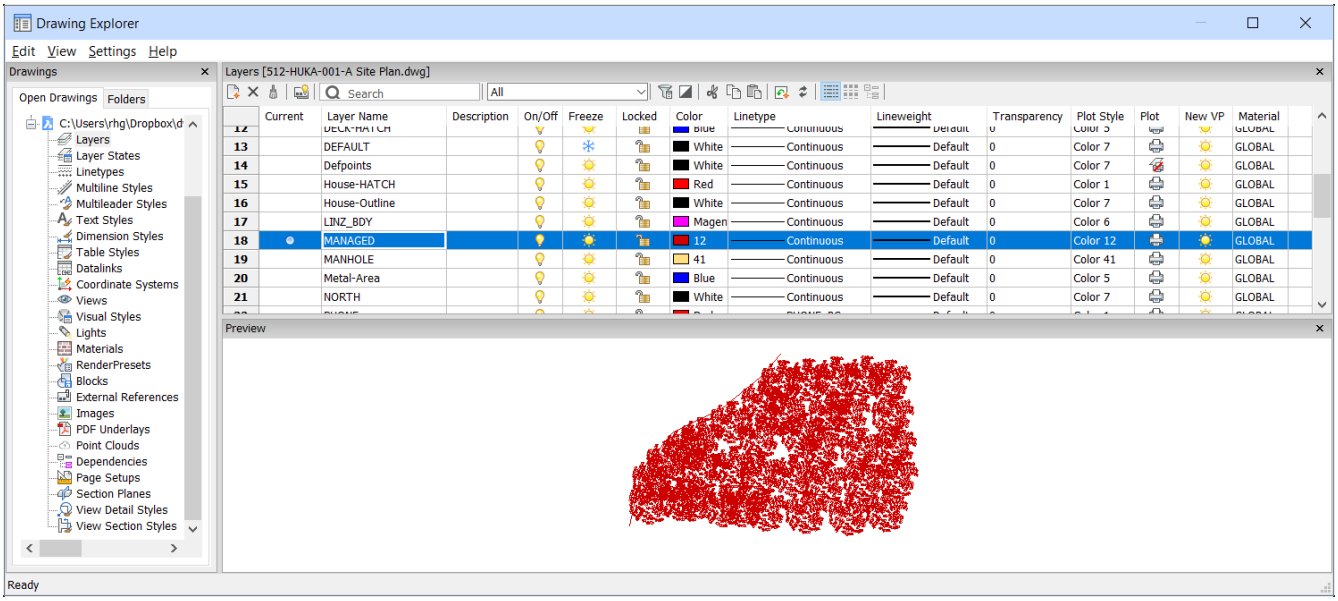
6.20.2 Layer

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates layers and layer states, changes their properties, and removes layers through the Drawing Explorer dialog box. **Note:** This dialog box must be closed to return to the drawing, unlike the Layers panel, which can remain open at all times; see the LayersPanelOpen (6.20.6) command.

Accessing the Command

 command bar: **layer** aliases: **la** **ddlmodes** **explayers** menu bar: **Settings | Layers** **Tools | Drawing Explorer | Layers** toolbar: **Entity Properties** | **Layers** | ribbon: **Home | Settings | Drawing Explorer | Layers** **Home | Layers | Layers** status bar: **Layer** field (**0**) : **Layer** Displays the Layers section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/layer-dlg.png}) Create layers or modify layer properties, and then click **X**. Press **F1** to access help.

Figure 6.1155



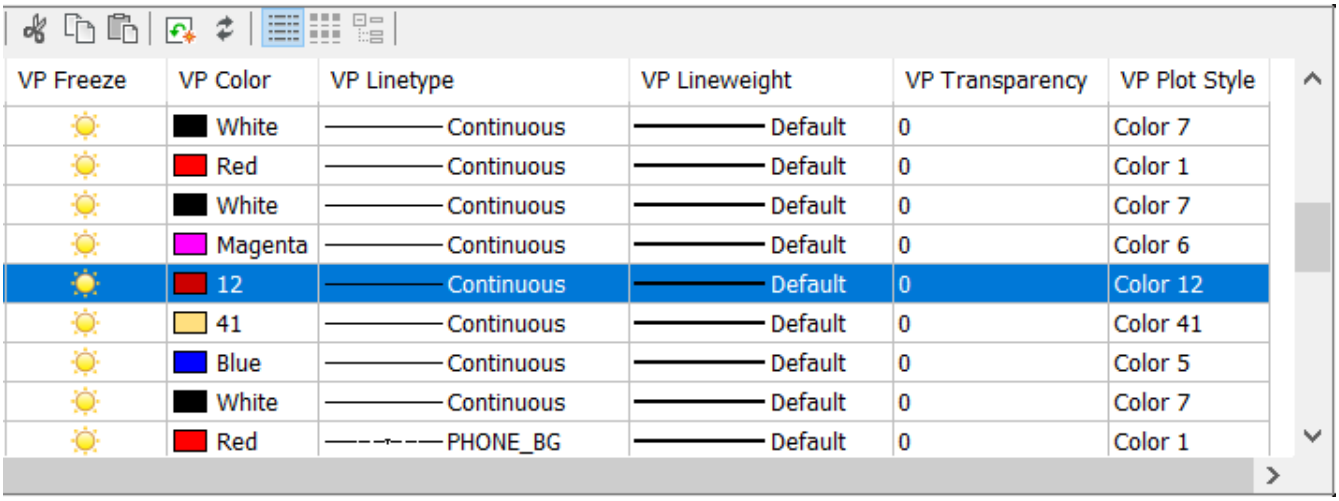
Properties of Layers in Viewports

When you open the Drawing Explorer - Layers dialog when in paper space, additional columns are displayed: **VP Freeze**, **VP Color**, **VP Linetype**, **VP Lineweight**, and **VP Plot Style**, where VP is short for viewport. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/layer-vl-dlg.png}) By default, these columns match the corresponding model space settings, but viewport layer properties can be set differently for each layout and for each paper space viewport.If the SHOWLAYERUSAGE variable is on, then the**Layer Usage** icons indicate when viewport settings for the current layout and paper space viewport are different from model space settings:

- : Current layer with viewport overrides.
- : Layer with viewport overrides.
- : Empty layer with viewport overrides.

If no paper space viewport is active, then the settings apply to entities in the current layout.If a viewport is active, thenviewport settings override the BYLAYER properties in the current viewport. The VP Freeze setting (or) allows you to control the visibility of model space entities in the current viewport.The New setting controls whether a layer is either Frozen ( ) are Thawed ( ) in new paper space viewports.

Figure 6.1156



Command Options









Option	Description
New	Creates new layers with the generic name "NewLayer1". Thenew layer that is created copies the properties of the currently selected layer.
Delete	Deletes layers, except for the following, whichcannot be deleted: • 0 • Any layer with entities
Purge	Purges layers from the current drawing, except for the following layers, which cannot be purged: • 0 • The current layer • Any layer with entities
Xref symbols On/Off	Toggle the display of Xref layers ("symbols"): • **On** - layers from xref'ed drawings are displayed • **Off** -layers from xref'ed drawings are not displayed
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/Cmd/Ref/source/LayerSearchField.png})	Filter the names of layers to shorten longer lists of layers. As you type characters, the program displays the layer names containing the text. Any layers containing the sequence of characters are listed, no matter where in the layer name the sequence is located.For example, entering "do" returns layers named DOOR and WINDOW.
Filter Used	Displays the current layer filter, if any are defined in the current drawing. Click the down-arrow button to choose a different layer filter The program creates the following layer filters automatically: • **All** - the only layer filter that appears in all drawings, and displays all layers • **Viewport Overrides** - you've overridden the properties of a viewport layer • **Xref** - the drawing has externally referenced drawings attached • 'xref name' - the name of each xref'ed drawing (note that layers from xref'ed drawings are prefixed automatically with the drawing's name)

Display Filter Panel	Toggles the display of the Filter Panel on and off. You can think of layer filters as "layer groups," groups of layers that you need to show or turn on-off all at once.The program creates layer filters in certain cases automatically, as described above. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/layer-filter-panel.png}) • To apply a filter, select its name in the list • To make a new layer filter, click the **New** button ( ); see the Layer Property Filter dialog box below • To delete selected filter(s), click the **Delete** button ( ); filters created by the program, such as All and Xrefs, cannot be deleted.
Invert Filter	Inverts the content of current filter, so that the dialog box displays all layers not included in the filter. For example, to see all layers **not** in an xref'ed drawing, choose the **Xref** filter, and then click **Invert Filter**.
Current	Specifies the current working layer. Only one layer can be current at a time. If the SHOWLAYERUSAGE variable is on, then following icons display in the **Layer Usage** column: • - Layer is current • - Layer is not empty • - Layer is empty In paper space layouts or viewports (model space with floating viewports), the following icons indicate when viewport overrides exist: • - Viewport overrides exist and layer is current • - Viewport overrides exist and layer is not empty • - Viewport overrides exist and layer is empty The **Indicate layers in use** item in the View menu toggles the SHOWLAYERUSAGE system variable ON/Off. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Header Options	**Description**

Layer Name	Specifies the name of the layer. In long lists of layers, you can typea character to jump to the first layer starting with that character, such as 'd' for DOORS. To sort the layer names from A to Z, click the **Layer Name** header; to reverse the order from Z to A, click the header a second time. To rename a layer: 1. Click the name twice, slowly (or right-click the layer name and then choose **Rename**) 2. Enter the new layer name
Description	(optional) Describes the content of the layer. Click the field, and then type.
On/Off	Toggles the selected layers between on and off. To change the status, select one or more layers, and then click a lightbulb icon: • **On** - displays entities on the layer • **Off** - hides entities from view; entities are included in the calculation of extents
Freeze	Toggles the selected layers between frozen and thawed.To change the status, select one or more layers, and then click a sun/snowflake icon: • **Thaw** - displays entities on the layer • **Freeze** - hides entities from view; entities are excluded in the calculation of extents, and are not exported with commands such as WBlock
Locked	Toggles the selected layers between locked and unlocked.To change the status, select one or more layers, and then click a padlock icon: • **Unlocked** - allows entities to be edited • **Locked** - prevents entities from being edited; entities are still visible
Color	Specifies the color of the layer. The default color is white (black).To change the color: 1. Choose one or more layers 2. Click a color square to display the Select Color dialog box; see Color (6.9.33) command 3. Choose a color, and then click **OK**

Linetype	Specifies the linetype of selected layers. The default linetype is ”Continuous.” To change the linetype: 1. Choose one or more layers 2. Click a linetype sample 3. Choose one from the droplist (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ To load additional linetype definitions: 1. Click the **Load** option. 2. In the Load Linetypes dialog box, choose a linetype, and then click **OK**. See Linetype (6.20.33) command.
Lineweight	Specifies the linetype of selected layers. Default is ”Default.” To change the lineweight: 1. Choose one or more layers 2. Click the lineweight sample 3. From the droplist, choose another lineweight (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ See LWeight (6.20.41) command.
Transparency	Sets the transparency of entities on the selected layer(s) between 0 (fully opaque) and 90 (fully transparent); the default is 0.To change the transparency: 1. Choose one or more layers 2. Click the transparency number 3. From the droplist, choose another number The 2DWireframe visual style ignores this setting; instead, use the TRANSPARENCYDISPLAYvariable to control the level of transparency.
Plot Style	Specifies the plot style of selected layers, if plot styles are enabled in the current drawing. The default is ”Color_7.” To change the plot style” 1. Choose one or more layers 2. Click a plot style name 3. From the droplist, choose another style name By default, plot styles are not enabled in drawings. To add plot styles to the drawing, see thePlotStyle (6.24.29) command.

Plot	Toggles selected layers between being printed and not printed.To change the status, select one or more layers, and then click the printer icon: •  **Print**- prints the layer •  **No Print**- does not print the layer Some layers created by the program also do not print by default, such as DefPoints.
New VP	Specifies whether the layer is visible (thawed) or invisible (frozen) when you make a new viewport (VP) in a layout tab.To change the status, select one or more layers, and then click a sun/snowflake icon: •  **Thaw** - displays entities on the layer in the new viewport •  **Freeze** - initially hides entities from view when the viewport is created See the MView (6.21.37) command.
Material	Specifies the material of selected layers. The default is material is name ”Global.” To change the material assigned to one or more layers: 1. Select one or more layers. 2. Click the **Material** field. 3. Click again to display the droplist, containing the materials availablein the drawing. 4. Select a material in the list. When you need to add more materials to the drawing, see the Materials (6.21.11) command.
Preview	Displays the entities on the selected layer.

Viewport Headers

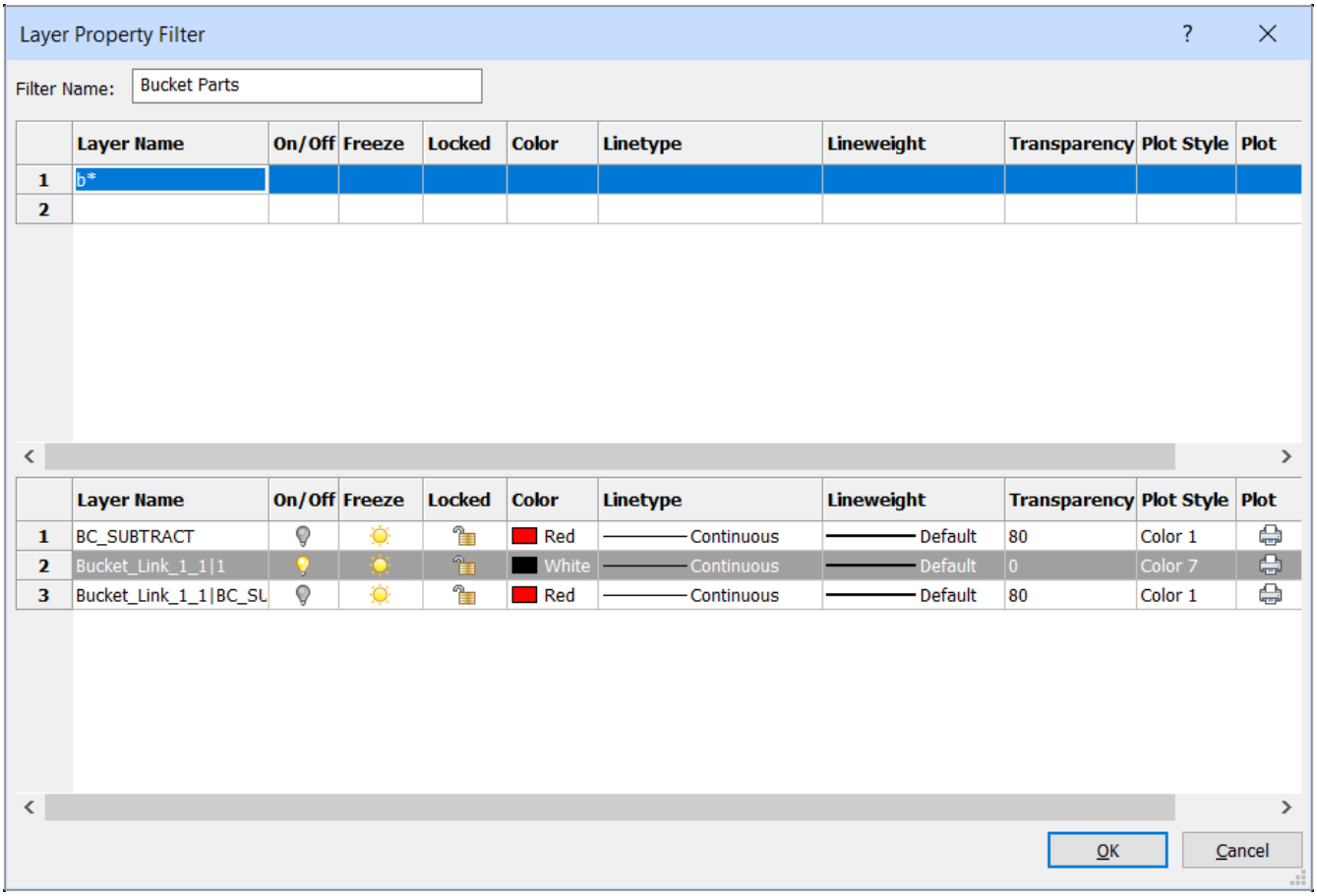
The following headers appear when you switch to from model tab to a layout tab, and then open the Drawing Explorer:

Header Option	Description
VP Freeze	Freezes the layer in the current layout; overrides the freeze/thaw setting in model space
VP Color	Specifies the color of entities on this layer in the current layout; overrides the color setting in model space
VP Linetype	Specifies the linetype for entities in the current layout; overrides the linetype setting in model space
VP Lineweight	Specifies the lightweight in the current layout; overrides the lightweight setting in model space
VP Transparency	Specifies the transparency of entities on this layer in the current layout; overrides the transparency setting in model space
VP Plotstyle	Specifies the plot style name in the current layout; overrides the plot style setting in model space

About Layer Filters

You can think of layer filters as ”layer groups,” groups of layers that you need to show or turn on-or-off all at once.The program creates layer filters in certain cases automatically, as described above. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/layer-filter-panel.png})

Figure 6.1157



- To apply a filter, select its name in the list
- To make a new layer filter, click the **New** button ( ); see the Layer Property Filter dialog box below
- To delete selected filter(s), click the **Delete** button ( ); filters created by the program, such as All and Xrefs, cannot be deleted

To create a new layer filter definition:

1. Click the **New** button ( ) on the Filter Panel toolbar.The Layer Property Filter dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/Layer-state-dlg.png})
2. **Filter Name** - enter a name for the filter in the Filter Name field. It can be descriptive of which layers the filter displays, such as 'Door and Windows.'
3. **Layer Name** - enter a layer name in theLayer Name field. You can use wildcards, such as ? and * to specify more than one layer with a similar name, such as Doors-* for Doors-Exterior and Doors-Interior.
4. **Properties** - to specify which layers to include according to their properties, select a property (such as Freeze), and the filter includes all layer(s) that currently are frozen.
5. Click**OK**to save the layer filter.
6. To use a layer filter, click on its name in either the Filters panel or from the Filters droplist. Notice that the layers shown in the Drawing Explorer changes.

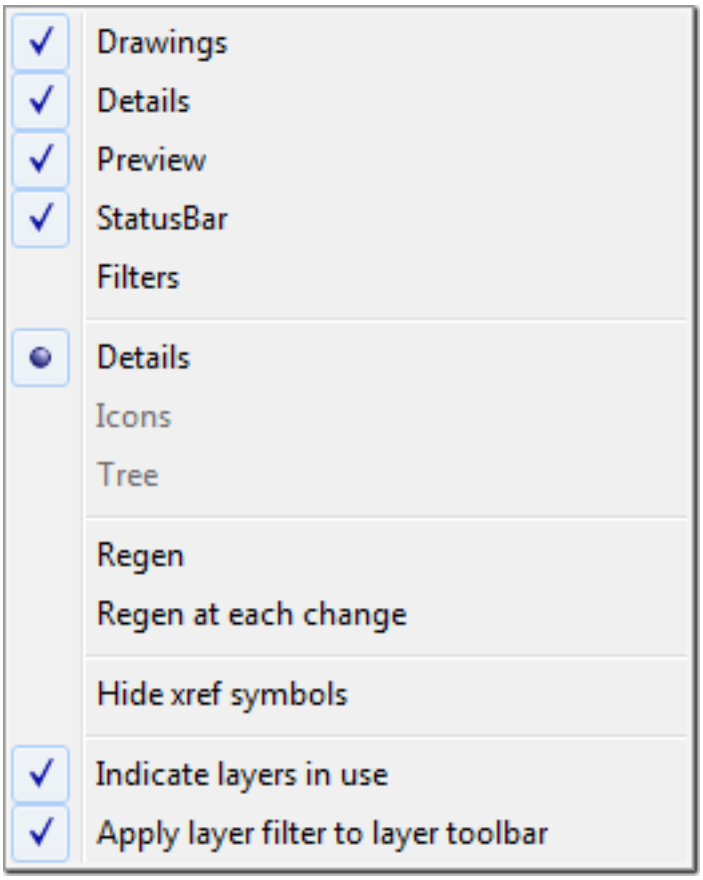
About the View Menu

The **View** menu on the Drawing Explorer's menu bar contains options specific to layers: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/layer-View-mnu.png})

Option	Description

Drawings	Toggles the display of the Drawings panel (left side of the Drawing Explorer).
Details	Toggles the display of the Layerspanel.
Preview	Toggles the display of the Preview panel at the bottom of the Drawing Explorer.
StatusBar	Toggles the display of the Status Barat the very bottom of the Drawing Explorer.
Filters	Toggles the display of the Filters panel.
Details Icons Tree	Only the Details view is available for layers.
Regen	Forces the drawing display to be regenerated. See the Regen (6.26.15) command.
Regen at Each Change	Toggles the automatic regeneration of the drawing display each time you make a change in the Drawing Explorer.
Show/Hide XRef Symbols	Toggles the display of layers from externally-referenced drawings (the word Symbols should be 'Drawings').
Indicate Layers in Use	Toggles the SHOWLAYERUSAGE variable On/Off to show or not show the in-use and not-in-use icons in the Current column: •  - Layer is not empty •  - Layer is empty
Apply layer filter to layer toolbar	Applies the current layer filter to the **Layer** control on the **Entity Properties** toolbar.

Figure 6.1158



About Wild-Card Characters

Wild-card characters can be used in the Filter text field and in Filter definitions to filter layers by name.

Character	ASCII code*	Description
#	35	# matches any digit For example, ## matches 12 and 34 but not 5 or 123
@	64	Matches any alphabetic character, so not digits or symbols For example, @@@@@ matches all 5-letter layer names
.	46	Matches non-alphanumeric characters, so symbols but not letters or numbers For example,.* matches all layer names starting with an non-alphanumeric character such as _SITEPLAN and +1_LEVEL
	42	Matches any string, including letters, numbers, and characters. For example, A* matches all layer names starting with the letter A
?	63	Matches any single character. For example, ?AB* matches L**AB**ORATORY and C**AB**ARET but not ST**AB**LE, because STABLE begins with two letters and the ? specifies a single letter

~	126	Negative match, in whichany string but the pattern is returned. For example, ~*A* matches all layer names not containing the letter A
[]	91 93	Matches any one of the characters enclosed inside the square brackets. For example, LA[YB] matches LA**Y** and LA**B**
[~]	91 126 93	Specifies any one of the characters not found in the square brackets. For example, LA[~YB] matches L**AD** but not L**AY** or L**AB**
[-]	91 4593	Specifies a range for a single character in the layer name. For example, LAY-[A-E] matches any combination of LAY- and A through E, such as LAY-**A** ... LAY-**E** but not LAY-F
‘	96	Matches the next string exactly tofind a layer name that contains a wild-card character when it is prefixed by the reverse quote character. Use ‘ to search for layer names that include characters like ? and ~. For example,‘~LAY* matches ~LAY-1,~LAYER-1, and ~LAY-B.

* ASCII code: if a character is not available on the keyboard, press and hold the **Alt**-key and then type the ASCII code on the numeric keypad.

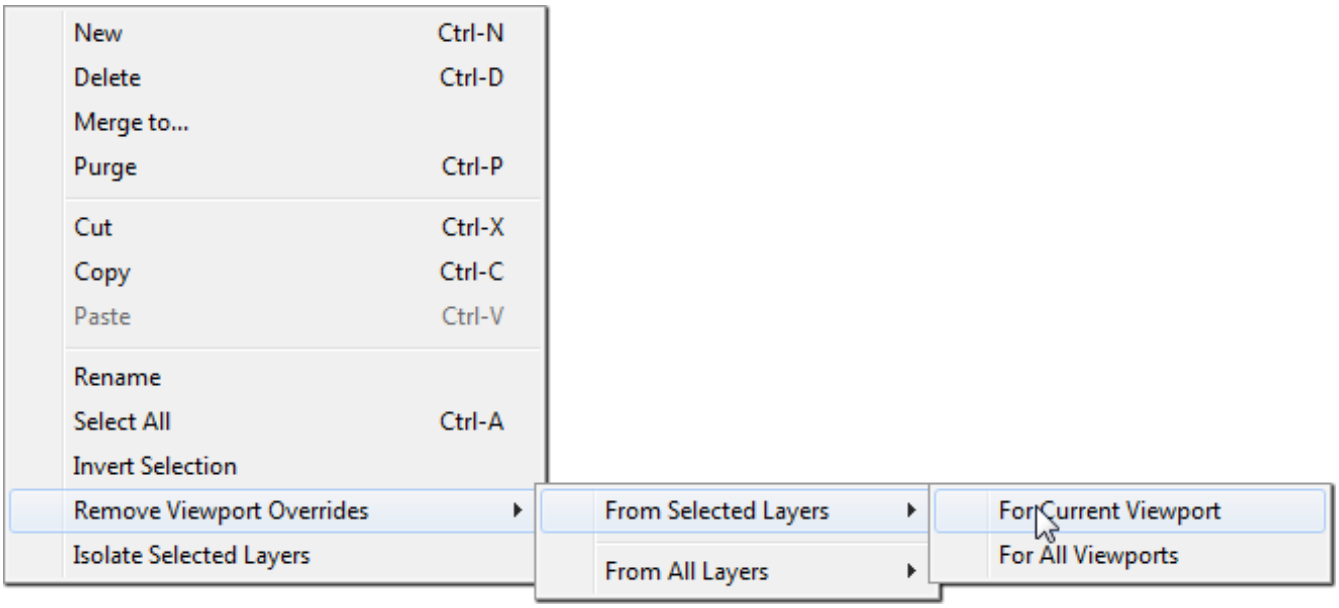
Shortcut Menu Options

Right-click a layer name to see the following shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS rightclick-layouts-mnu.png}) The keystrokes listed work only in the Drawing Explorer.

Option	Keystroke	Description
New	Ctrl+N	Creates a new layer.
Delete	Ctrl+D	Deletes layers from the drawing, except for the following ones: • 0 • Any layer with entities
Purge	Ctrl+P	Purges layers from the current drawing, except for the following ones: • 0 • Current layer • Any layer with entities
Cut	Ctrl+X	Cuts the layer and its properties to the Clipboard, which can then be pasted elsewhere in the Drawing Explorer, such as into the layer system of another drawing.
Copy	Ctrl+C	Copies the layer and its properties to the Clipboard.

Paste	Ctrl+V	Pastes the layer from the Clipboard into Drawing Explorer.
Rename		Renames the selected layer.
Select All		Selects all layers.
Set Current		Specifies the current working layer. Only one layer can be current at a time.
Remove Viewport Overrides		Removes overrides from properties that were set differently for viewports: **From Selected Layers** - **(select an option)**: • **For current viewport** - removes overrides from the current viewport • **For all viewports** -removes overrides from all viewports in the drawing **From All Layers** - removes the overrides form all layers in the drawing for either the current viewport or all viewports.
Isolate Selected Layers		Dims the entities on all other layers, except for the one(s) selected. Use the LayUnIso (6.20.18) command to unisolate layers. See the LayIso (6.20.9) command.

Figure 6.1159



About the Status Bar Menu

Right-click **0** (or other layer name) on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsysco
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image528.jpg})

Shortcut Option	Description
0 Defpoints	Chooses the current working layer.When the drawing contains additional layers, their names are listed here.
Properties	Opens the Layers section of the Drawing Explorer.

Figure 6.1160

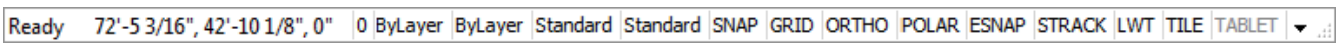
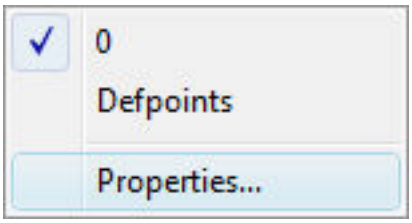


Figure 6.1161



Procedures

- To merge the content of two layers (1.14.2)

Related Commands

-Layer (6.20.3) - creates layers and changes their properties in the command bar. LayersPanelOpen (6.20.6) - opens the Layers panel, which allows to edit the layers in the current drawing, apply layer states and layer filters. LayersPanelClose (6.20.5) - closes the Layers panel. LayerState (6.20.7) - saves and restores layer states. LayMCur (6.20.11) - changes the working layer to that of a selected entity LayFrz (6.20.8) - freezes layers associated with selected entities. LayThw (6.20.16) - thaws all frozen layers. LayIso (6.20.9) - isolates layers associated with selected entities. LayLck (6.20.10) - locks layers associated with selected entities. LayUlk (6.20.17) - unlocks layers associated with selected entities. LayOff (6.20.12) - turns off layers associated with selected entities. LayOn (6.20.13) - turns on all layers that have been turned off. Materials (6.21.11) - creates materials and edits their properties through the Drawing Explorer. VpLayer (6.31.18) - changes the properties of layers in the current paperspace viewport. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.1162



Figure 6.1163

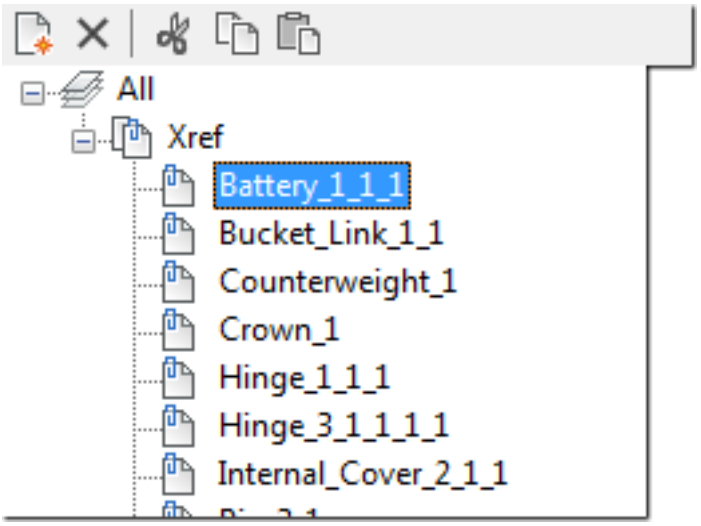


Figure 6.1164

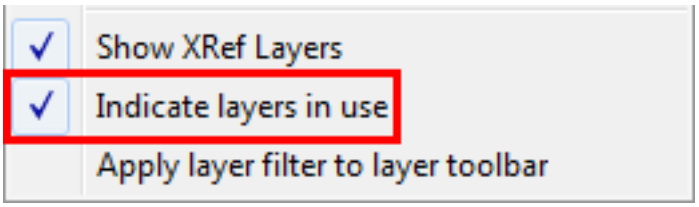


Figure 6.1165

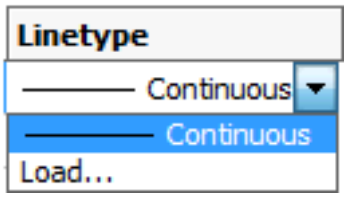


Figure 6.1166

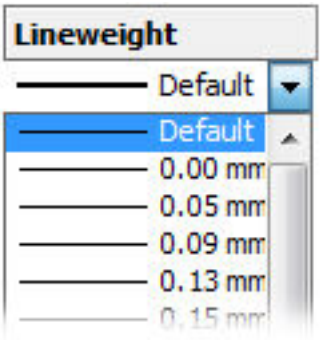


Figure 6.1167

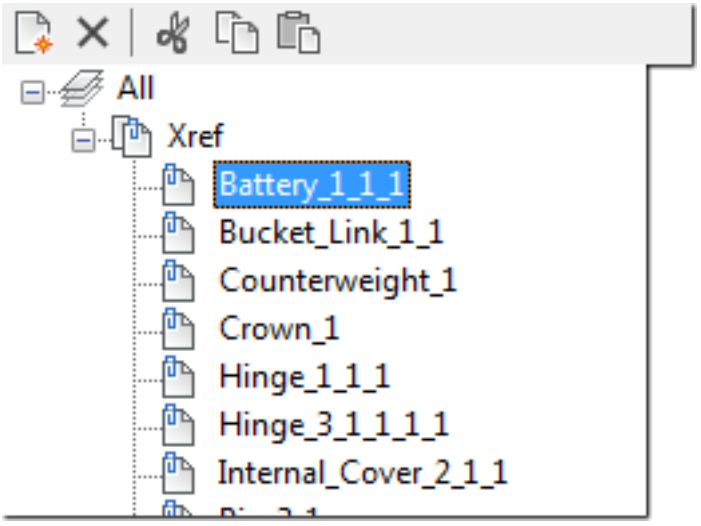


Figure 6.1168



6.20.3 -Layer

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and removes layers and layer states, and changes their properties through the command bar.

Accessing the Command

command bar: **-layer** transparent: **'-layer** alias: **-la** : **-Layer Prompts you in the command bar:** Layer [? to list/ New layer/ Make new current layer/ Set layer as current/ Color/ Linetype/ LineWeight/ TRansparency/MATerial/Plot/stAte/turn layer ON/turn layer OFf/ Freeze/ Thaw/ LOck/ Unlock]: **(Enter an option, or else press Enter to end the command)**

Command Options

Option	Description
? to list	Lists the names of layers in the current drawing. Press **F2** to flip to the Prompt window to see the list. Use wildcards, such as ? and *, to narrow down the list of names.
New layer	Creates new layers; prompts you: **Names for new layers** - **(enter one or more layer names.To create several new layers at the same time, separate each name with a comma, such as doors,windows,walls.)**
Make new current layer	Creates a new layer, and then makes it current. This option combines the New and Set options; prompts you: **New layer to make current** - **(enter a layer name; only one layer may be current.)**
Set layer as current	Sets a layer as the current one; prompts you: **New layer to make current** - **(enter a layer name; only one layer may be current.)**
Color	Changes the color of layers; prompts you: 1. **Color** - **(specify the color by number or name; see theColor (6.9.33) command.)** 2. **Layer(s) to be color<layername>** - **(specify the layer names or press Enter to change the color of the most-recently used layer; separate more than one layer name with commas.)**
Linetype	Changes the linetype of layers; prompts you: 1. **Linetype: ? to list <Continuous>** - **(enter an linetype name or an option)**; see the Linetype (6.20.33) and -Linetype (6.20.32) commands **? to list** - Lists the names of linetypes defined in the current drawing 2. **Layer(s) for linetype CONTINUOUS <CurrentLayer>** - **(enter the name(s) of layers, separating names with a comma, or else press Enter to apply the linetype to the default layer.)**

LineWeight

Changes the lineweight of layers; prompts you:

1. **Enter lineweight (0.00 - 2.11mm)** - **(enter a number between 0.00 and 2.11.)**
2. **Enter name list of layers(s) for lineweight 0.00 mm <CurrentLayer>** - **(enter the name(s) of layers, separating names with a comma, or else press Enter to apply the line weight to the default layer.)**

TRansparency

Sets the entity transparency factor between 0 (fully opaque) and 90 (fully transparent); prompts you:

- **Enter transparency (0 - 90)** - **(enter a number between 0 and 90; default is 0.)**
- **Layer(s) for transparency 0 <CurrentLayer>** - **(enter the name(s) of layers, separating names with a comma, or else press Enter to apply the transparency level to the default layer.)**

The 2DWireframe visual style ignores this setting; for it, use the TRANSPARENCYDISPLAYvariable to control the level of transparency.

MATerial

Sets the material property for all entities on the layer; prompts you:

1. **Material: ? to list/<Global>** - **(enter an option)**: **? to list -** Displays a list of the materials and their description in the Prompt History window
2. **Layer(s) for material GLOBAL <CurrentLayer>** - **(enter the name(s) of layers, separating names with a comma, or else press Enter to apply the material to the default layer.)**

The default material is Global, which cannot be deleted or renamed. When you need to add more materials to the drawing, see the Materials (6.21.11) command.

Plot

Changes the plot status of layers (plot or no plot); prompts you:

1. **Enter a plotting preference [Plot/No] <Plot>** - **(enter an option.)**
2. **Enter layer name(s) for this plot preference <CurrentLayer>** - **(enter the name(s) of layers, separating names with a comma, or else press Enter to apply the plot status to the default layer.)**

stAte

Creates, edits, renames, restores, deletes, imports, and exports layer states. See the LayerState (6.20.7) command.Prompts you: **Enter an option [? to list layer states/ Save/ Restore/ Edit/ reName/ Delete/ Import/ eXport]** - **(enter an option)**:

- **? to list layer states** - lists the names of saved layer states and their description in the Prompt History (6.29.13) window
- **Save** - saves the current layer state; prompts you:

1. **Enter a name for the new Layer State** - **(type a name.)**
2. **Enter state to change [On/ Frozen/ Lock/ Color/ lineType/ lineWeight/plot Style/ Plot/New vp frozen]** - all states are on by default; **(enter a state name to toggle its setting.)**

- **Restore** - Restores a saved layer state; prompts you:

1. **Enter name of Layer State to restore or [?]** - **(type the name of saved layer state or type ? to list the saved layer states.)**

- **Edit** - Edits a saved layer state. prompts you:

1. **Enter the name of the Layer State to edit or [?]** - **(type the name of saved layer state or type ? to list the saved layer states.)**

- **Rename** - Renames a saved layer state; pPrompts you:
- **Type the name of saved layer state or type ? to list the saved layer states** - **(type the name of saved layer state or type ? to list the saved layer states.)**
- **Enter the new name** - **(type the new name for the selected layer state.)**
- **Delete** - deletes a saved layer state; prompts you:
- **Enter the name of the Layer State to delete or [?]** - **(type the name of saved layer state or type ? to list the saved layer states.)**
- **Import** - imports a layer state from a file into the current drawing:

1. Displays a File dialog. Select a layer state file (*.las).
2. Click the **Open** button

- **Export** - exports a saved layer state in the current drawing to a layer state file (*.las). Prompts you:

1. **Enter the name of the Layer State to export or [?]** - **(type the name of saved layer state or type ? to list the saved layer states.)**
2. Displays a File dialog. Type a name for the layer state file, then press the **Save** button.

turn layer ON	Turns layers on that had been turned off with the OFf option; prompts you: **Layer(s) to turn ON** - **(specify the names of layer name(s) to turn on; separating multiple layer names with commas.)**
turn layer OFf	Turns layers off to hide their entities from view; prompts you: **Layer(s) to turn OFF** - **(specify the names of layer name(s) to turn off; separating multiple layer names with commas.)** The current layer cannot be turned off. When layers are off, they are hidden from view. In most cases, you should probably use the Freeze option, as it offers more features.
Freeze	Freezes layers so that they are hidden from view; prompts you: **Layer(s) to Freeze** - **(specify the names of layer name(s) to freeze; separating multiple layer names with commas.)** The current layer cannot be frozen. When layers are frozen... • Entities are hidden from view • Entities cannot be edited • The extents of entities are ignored when the program calculates the drawing extents • Entities are not included in operations, such as ViewBase, Print, Export, and PDFout.
Thaw	Thaws layers that were previously hidden with the Freeze option; prompts you: **Layer(s) to Thaw** - **(specify the names of layer name(s) to thaw; separating multiple layer names with commas.)**
LOck	Locks layers so that their entities remain visible but cannot be edited; prompts you: **Layer(s) to Lock** - **(specify the names of layer name(s) to lock; separating multiple layer names with commas.)** The current layer **can** be locked.
Unlock	Unlocks layers that were locked form editing with the LOck option; prompts you: **Layer(s) to Unlock** - **(specify the names of layer name(s) to unlock; separating multiple layer names with commas.)**

Related Commands

Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. LayersPanelOpen (6.20.6) - opens the Layers panel, which allows to edit the layers in the current drawing. LayersPanelClose (6.20.5) - closes the Layers panel. LayerState (6.20.7) - saves and restores layer states. LayMCur (6.20.11) - changes the working layer to that of a selected entity LayFrz (6.20.8) - freezes layers associated with selected entities. LayThw (6.20.16) - thaws all frozen layers. LayIso (6.20.9) - isolates layers associated with selected entities. LayLck (6.20.10) - locks layers associated with selected entities. LayUlk (6.20.17) - unlocks layers associated with selected entities. LayOff (6.20.12) - turns off layers associated with selected entities. LayOn (6.20.13) - turns on all layers that have been turned off.

Figure 6.1169



6.20.4 LayerP

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Undoes the previously applied change to layer properties and settings (short for Layer Previous). **Note:** This command operates only whenLAYERPMODE (6.20.4) variable is on.

Accessing the Command

 command: **layerp** menu bar:**Settings | Layers | Layer Previous** tooblar:**Settings | Layers** |  : **LayerP** Previous layer settings were restored.

Command Options

No options. The following changes cannot be undone by the LayerP command:

- Layer names were renamed
- Layers that were deleted or purged (use the U command to bring back deleted layers)
- Layers were added
- No layer changes were made

This command has no effect when the LAYERPMODE variable is set to off.

Related Commands

Layer (6.20.2) - creates layers, changes their properties, and removes layers through the Drawing Explorer. -Layer (6.20.3) - creates layers, changes their properties, and removes layers through prompts in the command bar.

Figure 6.1170



6.20.5 LayersPanelClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes the Layers panel opened by the LayersPanelOpen (6.20.6) command.

Accessing the Command

command bar: **layerspanelclose** : **LayersPanelClose** The Layers panel is closed.

Command Options

No command options.

Related Commands

LayersPanelOpen (6.20.6) - opens the Layers panel, which allows to edit the layers in the current drawing. -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Figure 6.1171



6.20.6 LayersPanelOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Layers panel, for editing layer names and properties in the current drawing, as well as create, edit, and apply layers, layer states, and layer filters. **Note:** This panel can remain open at all times, unlike the Drawing Explorer displayed by the Layer command.

Accessing the Command

command bar: **layerspanelopen** context menu: right-click the ribbon or any toolbar, and then choose **Layers** : **LayersPanelOpen** The Layers panel displayed when the drawing is in Model space: (\vref {fig:C:/Dev/Task-2448/images/staticbric
panel.png}) The Layers panel displayed additionalVP layer properties whenthe drawing is in a layout tab, which apply to the current paper space layout or a viewport in the current layout. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel
vp1-panel.png})

Figure 6.1172

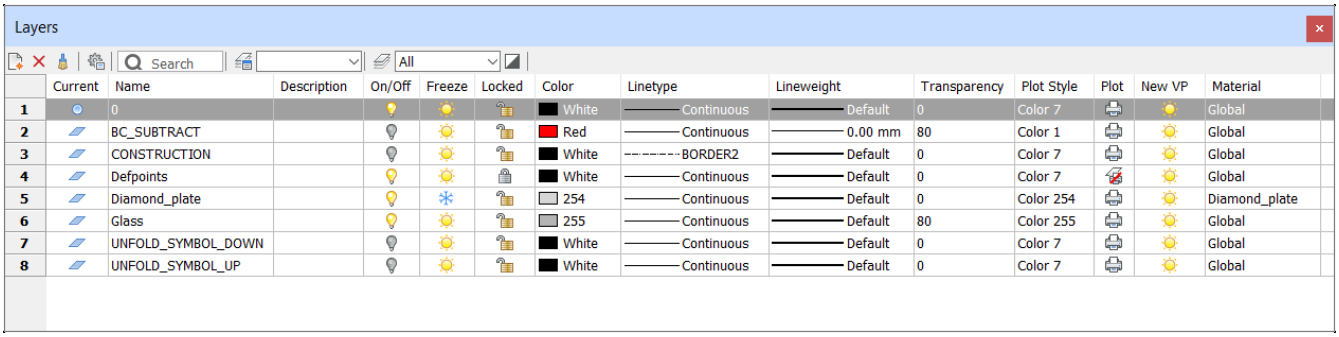
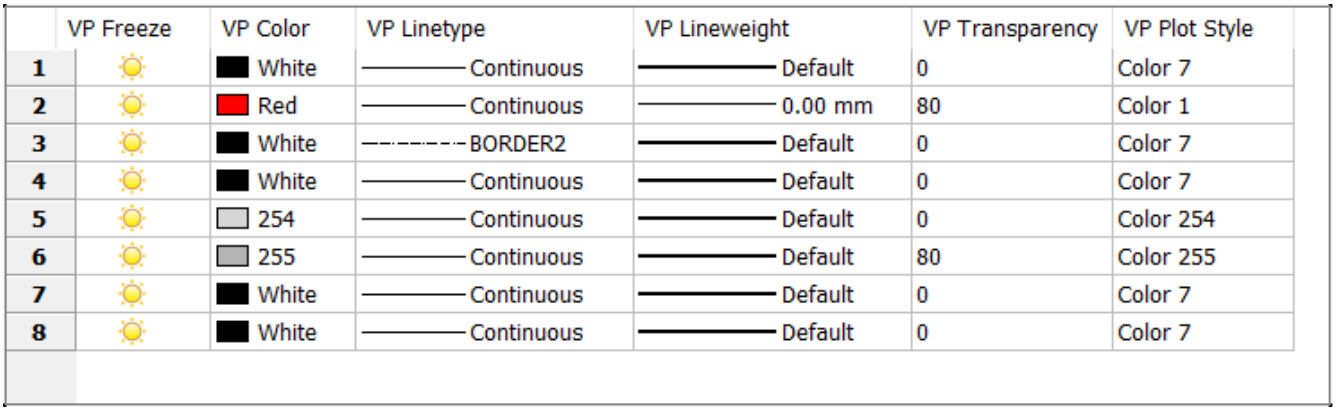


Figure 6.1173



	VP Freeze	VP Color	VP Linetype	VP Lineweight	VP Transparency	VP Plot Style
1		White	Continuous	Default	0	Color 7
2		Red	Continuous	0.00 mm	80	Color 1
3		White	BORDER2	Default	0	Color 7
4		White	Continuous	Default	0	Color 7
5		254	Continuous	Default	0	Color 254
6		255	Continuous	Default	80	Color 255
7		White	Continuous	Default	0	Color 7
8		White	Continuous	Default	0	Color 7

Command Options

Option	Description
New	Creates new layers with the generic name ”NewLayer1”. Thenew layer that is created copies the properties of the currently selected layer.
Delete	Deletes layers, except for the following, whichcannot be deleted: • 0 • Any layer with entities
Purge	Purges layers from the current drawing, except for the following ones, which cannot be purged: • 0 • The current layer • Any layer with entities See the Purge (6.24.57) command.

Settings	Displays the Layers Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ • **Hide xref layers** - when on, hides layers that came from externally-referenced drawings • **Apply layer filter to layer toolba**r - when on, shortens the list of layers displayed, according to the conditions of the filter. When off, displays all layer names. • **Indicate layers in use** - when on, displays icons in the Current column that indicate whether the layer has entities assigned to it; unused layers can be purged: - Layer is not empty - Layer is empty of entities • **Show row numbers** - toggles the display of the number at the start of each row. **Toolbar Controls** toggles the display of controls in the toolbar: • **Search Layers** - toggles the display of the Search field on and off • **Layer States** - toggles the display of the Layer States droplist • **Layer Filters** - toggles the display of the Layer Filters droplist **OK** - applies the changes and exit the dialog box **Cancel** - exits the dialog box without applying the changes
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/sourceLayerSearchField.png})	Filters the names of layers to shorten longer lists of layer names. As you type, the program displays the layer names containing the text. Any layers containing the sequence of characters are listed, no matter where in the layer name the sequence is located.For example, entering ”do” returns layers named DOOR and WINDOW.
Layer States Explorer	Displays the Layer States section of the Drawing Explorer. See the LayerState (6.20.7)command.
Layer States Used	Lists the layer states available in the drawing. Choose one from the droplist.See the LayerState (6.20.7)command.
Layer Explorer	Displays the Layer section of the Drawing Explorer. See the Layer (6.20.2) command.
Filter Used	Displays the current layer filter, if any are defined in the current drawing. Click the down-arrow button to choose a different layer filter The program creates the following layer filters automatically: • **All** - the only layer filter that appears in all drawings, and displays all layers • **Viewport Overrides** - you’ve overridden the properties of a viewport layer • **Xref** - the drawing has externally referenced drawings attached • ’xref name’ - the name of each xref’ed drawing (note that layers from xref’ed drawings are prefixed automatically with the drawing’s name)

 Invert Filter	Inverts the content of current filter, so that the dialog box displays all layers not included in the filter. For example, to see all layers **not** in an xref'ed drawing, choose the **Xref** filter, and then click **Invert Filter**.

Header Options	**Description**
 Current	Specifies the current working layer. Only one layer can be current at a time. When the **Indicate Layers In Use** option is turned on, then these additional icons are displayed: •  - Layer is not empty •  - Layer is empty In paper space layouts or viewports (model space with floating viewports), the following icons indicate that viewport overrides exist: •  - Viewport overrides exist and layer is current •  - Viewport overrides exist and layer is not empty •  - Viewport overrides exist and layer is empty
Layer Name	Specifies the name of the layer. In long lists of layers, you can typea character to jump to the first layer starting with that character, such as 'd' for DOORS. To sort the layer names from A to Z, click the **Layer Name** header; to reverse the order from Z to A, click the header a second time. To rename a layer: 1. Click the name twice, slowly (or right-click the layer name and then choose **Rename**) 2. Enter the new layer name
Description	(optional) Describes the content of the layer. Click the field, and then type.
On/Off	Toggles the selected layers between on and off. To change the status, select one or more layers, and then click a lightbulb icon: •  **On** - displays entities on the layer •  **Off** - hides entities from view; entities are included in the calculation of extents

Freeze	Toggles the selected layers between frozen and thawed.To change the status, select one or more layers, and then click a sun/snowflake icon: • **Thaw** - displays entities on the layer • **Freeze** - hides entities from view; entities are excluded in the calculation of extents, and are not exported with commands such as WBlock
Locked	Toggles the selected layers between locked and unlocked.To change the status, select one or more layers, and then click a padlock icon: • **Unlocked** - allows entities to be edited • **Locked** - prevents entities from being edited; entities are still visible
Color	Specifies the color of the layer. The default color is white (black).To change the color: 1. Choose one or more layers 2. Click a color square to display the Select Color dialog box; see Color (6.9.33) command 3. Choose a color, and then click **OK**
Linetype	Specifies the linetype of selected layers. The default linetype is ”Continuous.” To change the linetype: 1. Choose one or more layers 2. Click a linetype sample 3. Choose one from the droplist (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ To load additional linetype definitions: 1. Click the **Load** option. 2. In the Load Linetypes dialog box, choose a linetype, and then click **OK**. See Linetype (6.20.33) command.
Lineweight	Specifies the linetype of selected layers. Default is ”Default.” To change the lineweight: 1. Choose one or more layers 2. Click the lineweight sample 3. From the droplist, choose another lineweight (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ See theLWeight (6.20.41) command.

Transparency	Sets the transparency of entities on the selected layer(s) between 0 (fully opaque) and 90 (fully transparent); the default is 0.To change the transparency: 1. Choose one or more layers 2. Click the transparency number 3. From the droplist, choose another number The 2DWireframe visual style ignores this setting; instead, use the TRANSPARENCYDISPLAYvariable to control the level of transparency.
Plot Style	Specifies the plot style of selected layers, if plot styles are enabled in the current drawing. The default is ”Color_7.” To change the plot style” 1. Choose one or more layers 2. Click a plot style name 3. From the droplist, choose another style name By default, plot styles are not enabled in drawings. To add plot styles to the drawing, see thePlotStyle (6.24.29) command.
Plot	Toggles selected layers between being printed and not printed.To change the status, select one or more layers, and then click the printer icon: • **Print**- prints the layer • **No Print**- does not print the layer Some layers created by the program also do not print by default, such as DefPoints.
New VP	Specifies whether the layer is visible (thawed) or invisible (frozen) when you make a new viewport (VP) in a layout tab.To change the status, select one or more layers, and then click a sun/snowflake icon: • **Thaw** - displays entities on the layer in the new viewport • **Freeze** - initially hides entities from view when the viewport is created See the MView (6.21.37) command.
Material	Specifies the material of selected layers. The default is material is name ”Global.” To change the material assigned to one or more layers: 1. Select one or more layers. 2. Click the **Material** field. 3. Click again to display the droplist, containing the materials availablein the drawing. 4. Select a material in the list. When you need to add more materials to the drawing, see the Materials (6.21.11) command.

Viewport Headers

The following headers appear when you switch from model tab to a layout tab:

Header Option	Description
VP Freeze	Freezes the layer in the current layout; overrides the freeze/thaw setting in model space
VP Color	Specifies the color of entities on this layer in the current layout; overrides the color setting in model space
VP Linetype	Specifies the linetype for entities in the current layout; overrides the linetype setting in model space
VP Lineweight	Specifies the lightweight in the current layout; overrides the lightweight setting in model space
VP Transparency	Specifies the transparency of entities on this layer in the current layout; overrides the transparency setting in model space
VP Plotstyle	Specifies the plot style name in the current layout; overrides the plot style setting in model space

Procedures

- To open or close a panel (1.12.7)
- To dock a panel (1.12.7)
- To dock a panel with respect to another docked panel (1.12.7)
- To stack panels in a tabbed panel (1.12.7)

Related Commands

-Layer (6.20.3) - creates layers and changes their properties in the command bar. Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. LayersPanelClose (6.20.5) - closes the Layers panel. LayerState (6.20.7) - saves and restores layer states. LayMCur (6.20.11) - changes the working layer to that of a selected entity LayFrz (6.20.8) - freezes layers associated with selected entities. LayThw (6.20.16) - thaws all frozen layers. LayIso (6.20.9) - isolates layers associated with selected entities. LayLck (6.20.10) - locks layers associated with selected entities. LayUlk (6.20.17) - unlocks layers associated with selected entities. LayOff (6.20.12) - turns off layers associated with selected entities. LayOn (6.20.13) - turns on all layers that have been turned off. Materials (6.21.11) - Creates materials and edits their properties through the Drawing Explorer. -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line. VpLayer (6.31.18) - changes the properties of layers in the current paperspace viewport. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.1174

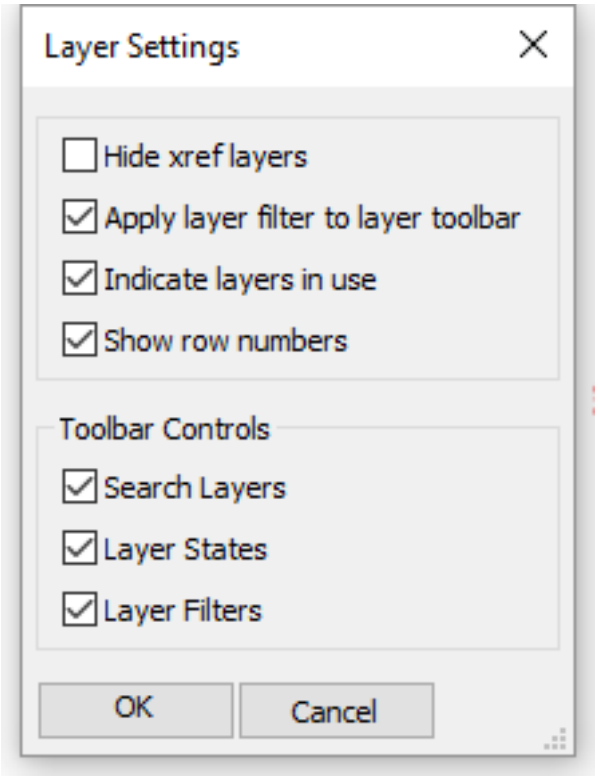


Figure 6.1175



Figure 6.1176

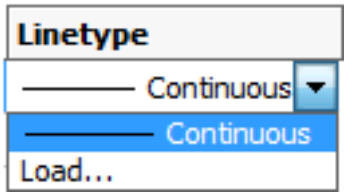


Figure 6.1177

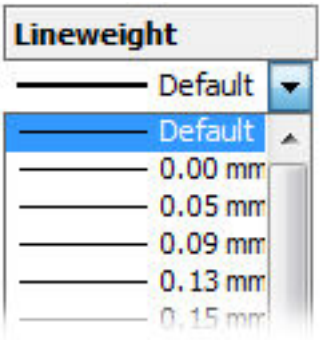


Figure 6.1178



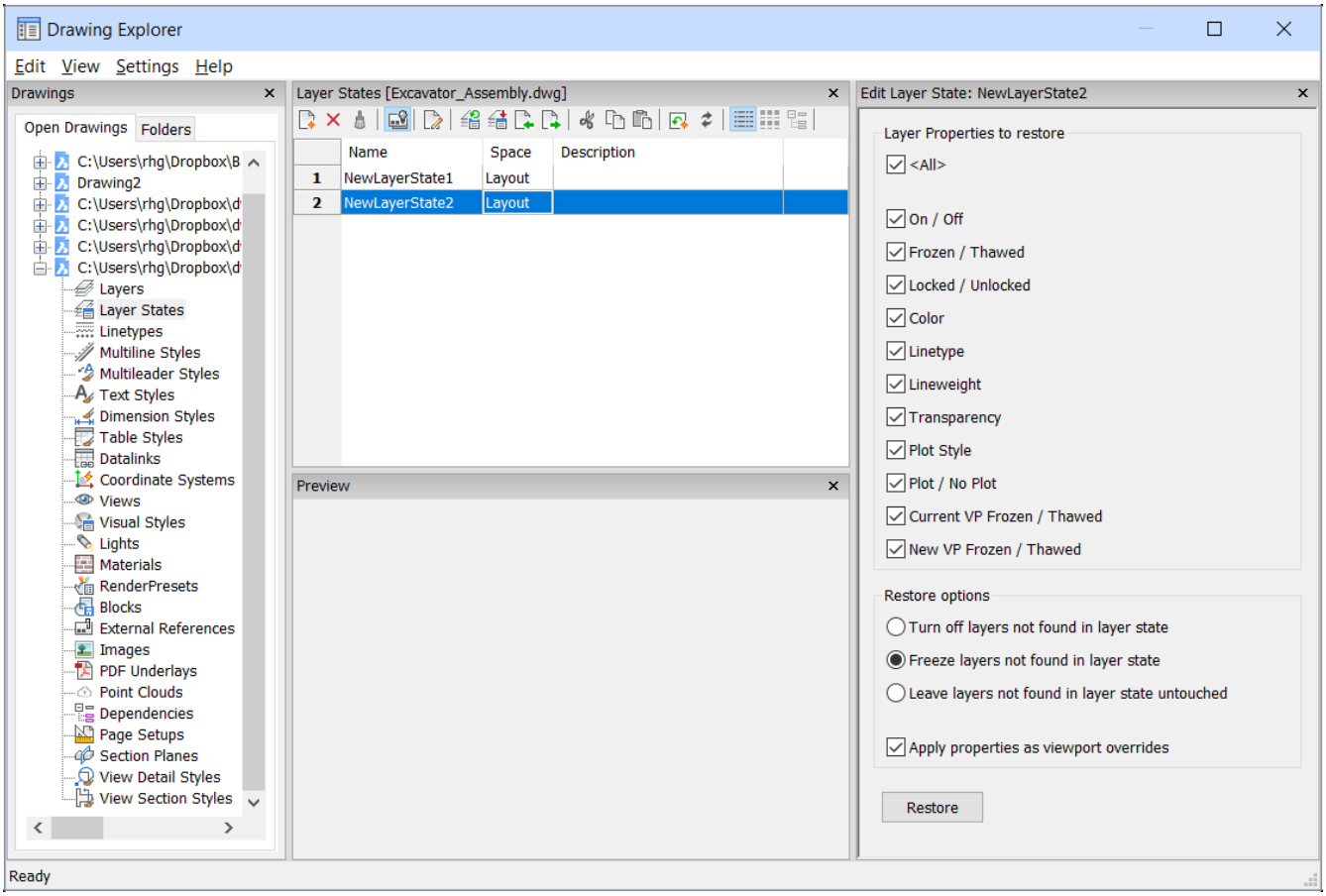
6.20.7 LayerState

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and edits layer states, which save the current state of the properties of layers, through the Layer States section of the Drawing Explorer.

Accessing the Command

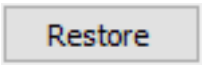
 command: **layerstate** alias: **las** menu bar: **Settings | Layers | Layer States... Tools | Drawing Explorer | Layer States...** ribbon:**Home | Settings | Drawing Explorer | Layer States Home | Layers | Layers | Layer States** toolbar: **Layers** | : **LayerState** Displays the Layer States section of the Drawing Explorer: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/layestate-dlg.png}) Create or choose a layer state, and then click **X**. Press **F1** to access help.

Figure 6.1179



Command Options

Toolbar Option	Description
New	Creates a new layer state with the default name of NewLayerState1. When a layer state is created, the program saves the current state of all layer properties, such as which layers are frozen, turned off, or locked.
Delete	Erases selected layer states. The program erases the layer state without warning.
Hide Xref Symbols On/Off	Toggles the display of layers from externally-referenced drawings (the word Symbols should be 'Drawings').
Edit Layer State	Displays the Edit Layer State dialog box to change the status of the layer state: • Adds layers from the layer state • Adds layers to the layer state • Changes which layer is current • Changes all properties of each layer But does not change the names of layers.
Restore	Restores the selected layer state. The properties of layer change their state after you close the Drawing Explorer dialog box.
Overwrite	Overrides the settings of the selected layer state with new settings changed by the Edit Layer State panel.

 Import	Imports layer states from a LAS file: 1. Displays the Import Layer States dialog box.Choose an LAS file. 2. Click **Open** to load the layer state into the current drawing
 Export	Exports layer states to an LAS file: 1. Displays the Export Layer States dialog box. Give a name tothe LAS file. 2. Click **Save**. The layer state file can be imported into other drawings or send to clients.
Name	Names the layer state.Click to rename the state.
Space	Reports which space the layer state was saved: • **Model** - saved in model tab. • **Layout** - saved in layout tab. You must start the LayerState command in either model or a layout tab.
Description	Optional description of the layer state.
Edit Layer State Options <All>	**Description** Selects or unselects all properties. See the Layer (6.20.2) command for the meaning of the properties.
Restore Options	Determines what happens with layers not found in the layer state, such as those added after the layer state was created.The selected option applies to all layer states • Turn off • Freeze • Leave untouched
Apply properties as viewport overrides	Applies the layer state as viewport overrides (VP layer properties (6.20.2)) to the current layout viewport. This option is available only on a layout within a layout viewport.
	Restores the selected layer state.

Procedures

- To save the current layer state (1.16.9)
- To edit a layer state (1.16.9)
- To restore a layer state (1.16.9)
- To copy layer states to another drawing (1.16.9)
- To export a layer state (1.16.9)
- To import a layer state (1.16.9)

Related Commands

Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. -Layer (6.20.3) - creates layers and changes their properties in the command bar. LayFrz (6.20.8) - freezes layers associated with selected entities. LayThw (6.20.16) - thaws all frozen layers. LayIso (6.20.9) - isolates layers associated with selected entities. LayLck (6.20.10) - locks layers associated with selected entities. LayUlk (6.20.17) - unlocks layers associated with selected entities. LayOff (6.20.12) - turns off layers associated with selected entities. LayOn (6.20.13) - turns on all layers that have been turned off. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.1180



6.20.8 LayFrz

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Freezes the layers associated with entities that you select in the drawing (short for "layer freeze"). **Note:** Use the Layer (6.20.2) command to thaw specific layers that were frozen with this command, or else use the LayThw (6.20.16) command to thaw all layers in the drawing.

Accessing the Command



command bar: **layfrz** menu bar: **Settings | Layers | Freeze Layer** ribbon: **Home | Layers | Freeze** toolbar: **Layers | Freeze** : **LayFrz Prompts you in the command bar:** Current setting: Viewports=Freeze, Block nesting level=Block Select an entity on the layer to be frozen or [Settings]: **(Pick an entity, or type S)** This layer has been frozen: LAYERNAME. Select an entity on the layer to be frozen or [Settings]: **(Pick another entity, or else press Enter to exit the command)** The current layer cannot be frozen; BricsCAD warns you: Cannot freeze layer. It is the current layer.

Command Options

Option	Description
Select an entity on the layer to be frozen	Selects a single entity whose layer should be frozen. Theprompt repeats itself until you press **Enter**so that you can freeze more than one layer.
Settings	Specifies the tricky case of how layers holding viewports and blocks (and xrefs) should be frozen; prompts you: **Enter setting type for [Viewports/Block selection]** - **(enter an option)**: • **Viewports** - specifies how viewports should be frozen; prompts you: **In paper space viewport use [Freeze/Vpfreeze]** - **(type F or V.) Freeze** - freezes layers in all viewports (default) **Vpfreeze** - freezes layers in the current viewport only • **Block selection** - specifies how blocks and xrefs should be frozen; prompts you: **Enter Block Selection nesting level [Block/Entity/None]** - **(type B, E, or N.) Block** - freezes all layers associated with the block, includes of nested blocks **Entity** - freezes all layers associated with entities in the block or xref(entities that make up blocks might be on different layers) **None** - freezes the layer of the block, and ignores other layers associated with entities and nested block

Related Commands

Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. -Layer (6.20.3) - creates layers and changes their properties in the command bar. LayerState (6.20.7) - saves and restores layer states. LayThw (6.20.16) - thaws all frozen layers. LayIso (6.20.9) - isolates layers associated with selected entities. LayLck (6.20.10) - locks layers associated with selected entities. LayUlk (6.20.17) - unlocks layers associated with selected entities. LayOff (6.20.12) - turns off layers associated with selected entities. LayOn (6.20.13) - turns on all layers that have been turned off.

Figure 6.1181



6.20.9 LayIso

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Isolates layers associated with entities that you select in the drawing; all other layers are either locked or turned off (hidden); short for "layer isolate." **Note:** Use the LayUnIso (6.20.18)command to unlock or unhide isolated layers.

Accessing the Command



command bar: **layiso** menu bar:**Settings | Layers | Isolate Layer** ribbon:**Home | Layers| isolate** toolbar: **Layers | Isolate** : **LayIso Prompts you in the command bar:** Current setting: Lock layers Select entities on the layer(s) to be isolated or [Settings]: **(Pick one or more entities, or type S)** Number of found objects: 1, number of selected objects: 1 Select entities on the layer(s) to be isolated or [Settings]**: (Choose more entities, or press Enter to exit the command)** Number of isolated layers: xx. This layer is current: LAYER_NAME.

Command Options

Option	Description
Select entities on the layer(s) to be isolated	Selects one or more entitieswhose layers should be isolated; theprompt repeats itself until you press **Enter** so that you can isolate more than one layer. All other layers are locked or turned off.
Settings	Specifies how layers and viewports should be isolated; prompts you: **Enter setting for layers not isolated [Off/Lock]** - **(enter an option)**: • **Off** - all other layers are turned off (made invisible) • **Lock** - all other layers are locked (visible and cannot be edited)

Related Commands

Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. -Layer (6.20.3) - creates layers and changes their properties in the command bar. LayerState (6.20.7) - saves and restores layer states. LayFrz (6.20.8) - freezes layers associated with selected entities. LayThw (6.20.16) - thaws all frozen layers. LayLck (6.20.10) - locks layers associated with selected entities. LayUlk (6.20.17) - unlocks layers associated with selected entities. LayOff (6.20.12) - turns off layers associated with selected entities. LayOn (6.20.13) - turns on all layers that have been turned off. LayUnIso (6.20.18)- unisolates locked or turned off layers.

Figure 6.1182



6.20.10 LayLck

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Locks the layer associated with an entity selected in the drawing; locked layers can be seen but not edited (short for "layer lock"). **Note:** Use the Layer (6.20.2) command to unlock specific layers that were locked with this command, or else use the LayUnlk (6.20.17)command to unlock all layers in the drawing at once.

Accessing the Command


command bar: **laylck** menu bar: **Settings | Layers | Lock Layer** ribbon: **Home | Layers | Lock** toolbar: **Layers | Lock** : **LayLck Prompts you in the command bar:** Select an entity on the layer to be locked: **(Choose an entity)** This layer has been locked: <LayerName> Locked layers can be shown faded. The LAYERLOCKFADECTL (1.13.11)variable sets the fading level between 0 (no fading) and 90 (very faded).

Command Options

Option	Description
Select an entity on the layer to be locked	Specifies the layer to be locked by the entity selected.

Related Commands

LayUlk (6.20.17) - unlocks layers associated with selected entities. Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. -Layer (6.20.3) - creates layers and changes their properties in the command bar. LayerState (6.20.7) - saves and restores layer states. LayFrz (6.20.8) - freezes layers associated with selected entities. LayThw (6.20.16) - thaws all frozen layers. LayIso (6.20.9) - isolates layers associated with selected entities. LayOff (6.20.12) - turns off layers associated with selected entities. LayOn (6.20.13) - turns on all layers that have been turned off. LayUnlk (6.20.17) - unlocks locked layers.

Figure 6.1183



6.20.11 LayMCur

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Changes the working layer to that of the entity selected (short for "layer make current").

Accessing the Command

 Command bar: **laymcur** alias: **setlayer** menu bar: **Settings | Layers | Set Layer by Entity** ribbon: **Home | Layers |Make Object's Layer Current** Toolbar: **Entity Properties }Make Object's Layer Current** : **LayMCur Prompts you in the command bar:** Select entity to set the current layer: **(Choose one entity)**

Command Options

Option	Description
Select entity to set the current layer	Choose one entity whose layer will be made the current layer.

Related Commands

Layer (6.20.2) - creates layers and changes their properties through a dialog box. -Layer (6.20.3) - creates layers and changes their properties through the command bar. LayerState (6.20.7) - saves and restores layer states.

Figure 6.1184



6.20.12 LayOff

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Turns off layers associated with entities selected in the drawing; off layers cannot be seen (short for "layer off"). Note:Use theLayer (6.20.2)command to turn on specific layers that were turned off with this command, or else use theLayOn (6.20.13)command to turn on all layers in the drawing at once.

Accessing the Command



command bar: **layoff** menu bar: **Settings | Layers | Layer Off** ribbon: **Home | Layers | Layer Off** toolbar: **Layers | Layer Off** : **LayOff Prompts you in the command bar:** Current setting: Viewports=Off, Block nesting level=Block Select an entity on the layer to be turned off or [Settings]: **(Choose an entity, or ekes type S)** This layer has been turned off: <LayerName>. Select an entity on the layer to be turned off or [Settings]: **(Choose another entity, or else press Enter to exit the command)**

Command Options

Option	Description
Select an entity on the layer to be turned off	Selects an entity whose layer is to be turned off.This prompt repeats itself until you press **Enter**so that you can turn off more than one layer.
Settings	Specifies how viewports, blocks, and xrefs should be turned off; prompts you: **Enter setting type for [Viewports/Block selection]** - **(enter an option)**: • **Viewports** - specifies how viewports should be turned off; prompts you: **In paper space viewport use [Freeze/Vpfreeze]** - **(type F or V.) Freeze** - turns off layers in all viewports (default) **Vpfreeze** - turns off layers in the current viewport only • **Block selection** - specifies how blocks and xrefs should be turns off; prompts you: **Enter Block Selection nesting level [Block/Entity/None]** - **(type B, E, or N.) Block** - turns off all layers associated with the block, includes of nested blocks **Entity** - turns off all layers associated with entities in the block or xref(entities that make up blocks might be on different layers) **None** - turns off the layer of the block, and ignores other layers associated with entities and nested block

Related Commands

LayOn (6.20.13) - turns on all layers that have been turned off. Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. -Layer (6.20.3) - creates layers and changes their properties in the command bar. LayerState (6.20.7) - saves and restores layer states. LayFrz (6.20.8) - freezes layers associated with selected entities. LayThw (6.20.16) - thaws all frozen layers. LayIso (6.20.9) - isolates layers associated with selected entities. LayLck (6.20.10) - locks layers associated with selected entities. LayUlk (6.20.17) - unlocks layers associated with selected entities.

Figure 6.1185



6.20.13 LayOn

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Turns on all layers that were turned off in the drawing, except for frozen layer (short for ”layer off”).

Accessing the Command

 command bar: **layon** menu bar: **Settings | Layers | Layer On** ribbon: **Home | Layers | Layer On** toolbar: **Layers | Layer On** : **LayOn Prompts you in the command bar:** All layers have been turned on.

Command Options

No options.

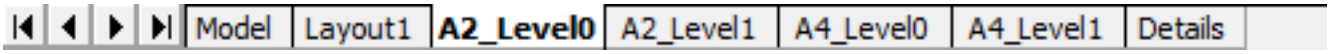
Related Commands

LayOff (6.20.12) - turns off layers associated with selected entities. Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. -Layer (6.20.3) - creates layers and changes their properties in the command bar. LayerState (6.20.7) - saves and restores layer states. LayFrz (6.20.8) - freezes layers associated with selected entities. LayThw (6.20.16) - thaws all frozen layers. LayIso (6.20.9) - isolates layers associated with selected entities. LayLck (6.20.10) - locks layers associated with selected entities. LayUlk (6.20.17) - unlocks layers associated with selected entities.

Figure 6.1186



Figure 6.1187



6.20.14 Layout

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates, copies, renames, and deletes layouts. (\vref {fig:C:/Dev/Task-2448/images/staticbric

Accessing the Command


command bar: **layout** menu bar: **Insert | Layout** ribbon:**Home | Settings Layouts** toolbar: **Insert |**  **Layout** |  : **Layout Prompts you in the command bar:** Enter layout option [? to list/Set/New/Copy/Delete/Rename/SAve/Template/n <Set>: **(Enter an option)**

Command Options

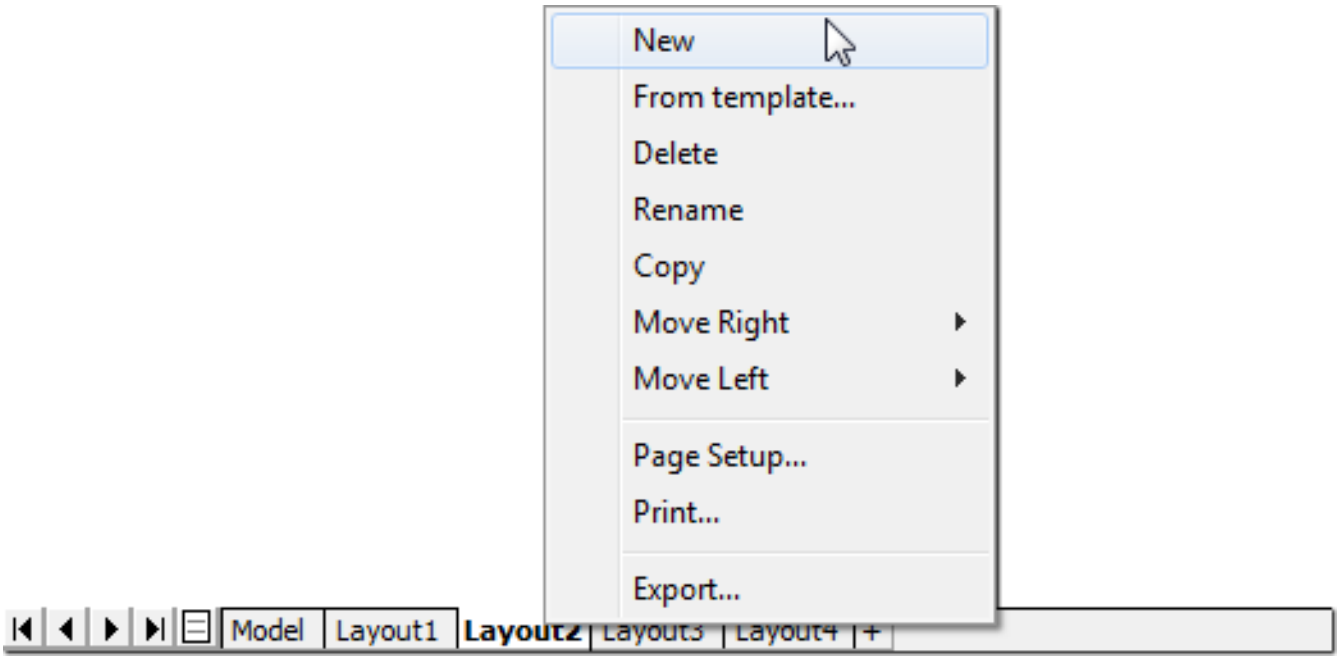
Option	Description
? to list	Lists the names of layouts already defined in this drawing: **Layout name:: Layout1 Block name:: *Paper_Space0 Layout name:: Layout2 Block name:: *Paper_Space1** If necessary, press **F2** to switch to the Prompt History window.
Set	Switches to the specified layout; prompts you: **Enter a layout name - (enter a name of a layout; the program immediatelyswitches to the layout.)**
New	Creates new layouts; prompts you: **Enter new layout name <layoutn>- (specify the name of the new layout, or else press Enter to accept the offered name.)** The program adds thelayout is added as a tab. The default name is "Layout**n**," where **n** is incremented with each new layout.
Copy	Creates a new layout by making a copy of an existing layout; prompts you: 1. **Enter the name of the layout to copy: <current> - (enter a name of an existing layout, or else press Enter to use the current layout name.)** 2. **Enter new layout name - (specify the name of the new layout; the program adds the copied layout as a new tab.)**

Delete	Erases layouts from the drawing; prompts you: 1. **Enter a layout name <current>** - **(specify the name of the layout to be removed,or else press Enter to use the current layout name.)** 2. The program displays a warning dialog box.Click **OK** to confirm the deletion or **Cancel** to preserve the tab The Model tab cannot be deleted.
Rename	Renames layouts; prompts you: 1. **Enter a layout name <current>** - **(enter a name of an existing layout,or else press Enter to use the current layout name.)** 2. **Enter new layout name** - **(specify the name of the new layout.)**
SAve	Saves layouts as DWG files, prompts you: 1. **Enter a layout name <current>** - **(enter a name of a layout,or else press Enter to use the current layout name.)** The program displays the Create Drawing File dialog box. 2. Enter a name for the drawing. 3. Choose whether to save the file in DWG or DXF format. 4. Click **Save**. The DWG file contains all layouts, including Model, but no drawing info. To use this file in another drawing, use the Import option.
Template	Imports layouts from DWG drawing files; displays the Select Template From File dialog box: 1. Choose a DWG or DXF file created by the SAve option. 2. Click **Open**. The program displays the Insert Layout(s) dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/template-dlg.png}) 3. Choose one or more layout names. 4. Click **OK**. If the layout name already exists, then the program appends its name. For example, whenthe imported layout is named ”Layout 2,” the program names it ”Layout3-Layout2.”
neXt	Displays the next layout.
Previous	Displays the previous layout.

Layout Tabs

Right-click a tab for the following shortcut menu, which mimics the Layout command: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/layout-tab-mnu.png}) Use the arrow buttons (left end of the tabs) to move along the tab bar, when the drawing contains too many layout tabs to see them all. **Print Setup** displays the Print Setup dialog box for the current layout; see the CPageSetup (6.9.55) command. **Print** displays the Print dialog box, from which layouts are printed; see the Print (6.24.46) command.

Figure 6.1188

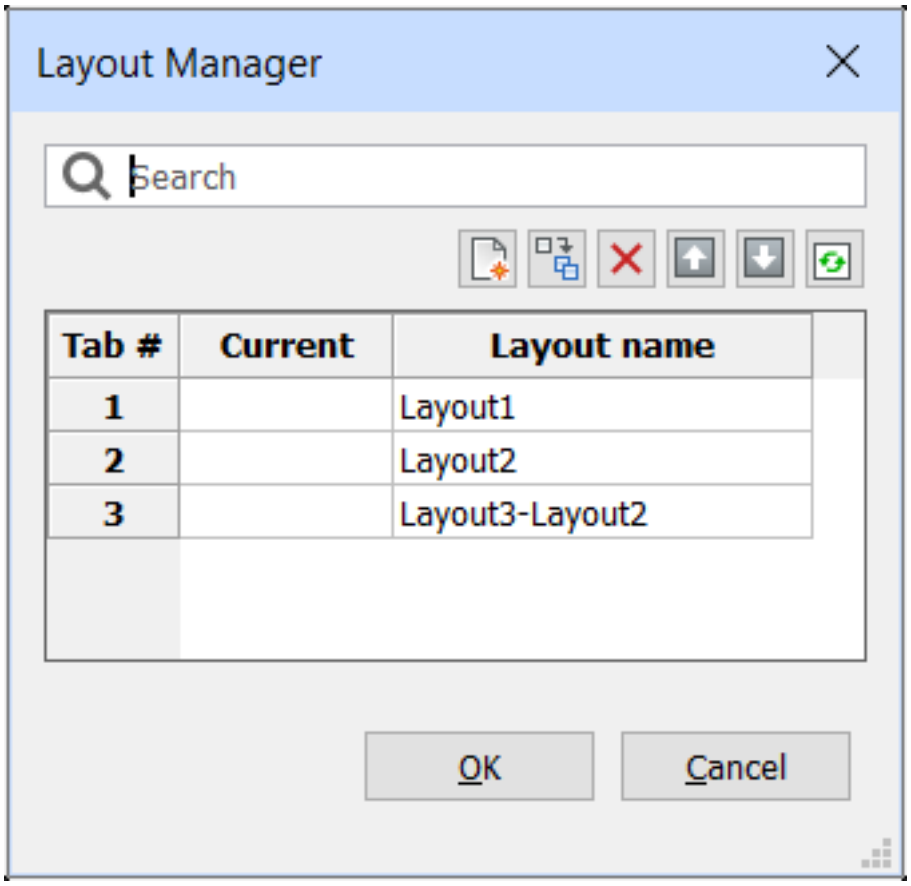


Layout Manager

Click the **Layout Manager** button on the layout tab to view the dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsys dlg.png})

Option	Description
Search	Searches for a layout name, useful when drawings contain many layouts
New	Creates a new layout
Copy Selected Layout(s)	Makes a copy of selected layouts, and adds them to the end of the list
Delete	Deletes the selected layouts
Up	Move the selected layouts up the list
Down	Move the selected layouts down the list
Clear Selection	Removes highlighting from layout names, thereby unselecting them
Current	Click in the Current column to make the layout current
Name	Double-click a layout name to exit the dialog box and make the layout current

Figure 6.1189



Status Bar Menu

Right-click TILE or LAYOUT on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/LayoutShortcutMenu.png})

Shortcut Menu Option	Description
Paper Space	(Available only in a layout tab) Switches the layout to paper space
Model Space	(Available only in a layout tab) Switches the layout to model space
Model	Switches the drawing editor to Model tab.
Layout1	Switches the drawing editor to Layout1.
Layout2	Switches the drawing editor to Layout2. When the drawing contains other layouts, they are also listed in the shortcut menu.

Figure 6.1190

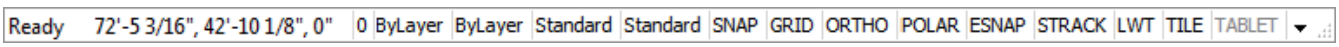
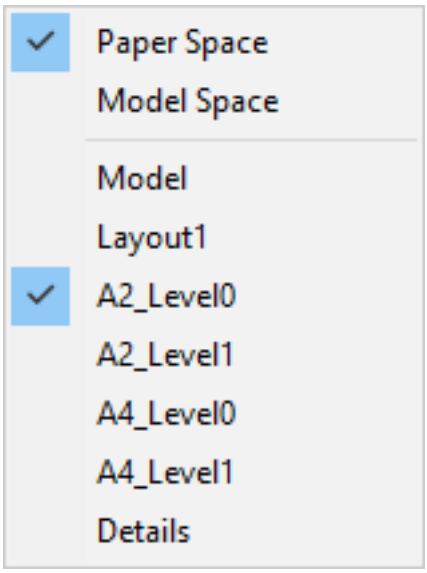


Figure 6.1191



Procedures

- Creating a new layout (1.16.15)
- Copying layouts (1.16.15)
- Importing layouts (1.16.15)
- Renaming layouts (1.16.15)
- Arranging the layout tabs (1.16.15)
- Deleting a layout (1.16.15)

Related Commands

LayoutManager (6.20.15) - manages layouts through a dialog box. MSpace (6.21.34) - switches to model space. PsetupIn (6.24.52) - imports page setup definitions from another drawing through a dialog box. -PsetupIn (6.24.53) - imports page setup definitions from another drawing through the command bar. PSpace (6.24.54) - switches to paper space. MView (6.21.37) - creates multiple viewports in paper space. VPorts (6.31.20) - creates multiple viewports in model space.

Figure 6.1192

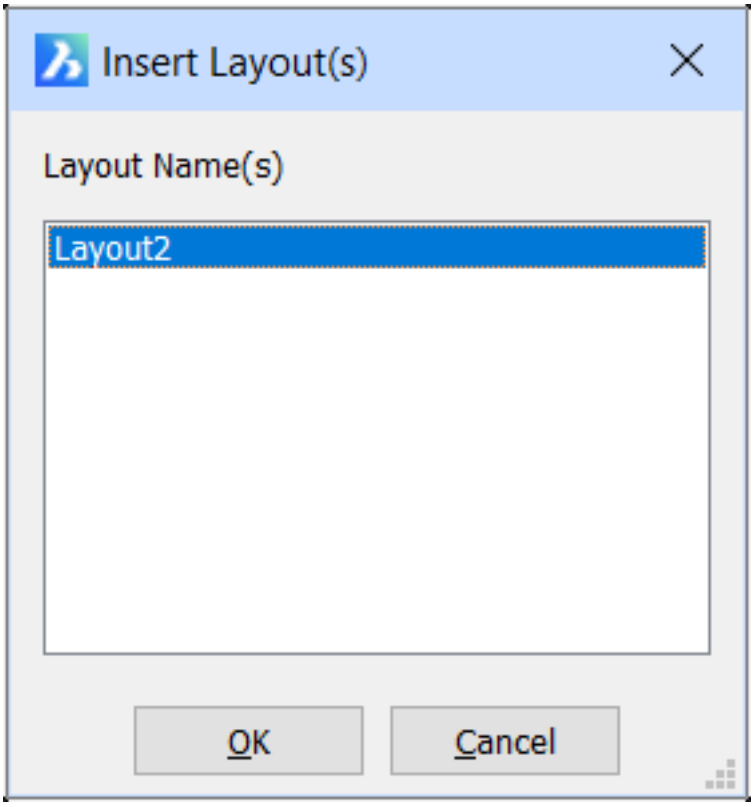


Figure 6.1193



6.20.15 LayoutManager

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Manages layouts through a dialog box.

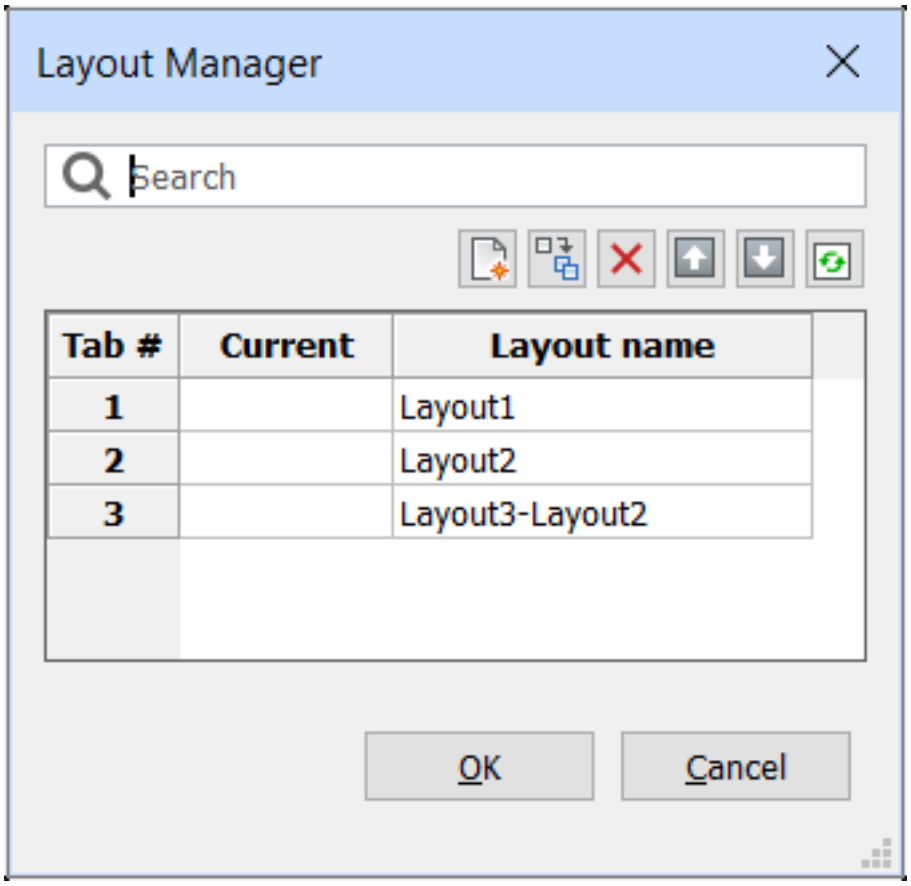
Accessing the Command

command bar: **layoutmanager** layout tab: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sc
: **LayoutManager** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L
dlg.png})

Figure 6.1194



Figure 6.1195



Dialog Box Options

Option	Description
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/LayerMan agerSearchField.png})	Searches for a layout name, useful when drawings contain many layouts
New	Creates a new layout.Type a name in the **Layout name** field to replace the default Layout**n** name.
Copy Selected Layout(s)	Makes a copy of selected layouts, and adds them to the end of the list
Delete	Deletes the currently selected layout(s)
Up	Moves the currently selected layout(s) up in the list, and to the left in the layout tabs
Down	Moves the currently selected layout(s) down in the list, and to the right in the layout tabs.
	Removes highlighting from layout names, thereby unselecting them
Current	Click in the Current column to make the layout current

Related Commands

Layout (6.20.14) - creates, copies, renames, and deletes layouts.

Figure 6.1196



6.20.16 LayThw

Thaws all frozen layers, but does not change layers that are turned off (short for "layer thaw").

Accessing the Command


command bar: **laythw** menu bar: **Settings | Layers** ribbon: **Home | Layers| Thaw All Layers** toolbar: **Layers | Thaw All Layers** : **LayThw Prompts you in the command bar:** All layers have been thawed.

Command Options

No options.

Related Commands

LayFrz (6.20.8) - freezes layers associated with selected entities. Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. -Layer (6.20.3) - creates layers and changes their properties in the command bar. LayerState (6.20.7) - saves and restores layer states. LayIso (6.20.9) - isolates layers associated with selected entities. LayLck (6.20.10) - locks layers associated with selected entities. LayUlk (6.20.17) - unlocks layers associated with selected entities. LayOff (6.20.12) - turns off layers associated with selected entities. LayOn (6.20.13) - turns on all layers that have been turned off.

Figure 6.1197



6.20.17 LayUlk

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Unlocks the layer associated with an entity that you select in the drawing (short for "layer unlock").

Accessing the Command


command bar: **layulk** menu bar: **Settings | Layers | Unlock Layer** ribbon: **Home | Layers | Unlock Layer** toolbar: **Layers | Unlock Layer** : **LayUlk Prompts you in the command bar:** Select an entity on the layer to be unlocked: **(Choose an entity)** This layer has been unlocked: <LayerName>

Command Options

Option	Description
Select an entity on the layer to be unlocked	Selects an entity whose layer should be unlocked.

Related Commands

LayLck (6.20.10) - locks layers associated with selected entities. Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. -Layer (6.20.3) - creates layers and changes their properties in the command bar. LayerState (6.20.7) - saves and restores layer states. LayFrz (6.20.8) - freezes layers associated with selected entities. LayThw (6.20.16) - thaws all frozen layers. LayIso (6.20.9) - isolates layers associated with selected entities. LayOff (6.20.12) - turns off layers associated with selected entities. LayOn (6.20.13) - turns on all layers that have been turned off.

Figure 6.1198



6.20.18 LayUnIso

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Restores layers isolated (either locked or turned off) by the LayIso command (short for ”layer unisolate”).

Accessing the Command



command bar: **layuniso** menu bar: **Settings | Layers | Unisolate Layer** ribbon: **Home | Layers | Unisolate Layer** toolbar: **Layers | Unisolate Layer** : **LayUnIso Prompts you in the command bar:** Layers isolated by LAYISO command have been restored.

Command Options

No options.

Related Commands

LayOff (6.20.12) - turns off layers associated with selected entities. Layer (6.20.2) - creates layers and changes their properties in the Drawing Explorer. -Layer (6.20.3) - creates layers and changes their properties in the command bar. LayerState (6.20.7) - saves and restores layer states. LayFrz (6.20.8) - freezes layers associated with selected entities. LayThw (6.20.16) - thaws all frozen layers. LayIso (6.20.9) - isolates layers associated with selected entities. LayLck (6.20.10) - locks layers associated with selected entities. LayUlk (6.20.17) - unlocks layers associated with selected entities.

Figure 6.1199



6.20.19 LConnect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Creates an L-connection between solids.

Accessing the Command


command bar: **lconnect** menu bar: **BIM | BimConnect | Lconnect** toolbar: **BIM Connect** Quad Tab: **Model** Ribbon Tab | Panel: **Model | Modify 3D - Connect** : **LConnect Prompts you in the command bar:** Select solids or faces* to connect: **(Select the first solid or solid face.)** Select solids or faces* to connect: **(Select the second solid or solid face.)** Selectbase solid [Switch] <Accept>: **(Press Enter to create the connection or choose an option.)** * Whensolids are selected the larger part of the solid(s) is kept; otherwise select the faces to be connected. L-Connecting walls (select solids or faces)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Walls1Start.png}) Before	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Walls1Result.png}) Result

L-Connecting roof slabs(select solids or faces)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/RoofSlabsStart.png}) Before	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/RoofSlabsResult.png}) Result

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/ColumnStart.png}) Before	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/ColumnResult.png}) Result

L-Connecting solids (select faces)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Solids1Start.png}) Before	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Solids1Result.png}) Result

Command Options

Option	Description
Switch	Allows to choose between two butted L-connections: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Parallel1.png})or (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Parallel2.png}) or a mitered L-connection: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Bisector.png}) Prompts you: **Select base solid [Switch] <Accept>: (type s then press Enter or tap the Ctrl-key. The connection between the solids changes. Press Enter to accept the selected connection.)** If the HOTKEYASSISTANT = ON, the the Hotkey Assistant indicates the currently selected L-connection type. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/HotKey

Procedures

- Creating L-Connections (2.4.4)

Figure 6.1200

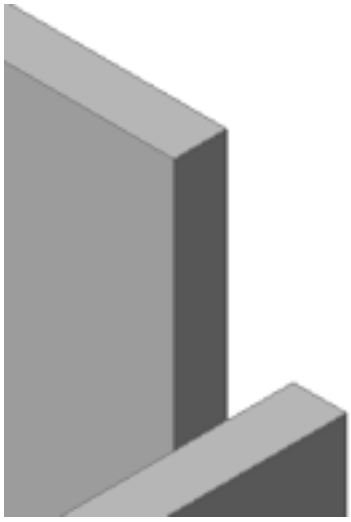
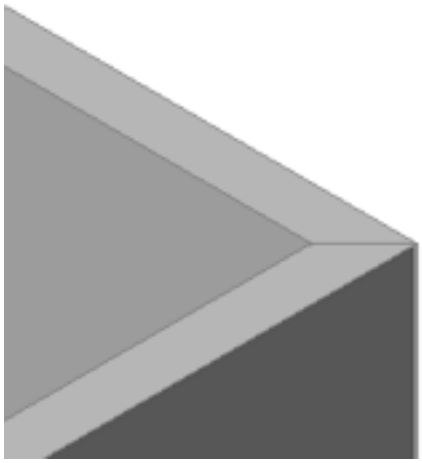


Figure 6.1201



Related Commands

TConnect (6.29.10) - connects solid faces to other solids. BimDrag (6.7.8) - moves 3D solids perpendicular to a selected face by dragging that face. Optionally preserves connections to other solids. BimFlowConnect (6.7.11) - creates a connection between flow segments, such as pipes or HVAC ducts. BimStructuralConnect (6.7.30) - connects structural profile solids such as beams and columns.

Figure 6.1202

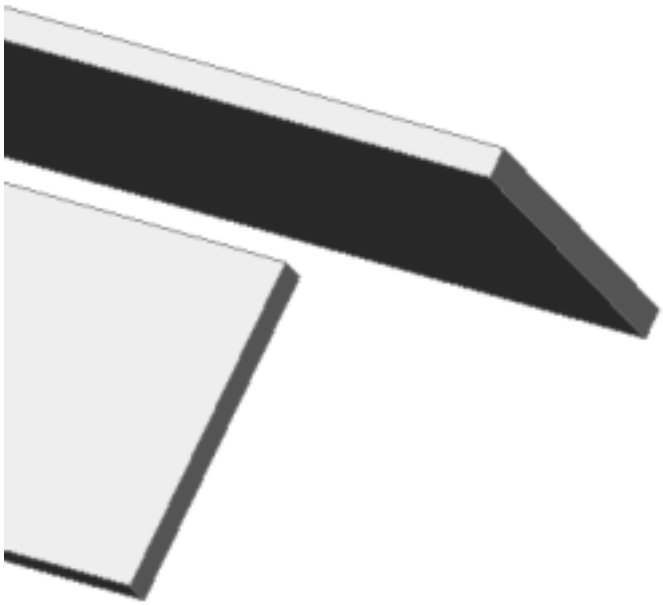


Figure 6.1203

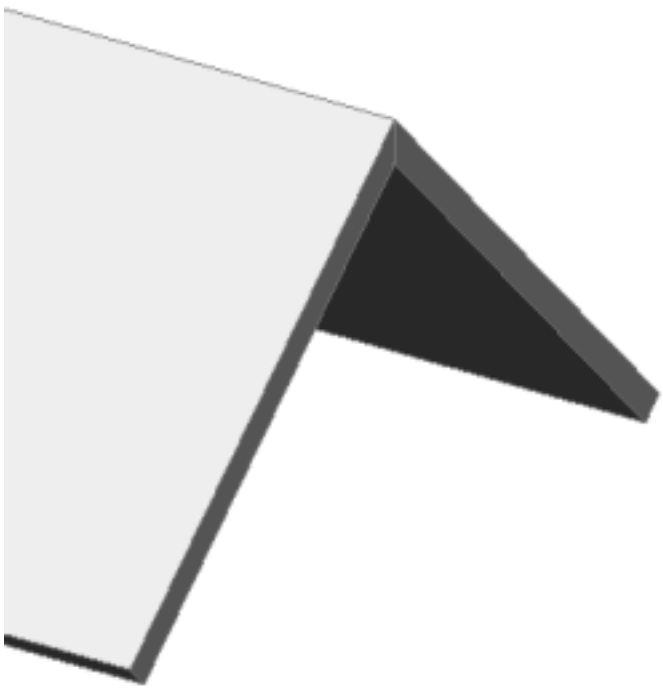


Figure 6.1204

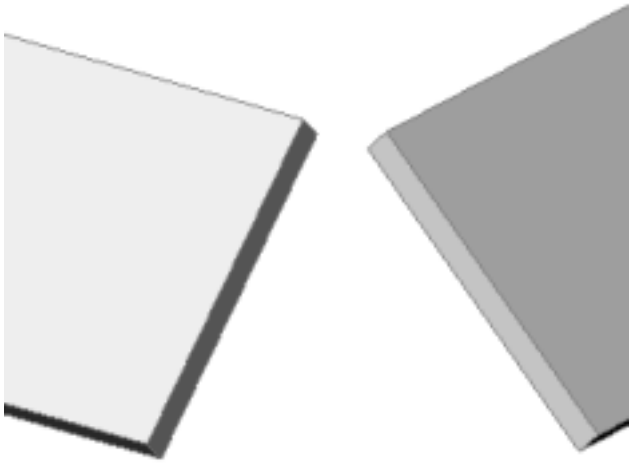


Figure 6.1205

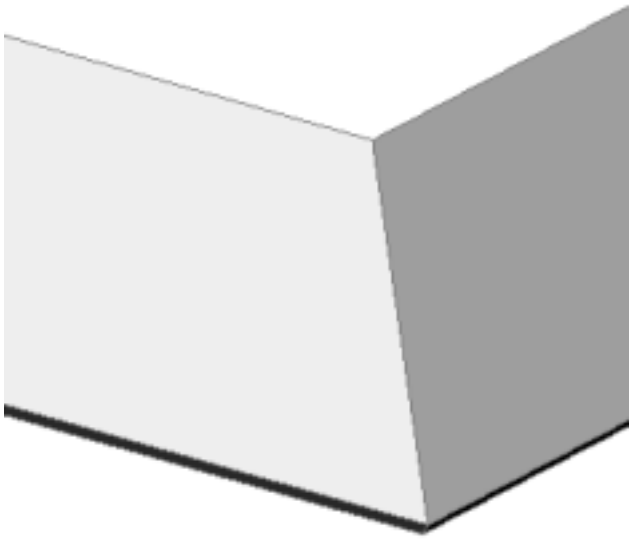


Figure 6.1206

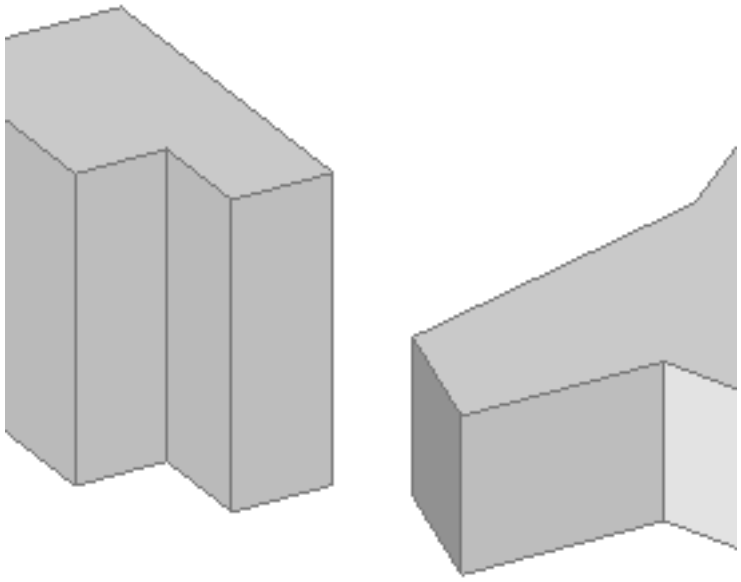


Figure 6.1207

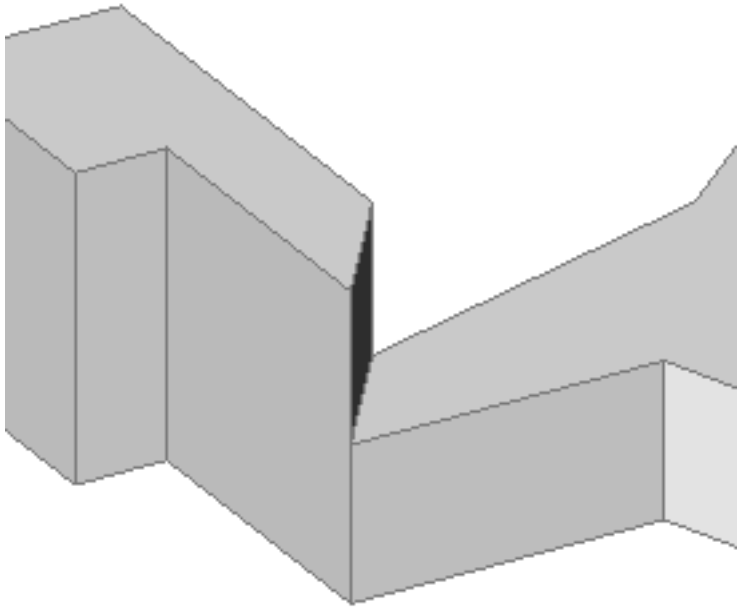


Figure 6.1208



Figure 6.1209



Figure 6.1210

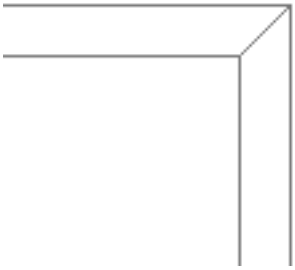


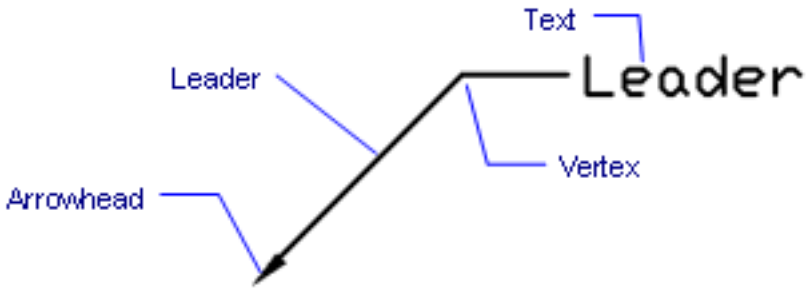
Figure 6.1211



Figure 6.1212



Figure 6.1213



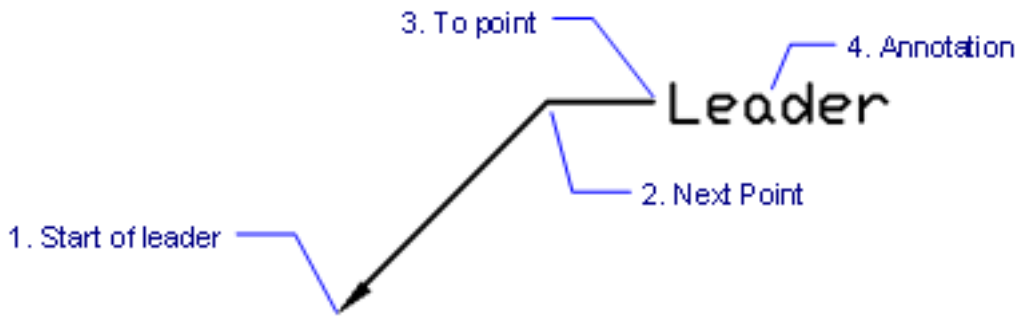
6.20.20 Leader

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws leaders. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdR

Accessing the Command

command bar: **leader** aliases: **le lead** : **Leader Prompts you in the command bar:** Start of leader: **(Pick a point)** Next point: **(Pick a point)** To point [Format/Undo/Annotation] <Annotation>: **(Pick a point, or else enter an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image270.png})

Figure 6.1214



Command Options

Option	Description
To point	Specifies the next vertex: . • Pick a point. • Press **Enter** to stop adding vertices and start the Annotation option.
Format	Specifies the leader style; prompts you: **[Arrow/None/SPline/STraight/Exit] <Exit>** - **(enter an option.)** • **Arrow** - Draws an arrowhead at the leader's start point (default) • **None** - Draws no arrowhead • **Spline** - Draws the leader as a spline (6.27.65) • **Straight** - Draws the leader as straight line segments (default) • **Exit** - Returns to the previous prompt
Undo	Undoes the last leader line segment
Annotation	Starts adding text to the end of the leader line. The annotation is created as an Mtext (6.21.35) entity; prompts you: 1. **First line of annotation text <options>** - **(type a line of text, and then press Enter to start a second line of text. Alternatively, press Enterright away to choose the Options option; see below.)** 2. **Next line of annotation text** - **(press Enter to stop adding text, or else type the next line, then press Enter.)** Caution: the annotation is independent of the leader line; when moving a leader, be sure to include the annotation in the selection set.
Options	Specifies the type of annotation; prompts you: **Dimension text options [Block/Copy/None/Tolerance/Mtext] <Mtext>** - **(enter an option)**: • **Block** - uses a block as the annotation; displays prompts identical to that of the -Insert (6.17.14) command • **Copy** - uses any other text entity in the drawing as the annotation; prompts you: **Select an entity** - select a existingtext (6.29.12) or mtext (6.21.35) entity, and converts text entities to mtext • **None** - creates a leader without annotation. • **Tolerance** - enter the annotation as a tolerance notation through the Geometric Tolerance (1.21.16) dialog box • **Mtext** - enter annotation as mtext through a Text Formatting toolbar; see theMtext (6.21.35)command

Related Commands

DimLeader (6.11.13) - places leaders with text, blocks, entities, or tolerances. QLeader (6.25.1) - draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders. MLeader (6.21.20) - creates multileader entities using the current multileader style. DimLinear (6.11.14) - places linear dimensions in drawings: horizontal, vertical, and rotated. DimAligned (6.11.3) - places aligned dimensions. DimBaseline (6.11.6) - places additional dimensions from a common baseline. DimContinue (6.11.9) - continues dimensions. DimDiameter (6.11.10) - places diameter dimensions. DimRadius (6.11.17) - places radius dimensions. DimCenter (6.11.8) - places center marks. DimAngular (6.11.4) - places angular dimensions. DimOrdinate (6.11.15) - places x and y ordinate dimensions. Tolerance (6.29.17) - places tolerance tags in drawings. Dim (6.11.1) - places and edits dimensions at the 'Dimensioning command:' prompt. Dim1 (6.11.2) - places and edits a single dimension at the 'Dimensioning command:' prompt. DimEdit (6.11.12) - changes the wording and angle of dimension text; changes the angle of extension lines. DimTEdit (6.11.23) - changes the position of dimension text. DdEdit (6.10.14) - edits dimension text. Properties (6.24.49) - edits the properties of dimensions through a palette. DimStyle (6.11.20) - creates and edits dimension styles through the Drawing Explorer. -DimStyle (6.11.21) - saves and applies dimensions styles at the command line. Style (6.27.53) - sets the text style used by dimensions. DimOverride (6.11.16) - overrides the values of the current dimension style.

Figure 6.1215



6.20.21 Lengthen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Changes the length of open objects, such as lines, polyline segments, and arcs. **Note:** This command is largely superseded by grips editing, as well as by the Trim (6.29.26) and Extend (6.13.19) commands.

Accessing the Command



command bar: **lengthen** aliases: **len editlen** menu bar: **Modify | Lengthen** toolbar:**Modify |** : **Lengthen Prompts you in the command bar:** Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>: **(Enter an option, or choose an entity)**

Command Options

Option	Description
Select entity to list length	Select a valid entity, and then the program reports the current length of an open entity in the current units,such as **Current length: 7’-11 7/8”**
DYnamic	Changes the length interactively; prompts you: **Select entity to change or [edit Mode]- (select an entity; the prompt that appears next depends on the entity you selected)**: • Line: **New end point for line** - drag the line longer or shorter • Arc: **New start point** -drag the arc longer or shorter • Polyline: **New end point** - drag the segment longer or shorter **edit Mode**optionreturns to original prompt to switch lengthening modes.)
Increment	Changes the length or angle by a specified amount; prompts you: **Enter length to increment or [Angle] <0”> - (specify a positive number to lengthen the entity, or a negative number to shorten it, or else enter A)**: • **Select entity to change or [edit Mode] - (select the entity to lengthen.)** • **Enter angle increment <0>- (specify the angle by which to change the entity (applies to arcs only).)**
Percent	Changes the length of entities by a percentage; prompts you: **Enter percent length <0.0000> - (enter a number to lengthen or shortens the entity to a percentage of the current length.)** For example, enter 25 (for 25%), and a 1”-long line is shorted to 0.25”.Enter 150%, and the 1” line becomes 1.5”.

Total	Changes the length to the amount specified; prompts you: **Angle/<Enter total length (0.0000)>** - **(enter an option)**: • **Enter total length** - specifies the size to which to lengthen or shorten the entity; for example, enter 2 and a 1”-long line is lengthened to 2”. • **Enter angle increment** -specifies the angle to which to size the arc; for example, enter 45 and a 180-degree arc is shortened to 45 degrees

Procedures

- Change the length of an entity dynamically (1.23.22)
- Modify the included angle of an arc (1.23.22)

Related Commands

Extend (6.13.19) - lengthens open entities. Stretch (6.27.49) - stretches entities longer and shorter. Change (6.9.10) - changes the length of lines. Trim (6.29.26) - shortens entities.

Figure 6.1216



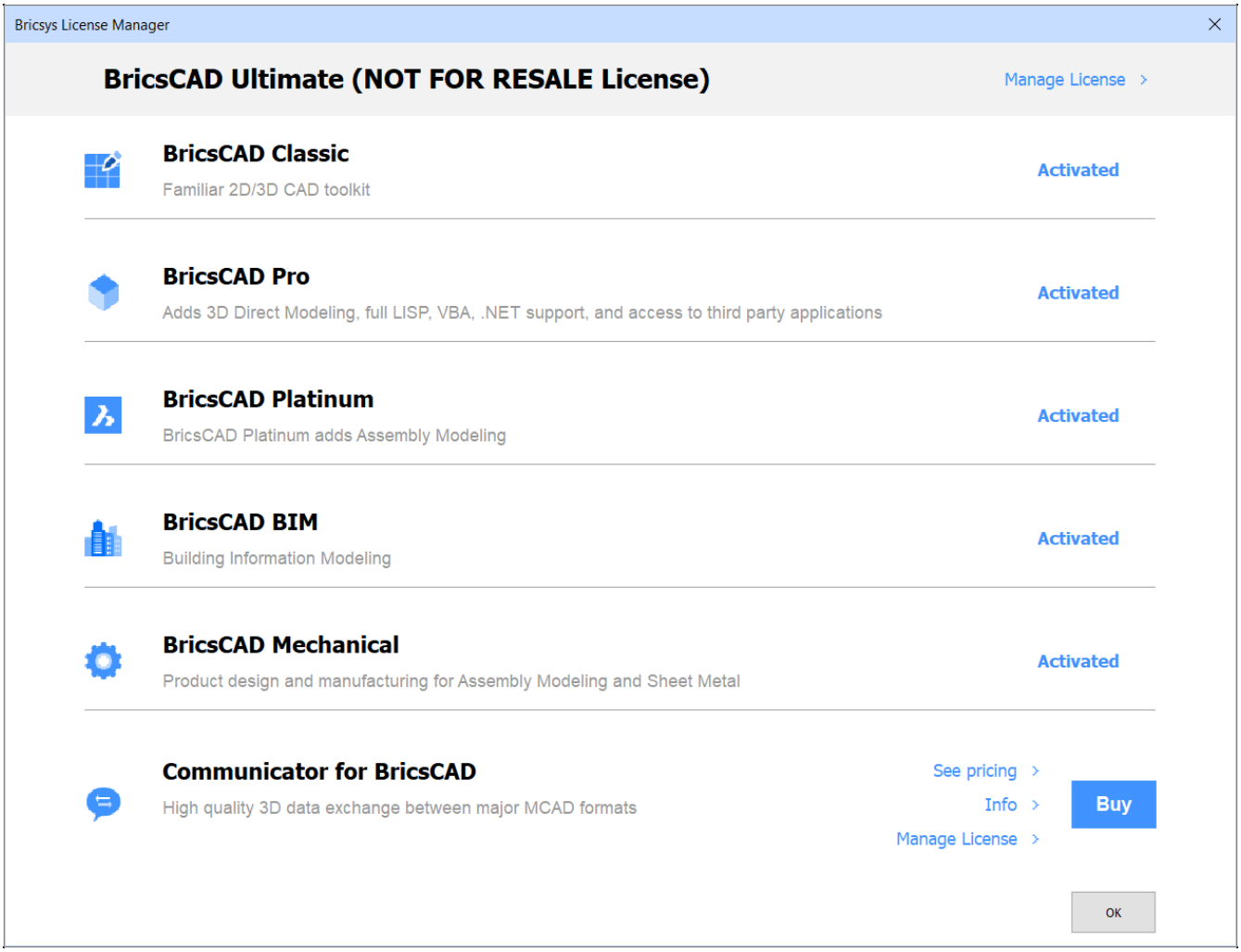
6.20.22 LicenseManager

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Summarizes the software's licenses for BricsCAD, BIM, Mechanical, and Communicator through a dialog box.

Accessing the Command

 command bar: **licensemanager** menu bar: **Help | License Manager** ribbon:**Home | Help | BricsCAD License** : **LicenseManager** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/dlg.png})

Figure 6.1217



Command Options

Option	Description
Manage License	Displays the Bricsys License Manger dialog box for activating the license; see LicProperties (6.20.24) command
See pricing	Opens the computer's default Web browser to the https://www.bricsys.com/estore/ page, from which the license can be purchased
Info	Reports on the status of the license by displaying the Info dialog box, such as this one: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/licensecheck-dlg.png})
Buy	Opens the Web browser to the https://www.bricsys.com/estore/ page

Related Commands

LicEnterKey (6.20.23) - enters the license key number. LicProperties (6.20.24) - reports the BricsCAD license information in a dialog box. Allows to modify or deactivate a BricsCAD single user or volume license key. LicPropertiesBim (6.20.25) - reports the BricsCAD BIM license information in a dialog box. Allows to modify or deactivate a BricsCAD BIM single user or volume license key. LicPropertiesCommunicator (6.20.26) - reports the BricsCAD Communicator license information in a dialog box. Allows to modify or deactivate a BricsCAD Communicator single user or volume license key. LicPropertiesMechanical (6.20.27) - reports the BricsCAD Mechanical license information in a dialog box. Allows to modify or deactivate a BricsCAD Mechanical single user or volume license key. RunAsLevel (6.26.46) - allows to run BricsCAD in a lower license level.

Figure 6.1218

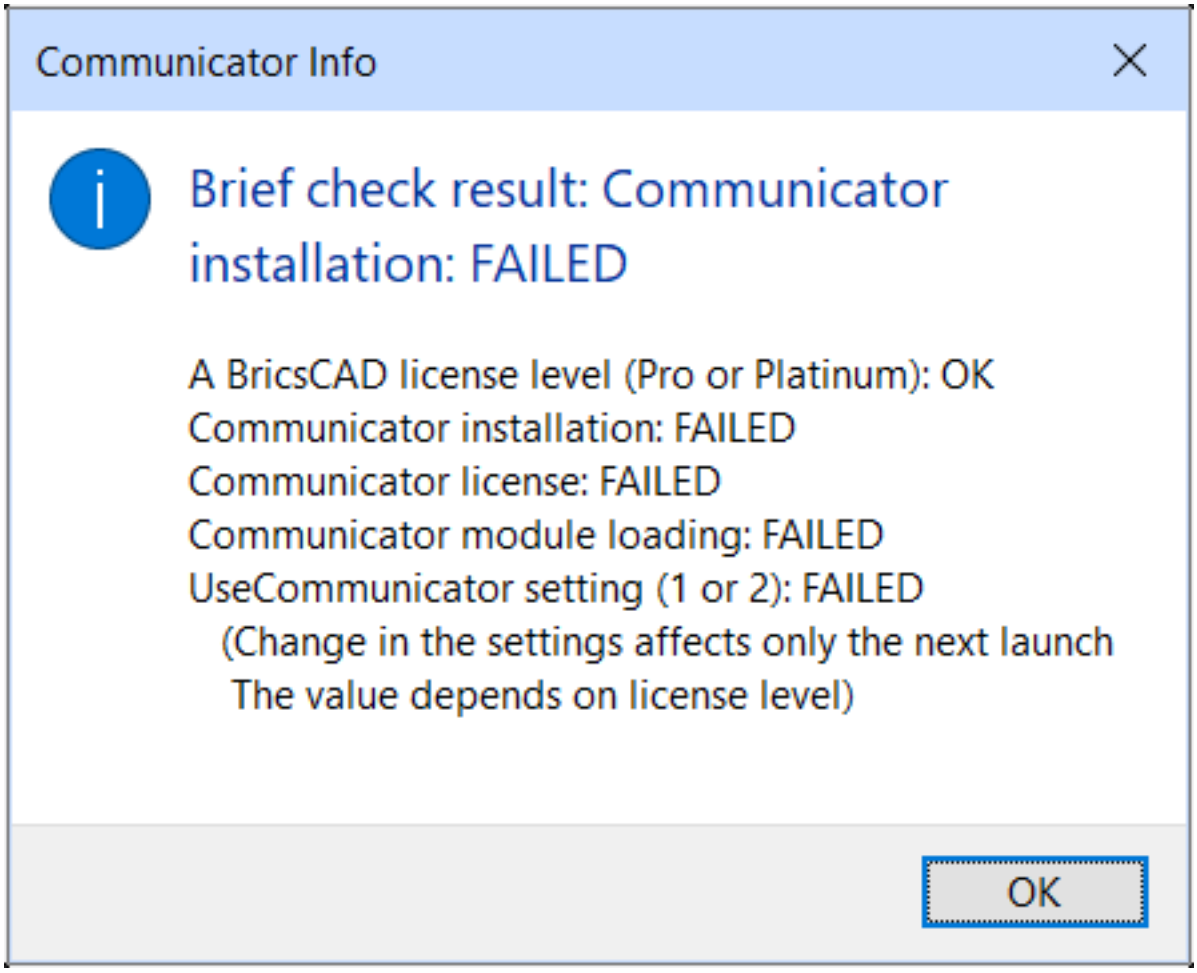


Figure 6.1219



6.20.23 LicEnterKey

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Enters a license key number in a dialog box (short for "licence enter key") **Notes:** A licence number is required to run BricsCAD longer than 30 days.A separate license key is required for each of the Classic, Pro, and Platinum version of BricsCAD, and for the BIM, Mechanical, and Communicator add-ons.

Accessing the Command

command bar: **licenterkey** transparent: **'licenterkey** : **LicEnterKey** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/LicenseManagerActivate.png}) Enter your license information, and then click OK. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/License
Close BricsCAD, and then restart it. Use this command only to change the license number. To view the license number, use the LicProperties (6.20.24) command.

Figure 6.1220

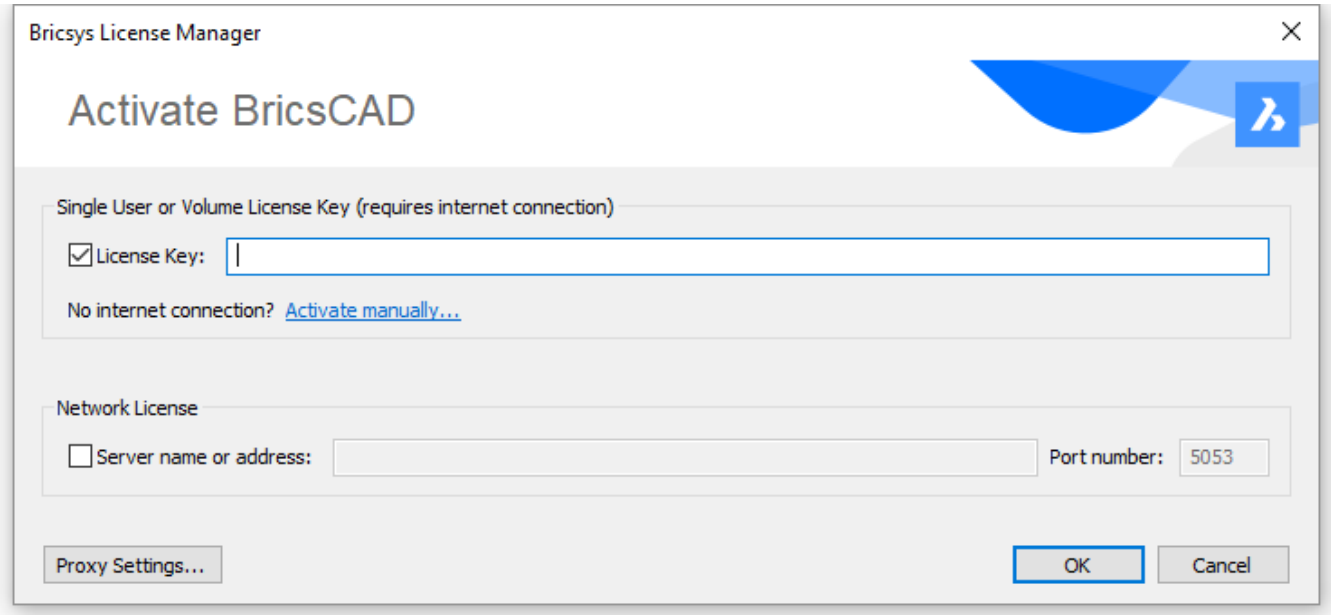
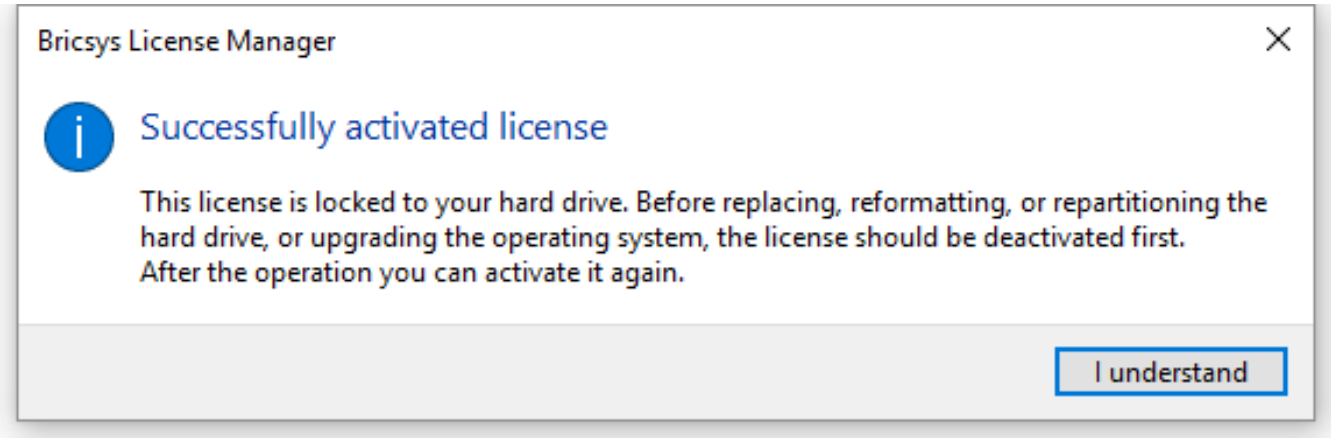


Figure 6.1221



Command Options

Option	Description
License Key	Activates a single-user license and a volume-license key for computers connected to the Internet: 1. Enter a license key in the **License Key** field. 2. Press the **OK** Button. 3. Wait for the license to be activated over the internet. A license file is created on your computer.
Activate manually...	Activates a single-user or volume-license key on a computer not connected to the internet. For this to work, your computer must be connected to another computer with an internet connection, as it is needed to retrieve a valid license file from the BricsCAD license server. 1. Logon to your account on the Bricsys website and select **Licenses and support contracts**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands 2. Click the **Manage**button of the license you want to activate. 3. Copy/paste the computer host ID of the computer you want to install BricsCAD on in the **Host ID** field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so 4. Click the **Activate** button. A license file is generated and made available for download. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so 5. Click the **Download License File** button to download the **BricsCADVxx.lic** file. 6. Copy the license file to the computer onto which you want to install BricsCAD. 7. Click the **Browse** button () in the Bricsys License Manager - Manual Activation dialog box. 8. Select the license file. 9. Click**OK**.

Network License	Retrieves a network license from a license server: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/Lic 1. Enter the host name or IP address of the license server in the **Server name or address** field. 2. Type the port number in the **Port Number** field. 3. Click**OK**.
Proxy Settings...	Specifies proxy settings when your computer connects to the internet or a local network through a proxy server: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/Lic 1. Enter a name or IP address in the **Host Name** field. 2. Specify a number in the Port Number field. 3. If necessary, turn on the **Username** option, and then enter a username and password. 4. Click **OK**.

Related Commands

Status (6.27.46) - reports on the state of the drawing. LicProperties (6.20.24) - reports the license number and owner. LicenseManager (6.20.22) -displays the License Manager dialog, summarizing your available licenses (BricsCAD / BIM / Mechanical / Communicator).

Figure 6.1222

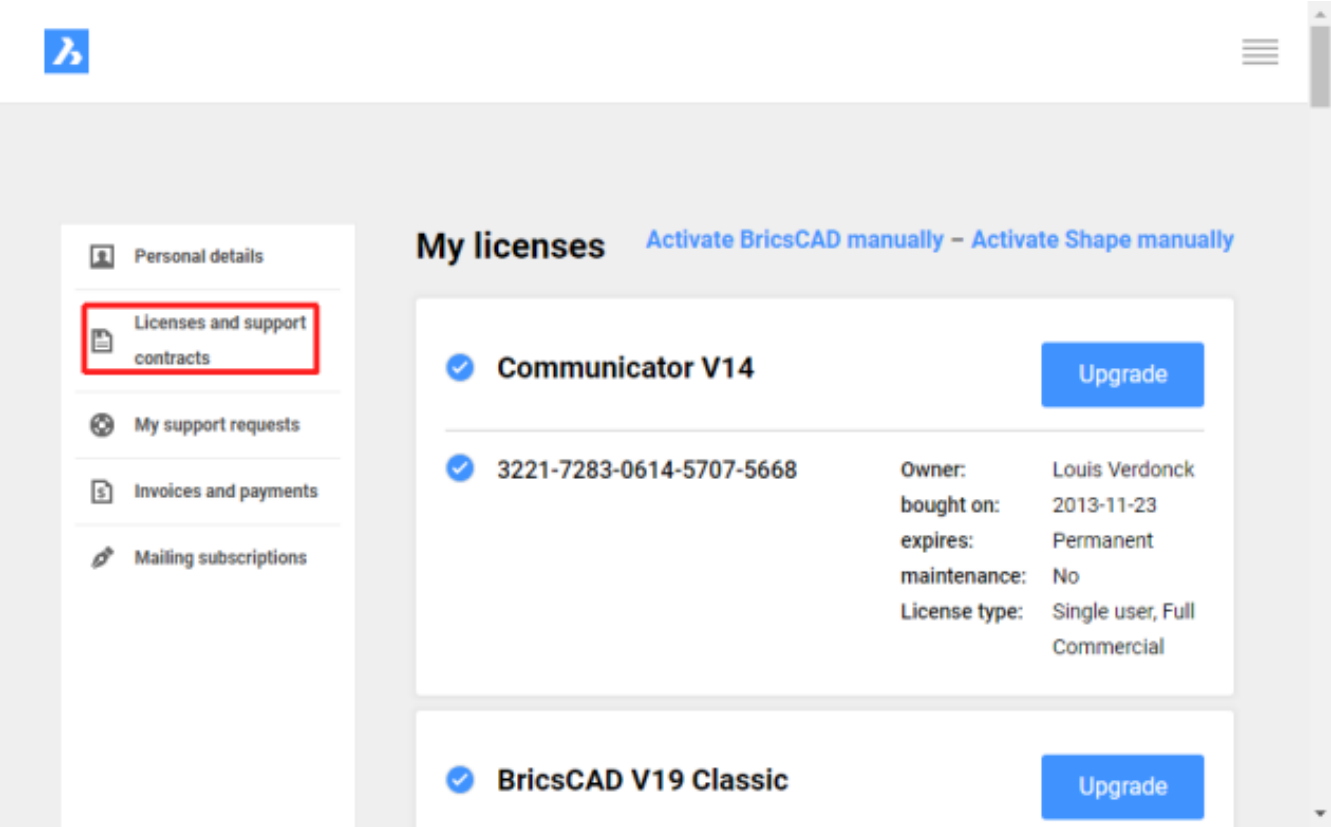


Figure 6.1223

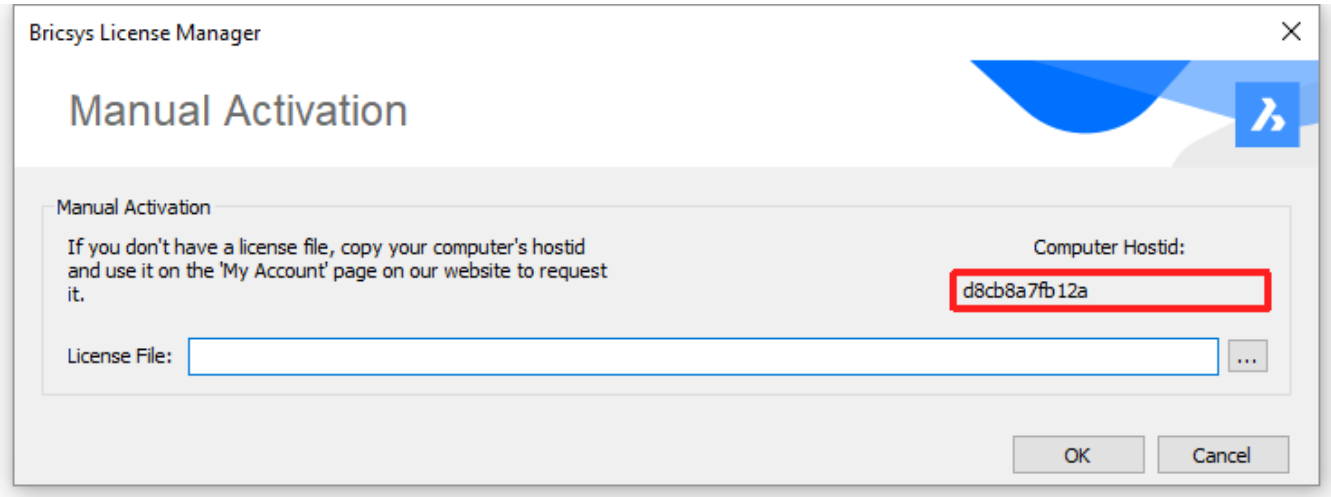


Figure 6.1224

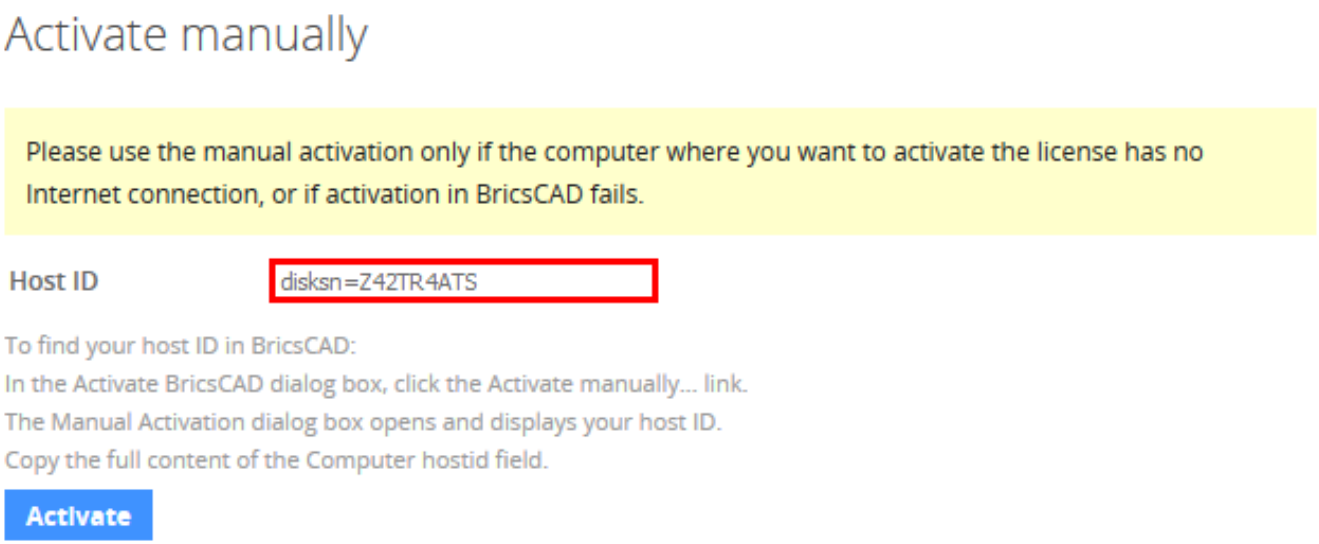


Figure 6.1225

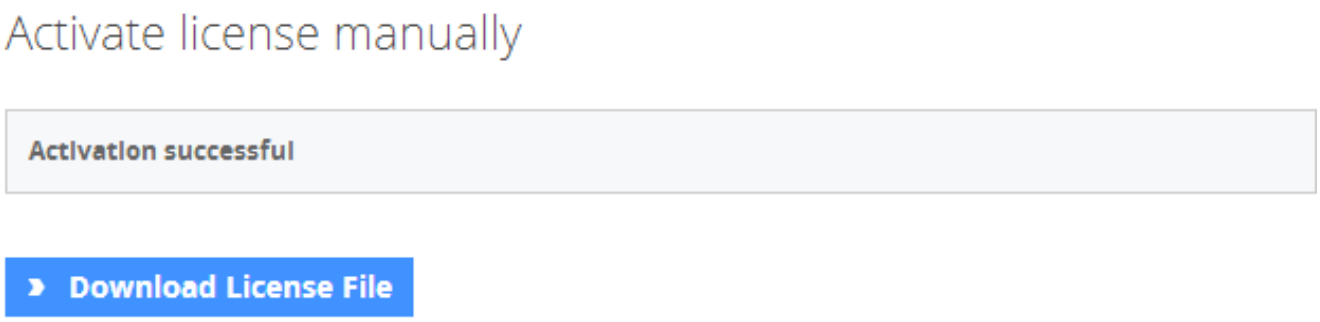


Figure 6.1226

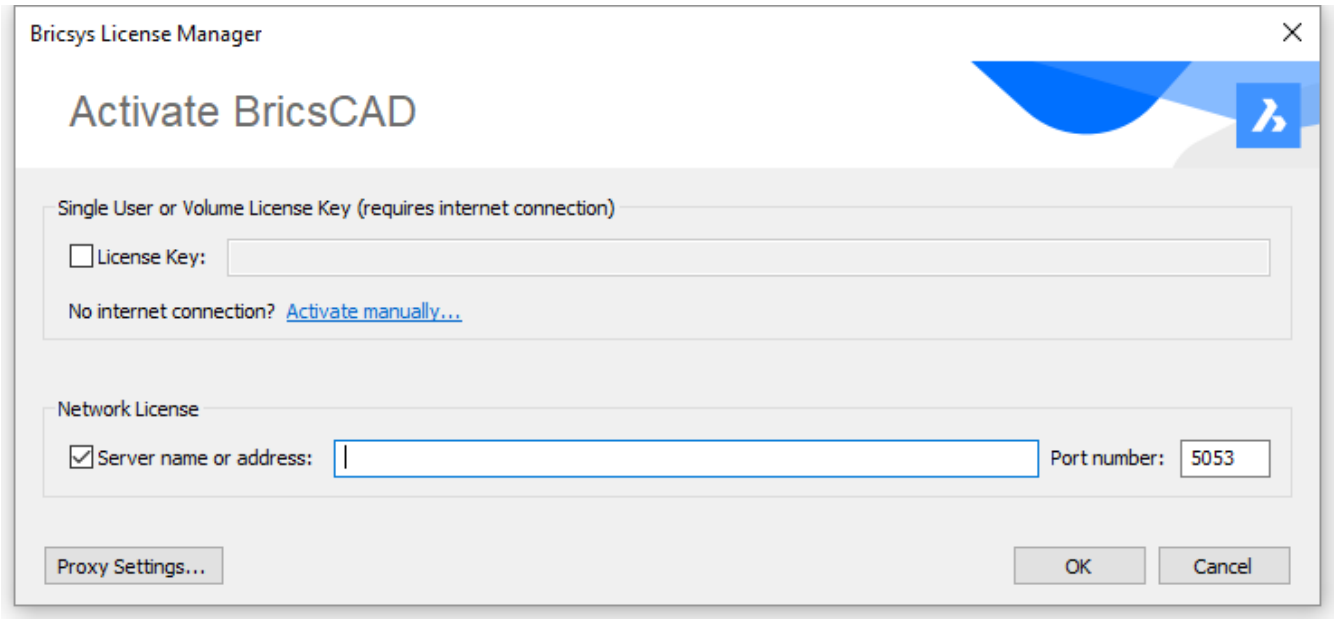


Figure 6.1227

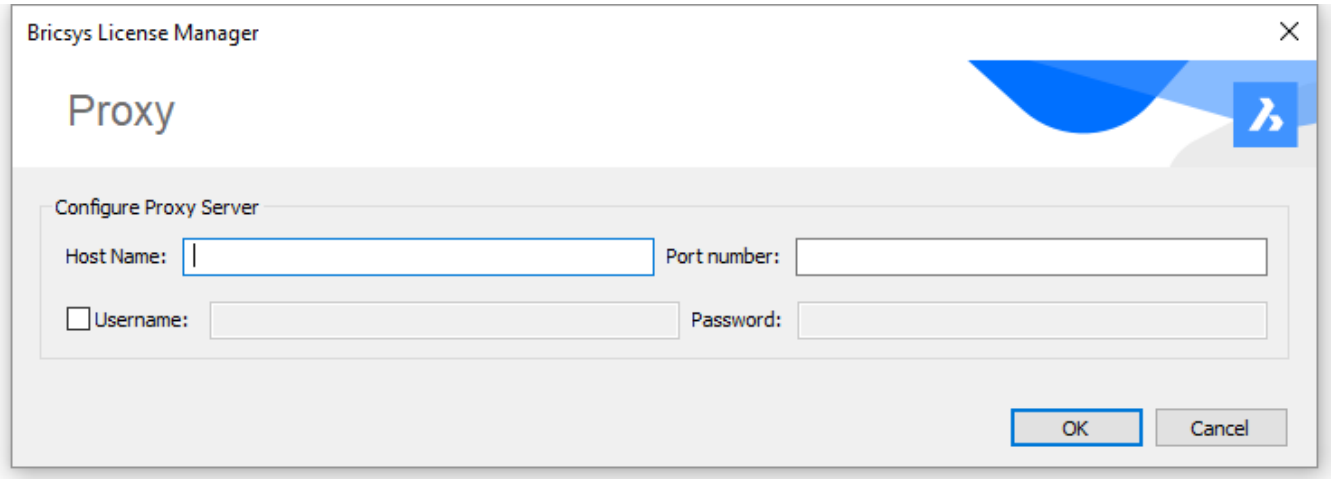


Figure 6.1228



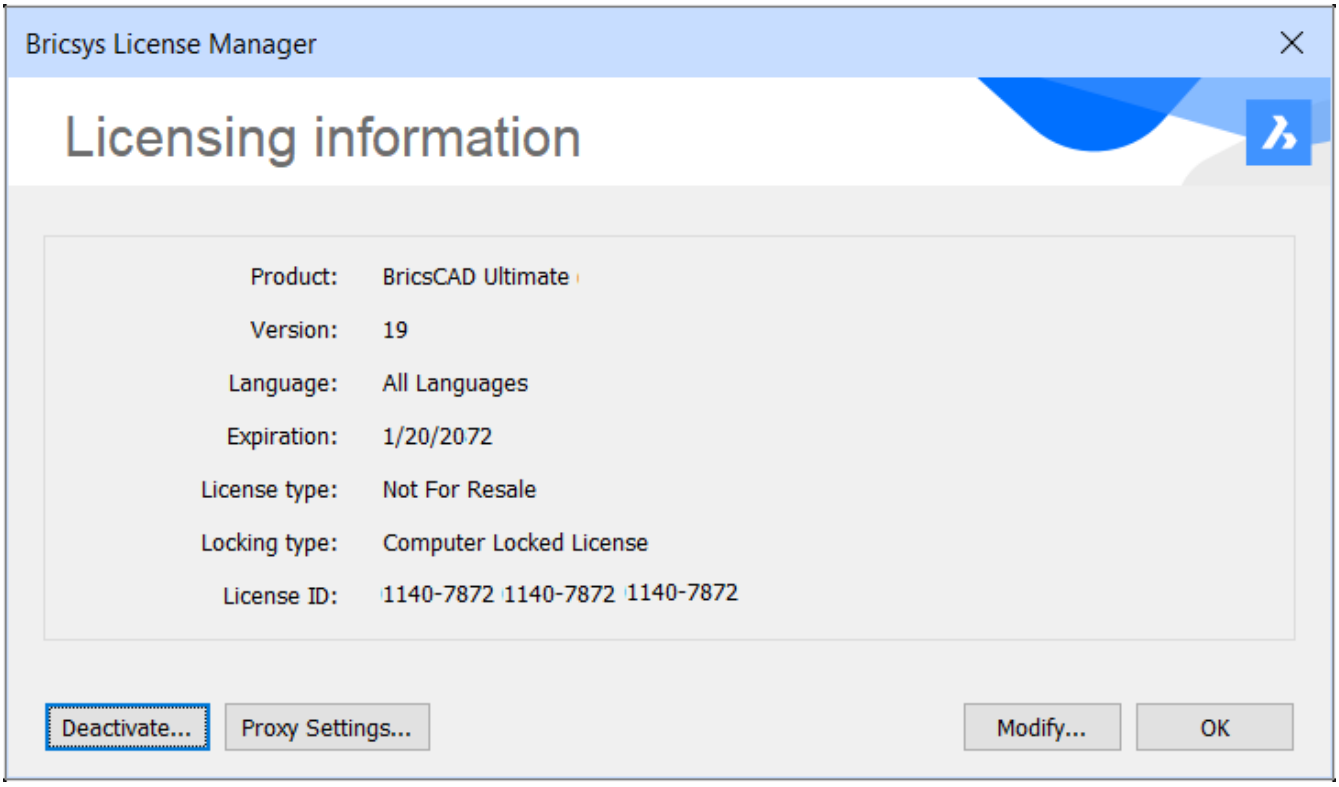
6.20.24 LicProperties

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Reports on,modifies, and deactivates BricsCAD single-user or volume-license keys.

Accessing the Command

command bar: **licproperties** transparent: **'licproperties** : **LicProperties** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/licencemanager-managelicense-dlg.png}) Review license information, and then click **OK**.

Figure 6.1229



Command Options

Option	Description
Deactivate	To use this license on another computer, deactivate the license key; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/Lic Click **Yes**, and BricsCAD can no longer be used on this computer, but is available for activation on another computer. An internet connection is needed to deactivate a license key.You deactivate a license key • To migrate the license key to a different computer • When removing BricsCAD from a computer permanently • Before installing a new disk drive, or when changing partitions, or for significant upgrades to the operating system, such as from Windows 7 to 8

Proxy Settings	Specifies proxy settings necessary when your computer connects to the internet through a proxy server; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/Lic 1. Enter a name or IP address in the **Host Name** field. 2. Specify a number in the Port Number field. 3. If necessary, turn on the **Username** option, and then enter a username and password. 4. Click **OK**.
Modify	Displays the **Activate BricsCAD** dialog box; see theLicEnterKey (6.20.23) command.

Related Commands

Status (6.27.46) - reports on the state of the drawing. LicEnterKey (6.20.23) - changes the license number and owner. LicenseManager (6.20.22) - displays the License Manager dialog, summarizing your available licenses (BricsCAD/Communicator/Sheet Metal) LicPropertiesBim (6.20.25) - reports the BricsCAD BIM license information in a dialog box. Allows to modify or deactivate a BricsCAD BIM single user or volume license key. LicPropertiesCommunicator (6.20.26) - reports the BricsCAD Communicator license information in a dialog box. Allows to modify or deactivate a BricsCAD Communicator single user or volume license key. LicPropertiesMechanical (6.20.27) - reports the BricsCAD Sheet Metal license information in a dialog box. Allows to modify or deactivate a BricsCAD Sheet Metal single user or volume license key.

Figure 6.1230

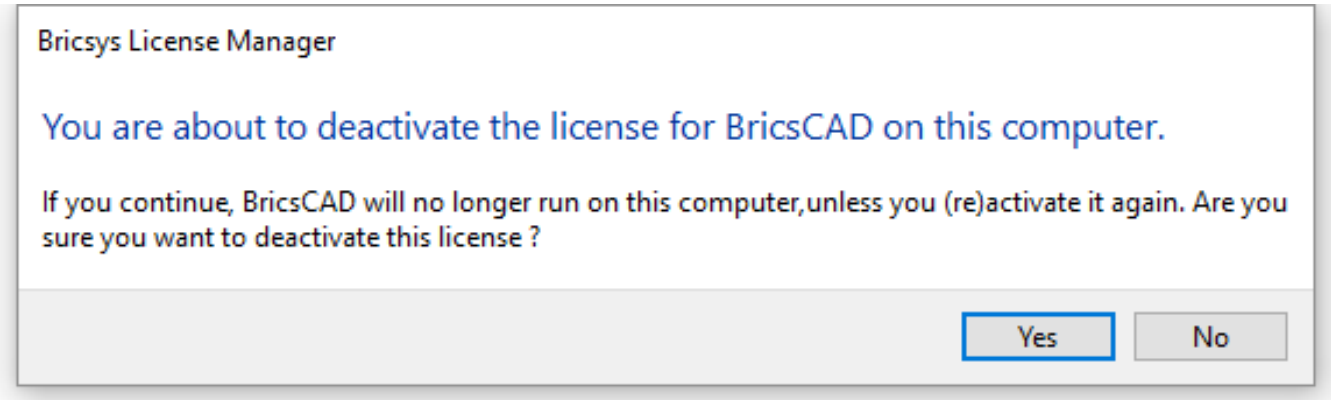


Figure 6.1231

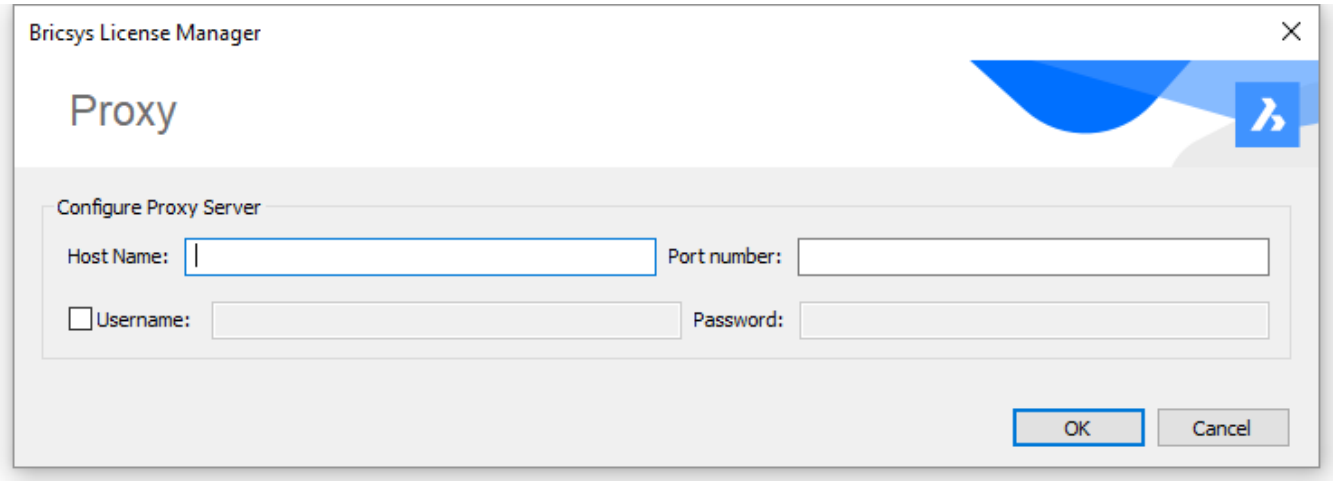


Figure 6.1232



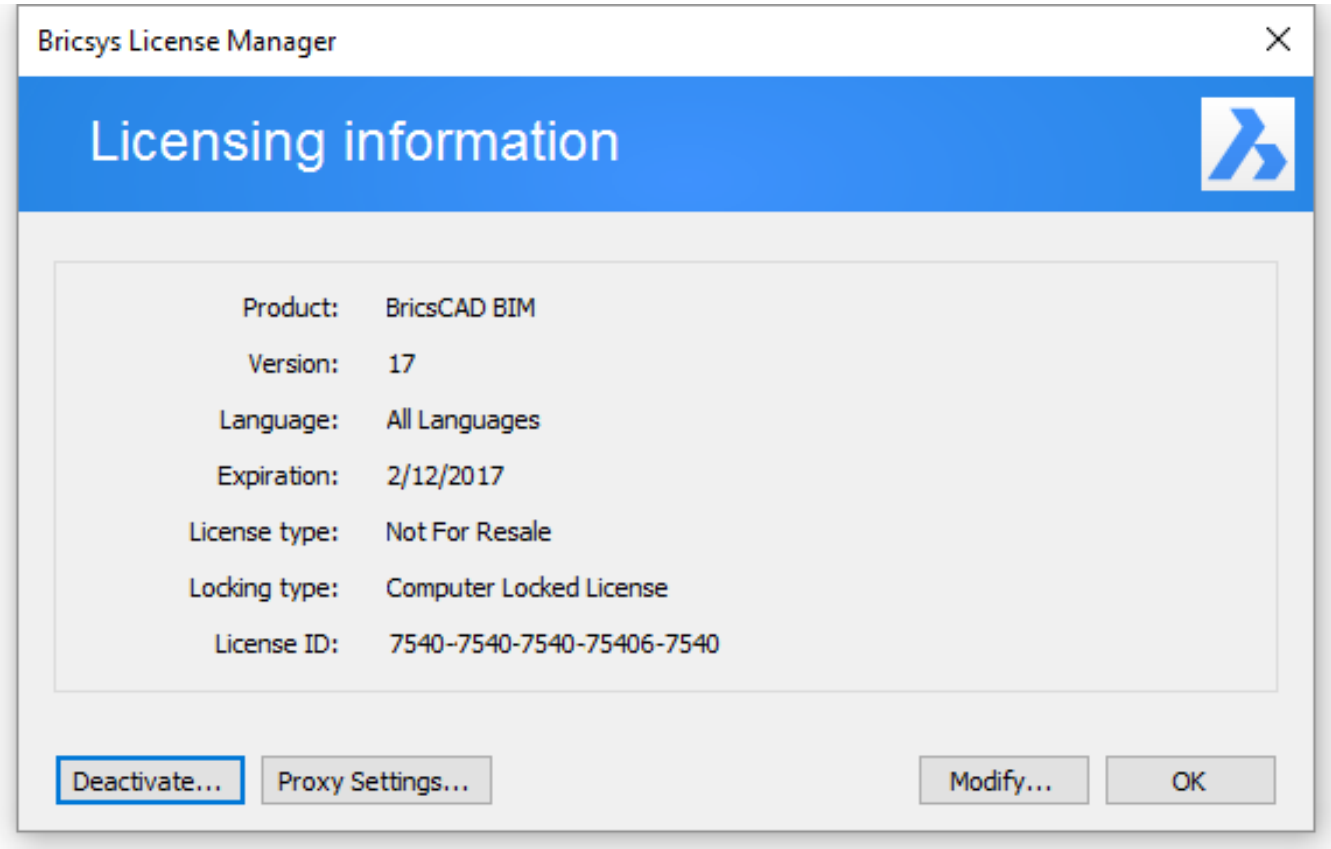
6.20.25 LicPropertiesBim

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Reports on, modifies, and deactivates the BricsCAD BIM single-user or volume-license key though a dialog box.

Accessing the Command

command bar: **licpropertiesbim** transparent: **'licpropertiesbim** : **LicPropertiesBim** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/LicenseInformationBIMDialog.png}) Review license information, and then click **OK**.

Figure 6.1233



Command Options

See LicProperties (6.20.24) command.

Related Commands

Status (6.27.46) - reports on the state of the drawing. LicEnterKey (6.20.23) - changes the license number and owner. LicenseManager (6.20.22) - displays the License Manager dialog, summarizing your available licenses (BricsCAD/Communicator/Sheet Metal) LicProperties (6.20.24) - reports the BricsCAD license information in a dialog box. Allows to modify or deactivate a BricsCAD single user or volume license key. LicPropertiesCommunicator (6.20.26) - reports the BricsCAD Communicator license information in a dialog box. Allows to modify or deactivate a BricsCAD Communicator single user or volume license key. LicPropertiesMechanical (6.20.27) - reports the BricsCAD Mechanical license information in a dialog box. Allows to modify or deactivate a BricsCAD Mechanical single user or volume license key.

Figure 6.1234



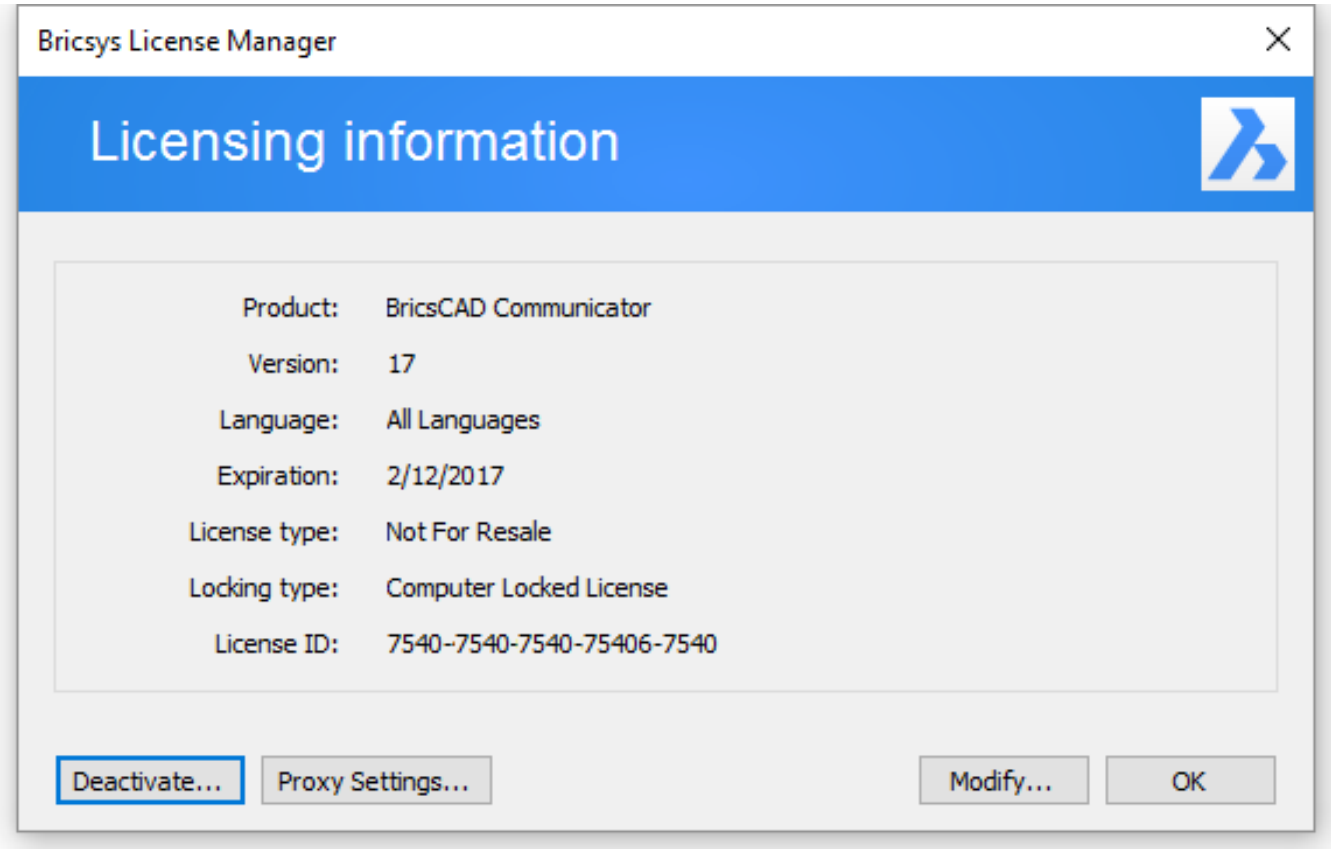
6.20.26 LicPropertiesCommunicator

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Reports on, modifies, and deactivates the BricsCAD Communicator single-user or volume-license key though a dialog box.

Accessing the Command

command bar: **licpropertiescommunicator** transparent: **'licpropertiescommunicator** : **LicPropertiesCommunicator** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/Li
Review license information, and then click OK.

Figure 6.1235



Command Options

See LicProperties (6.20.24) command.

Related Commands

Status (6.27.46) - reports on the state of the drawing. LicEnterKey (6.20.23) - changes the license number and owner. LicenseManager (6.20.22) - displays the License Manager dialog, summarizing your available licenses (BricsCAD/Communicator/Sheet Metal) LicProperties (6.20.24) - reports the BricsCAD license information in a dialog box. Allows to modify or deactivate a BricsCAD single user or volume license key. LicPropertiesBim (6.20.25) - reports the BricsCAD BIM license information in a dialog box. Allows to modify or deactivate a BricsCAD BIM single user or volume license key. LicPropertiesMechanical (6.20.27) - reports the BricsCAD Mechanical license information in a dialog box. Allows to modify or deactivate a BricsCAD Mechanical single user or volume license key.y.

Figure 6.1236



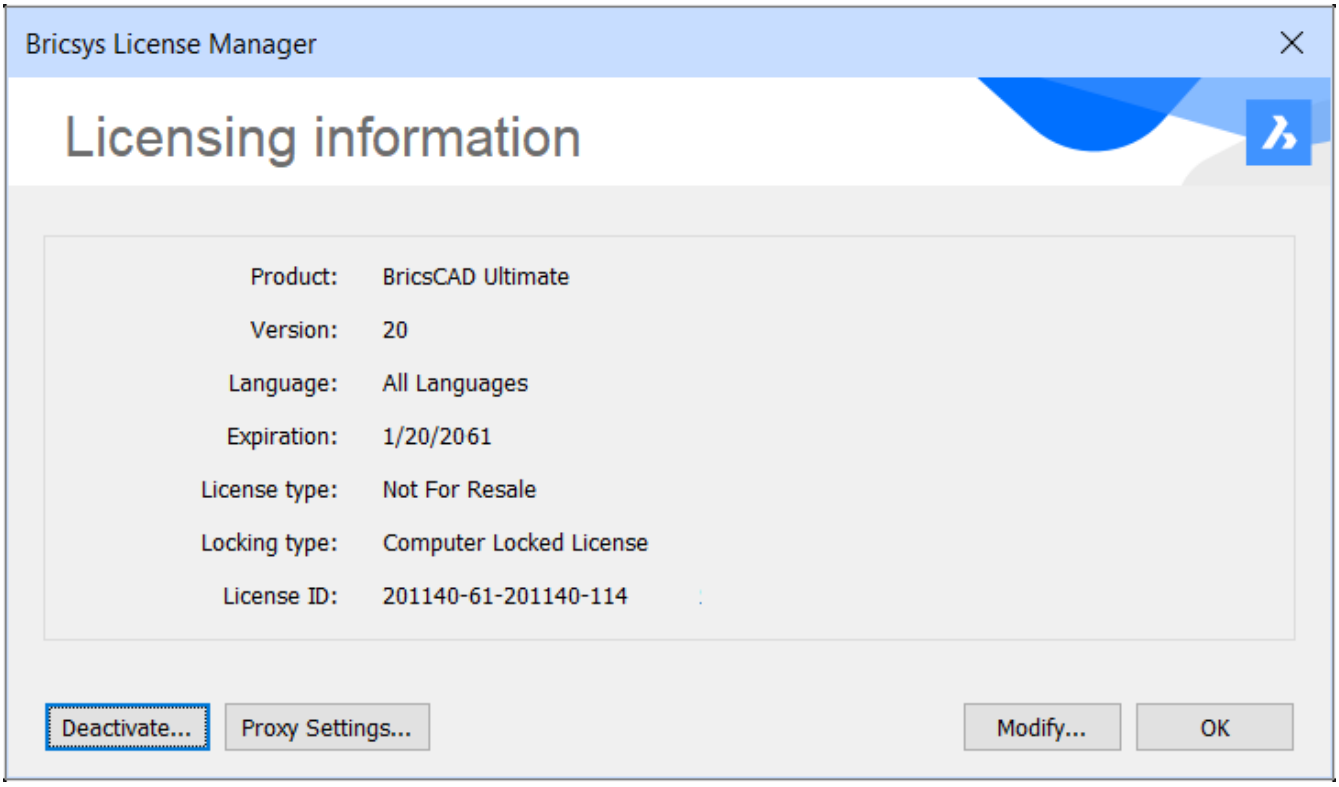
6.20.27 LicPropertiesMechanical

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Reports on, modifies, and deactivates the BricsCAD Mechanical single-user or volume-license key though a dialog box. (This command supersedes the LicPropertiesSheetMetal command.)

Accessing the Command

command bar: **licpropertiesmechanical** transparent: **'licpropertiesmechanical** : **LicPropertiesMechanical** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/L/licpropertiesmechadlg.png}) Review license information, and then click **OK**.

Figure 6.1237



Command Options

See LicProperties (6.20.24) command.

Related Commands

Status (6.27.46) - reports on the state of the drawing. LicEnterKey (6.20.23) - changes the license number and owner. LicenseManager (6.20.22) - displays the License Manager dialog, summarizing your available licenses (BricsCAD/Communicator/Sheet Metal) LicProperties (6.20.24) - reports the BricsCAD license information in a dialog box. Allows to modify or deactivate a BricsCAD single user or volume license key. LicPropertiesBim (6.20.25) - reports the BricsCAD BIM license information in a dialog box. Allows to modify or deactivate a BricsCAD BIM single user or volume license key. LicPropertiesCommunicator (6.20.26) - reports the BricsCAD Communicator license information in a dialog box. Allows to modify or deactivate a BricsCAD Communicator single user or volume license key.

Figure 6.1238



6.20.28 Light

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Places light glyphs in drawings for generating more realistic renderings.

Accessing the Command

command bar: **light** alias: **lighting** ribbon:**View | Lights | Create | Light** : **Light Prompts you in the command bar:** Enter light type [Point/Spot/Web/Distant] <Spot>: **(Enter an option, or else press Enter to create a spot light).**

Command Options

Option	Description
Point	Creates a point light; see thePOINTLIGHT (6.24.42) command for the meaning of its options
Spot	Creates a spot light; see theSPOTLIGHT (6.27.42).command for the meaning of its options
Web	Creates a web light; see theWEBLIGHT (6.33.6) command for the meaning of its options
Distant	Creates a distant light; see DISTANTLIGHT (6.10.32)command for the meaning of its options

Related Commands

Distantlight (6.10.32) - creates a distant light. Lightlist (6.20.29) - displays the Drawing Explorer - Lights dialog window. Materials (6.21.11) - creates materials and edits their properties through the Drawing Explorer. Pointlight (6.24.42) - creates a point light. Render (6.26.22) - calculates a photorealistic image of the 3D model. Spotlight (6.27.42) - creates a spot light. Sunproperties (6.27.57) - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog. Weblight (6.33.6) - creates a web light.

Figure 6.1239



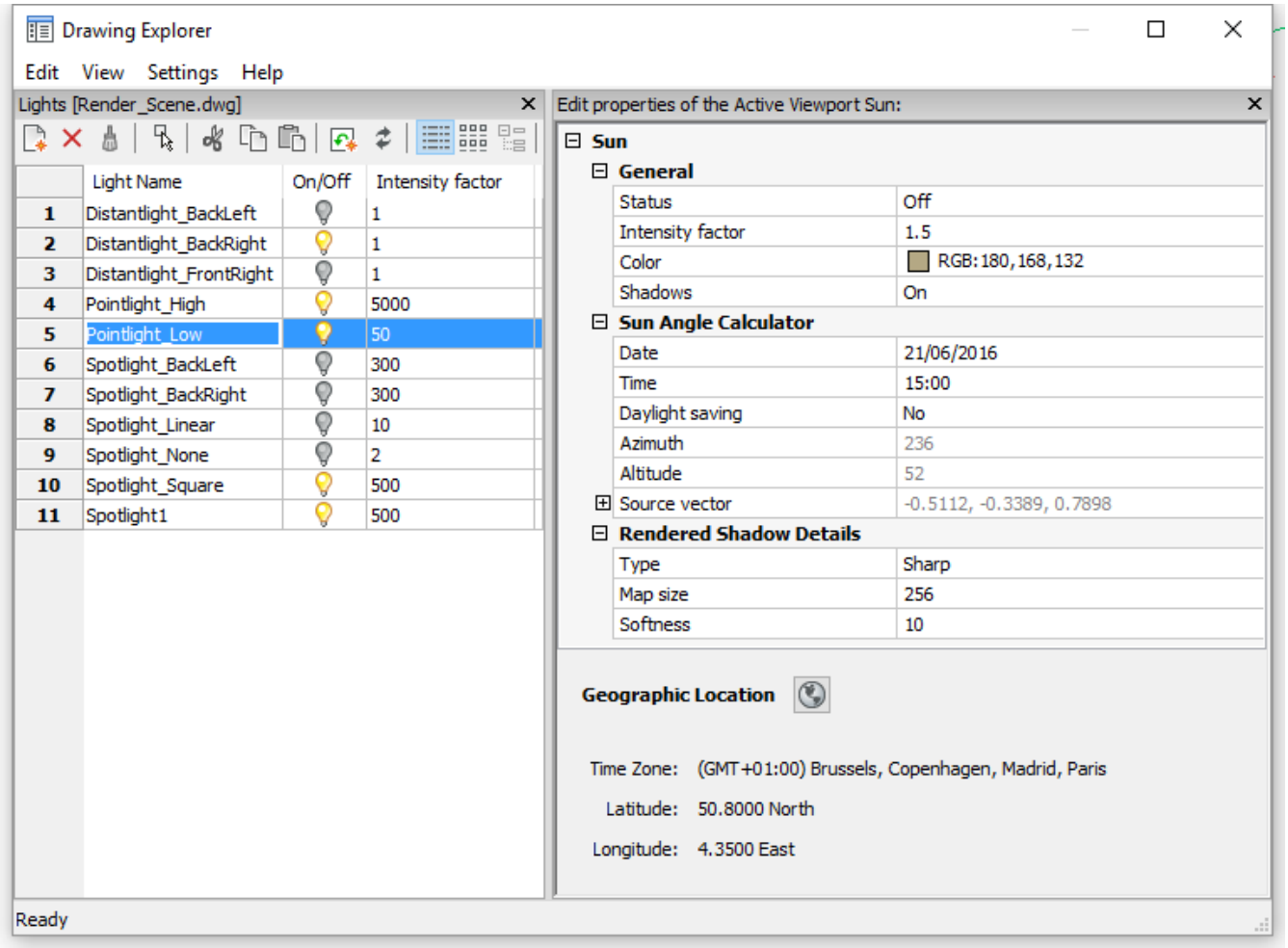
6.20.29 LightList

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and edits lights in the current drawing through the Light section of the Drawing Explorer dialog box.

Accessing the Command

 command bar: **lightlist** alias: **LL** menu bar: **View | Rendering | Lights | Lights Tools | Drawing Explorer | Lights** ribbon:**Home | Settings | Drawing Explorer | Lights View | Lights | Lights** toolbar: **Lights |**  : **LightList** Displays the Lights section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

Figure 6.1240



Command Options

Toolbar Option	Description
 New	Adds a light to the current drawing.The Drawing Explorer disappears, and then you are prompted at the command line by the Light command. When done, the Drawing Explorer returns. See the Light (6.20.28) command.
 Delete	Deletes the selected light(s) without confirmation.
Select in drawing	Displays a light's properties int he Properties panel: 1. Click the **Select in Drawing** button.The Drawing Explorer disappears. 2. Select a light's glyph in the drawing. Its properties appear in the Properties panel. The Drawing Explorer does not return, so you need to enter the LightList command again.

Lights Options	Description
Light Name	Specifies the name of the light; click to rename.
 ON /  OFF	Turns the selected light(s) on or off.
Intensity Factor	Specifies an intensity factor that makes the light light brighter or dimmer, at its source.
Edit Properties of the Active Viewport Sun	Edits the properties of a distance light named Sun, which simulates sunlight. See SunProperties (6.27.57) command.

Context Menu Options	Description
Rename	Edits the name of the light.
Select all	Selects all lights in the light list.
Invert Selection	Inverts the lights that are selected, by unselecting the selected ones and vice versa.

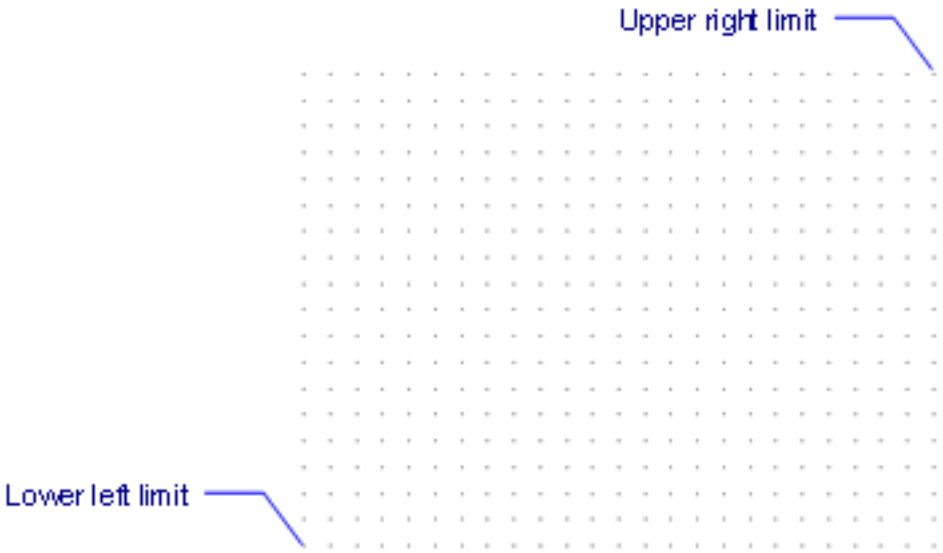
Related Commands

Light (6.20.28): Creates a new light. Render (6.26.22): Calculates a photorealistic image of the 3D model. Sunproperties (6.27.57): Displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog. GeographicLocation (6.15.15): Sets the geographic location of the drawing. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.1241



Figure 6.1242



6.20.30 Limits

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Limits the extent of the drawing and, optionally, of the grid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image267.png})

Accessing the Command

 command bar: **limits** transparent: **'limits** menu bar: **Settings | Drawing Limits** toolbar: **Settings | Limits** : **Limits Prompts you in the command bar:** Limits are off: Set lower left corner or [turn limits ON] <0,0>: **(Enter x,y coordinates, pick a point in the drawing, press Enter to accept the default, or else enter ON)** Upper right corner <420,297>: **(Enter x,y coordinates, pick a point, or else press Enter to accept the default coordinates)**

- When the LIMITSCHECK variable is on, you cannot draw outside of the area defined by this command; the commandreports, "Selected point is outside limits. Please select another point."
- This commands limits theextent of the grid display when1 flag of the GRIDDISPLAY (6.15.21)variable is not set.

Command Options

Option	Description
ON	Turns on limits.
OFF	Turns off limits.
Lower left corner	Specifies the lower left corner of the limits.
Upper right corner	Specifies the upper right corner of the limits.

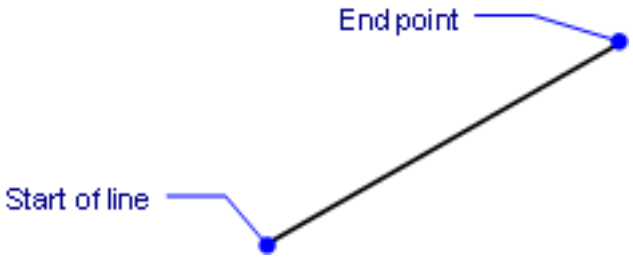
Related Commands

Grid (6.15.21) - uses the limits to specify the extent of the grid display. Plot (6.24.26) and Print (6.24.46) - plot to the limits of the drawing. Zoom (6.35.7) - zooms to the limits (or extents) with the All option.

Figure 6.1243



Figure 6.1244



6.20.31 Line

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws line segments. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image263.png})

Accessing the Command



Command bar: **line** aliases:**l, 3dline** menu bar: **Draw | Line** quad Tab: **Draw | Line** ribbon:**Draw | Draw | Line Model | Draw | Line Sketch | Draw | Line** toolbar: **Draw** : **Line The first time you use the Line command in a drawing, prompts you:** Start of line: **(Pick a point, or enter x,y,z coordinates)** Set end point or [Angle/Length/Undo]: **(Pick a point, enter x,y,z coordinates, or else choose an option)** Set end point or [Angle/Length/Follow/Undo]: **(Pick a point, enter x,y,z coordinates, choose an option, or else press Enter to exit)** Set end point or [Angle/Length/Follow/Close/Undo **(Pick a point, enter x,y,z coordinates, choose an option, or else press Enter to exit) The next time you use the command in the drawing, prompts you:** Start of line or [Follow] <Last point> **- ((Pick a point, enter x,y,z coordinates, or else choose an option)**

Command Options

Option	Description
Start of line	Specifies the starting point of the first line segment: • Pick a point, optionally using entity snaps or an other modifier • Enter x,y coordinates to start a 2D line • Enter x,y,z coordinates to start a 3D line
Last point	Starts the new line at the end of the previously-drawn line: Press **Enter** to use the most recently specified point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
End point	Specifies the endpoint of each line segment: • Pick another point, optionally using entity snaps or an other modifier • Enter x,y coordinates to continue a 2D line • Enter x,y,z coordinates to continue a 3D line

Angle	Draws the line at a user-specified angle and distance; prompts you: 1. **Angle of line** - **(specify the angle, which is measured counter-clockwise from the positive x-axis.)** 2. **Length of line** - **(specify the length of the segment.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Length	Draws the line at a user-specified distance and angle; prompts you: • **Length of line** - **(specify the length of the segment.)** • **Angle of line** - **(specify the angle, measured counterclockwise from the positive x-axis.)**
Undo	Undoes (erases) the last line segment.
Follow	Draws the next line segment at the same angle as the previous segment; prompts you: **Length of line** - **(specify the length of the segment.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Close	Closes the ”polygon” by drawing a line segment between the current endpoint and the first start point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Grips Editing

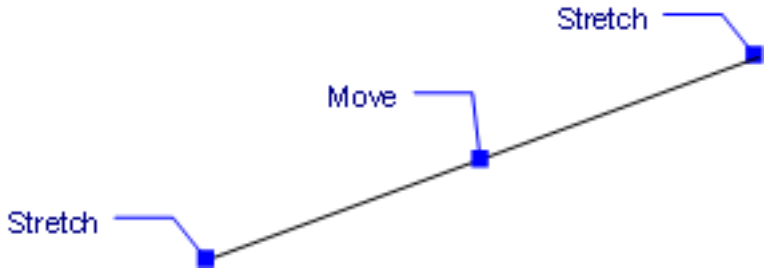
Lines can be edited directly through grips:

1. Select the line. Notice that it has three grips.
2. Drag the grips to perform the following actions

- Drag an endpoint grip to stretch the line.
- Drag the midpoint grip to move the line.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image752.png})

Figure 6.1245



Procedures

- Drawing lines (1.17.9)

Related Commands

MLine (6.21.26) - draws multilines. PLine (6.24.27) - draws connected lines and arcs. Ray (6.26.1) - draws semi-infinite lines. Sketch (6.27.34) - draws sketched (freehand) lines. Trace (6.29.24) - draws wide line segments. XLine (6.34.5) - draws infinite lines. Properties (6.24.49) - changes the properties of lines through a dialog box. Change (6.9.10) - changes the properties of lines through the command bar.

Figure 6.1246

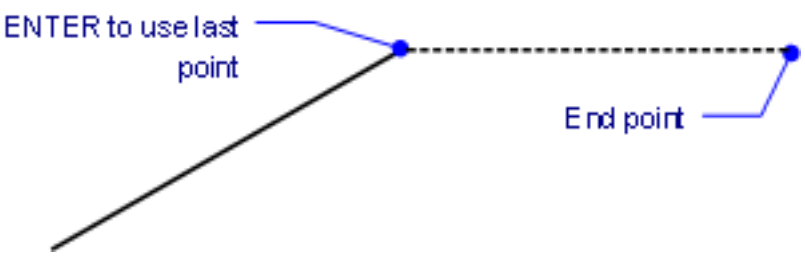


Figure 6.1247

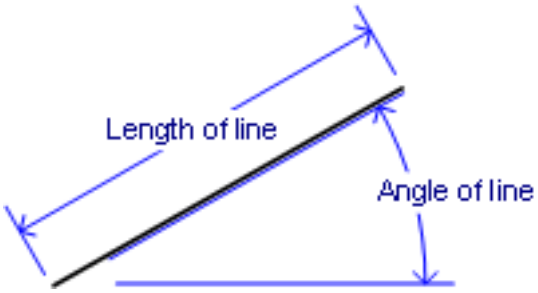


Figure 6.1248

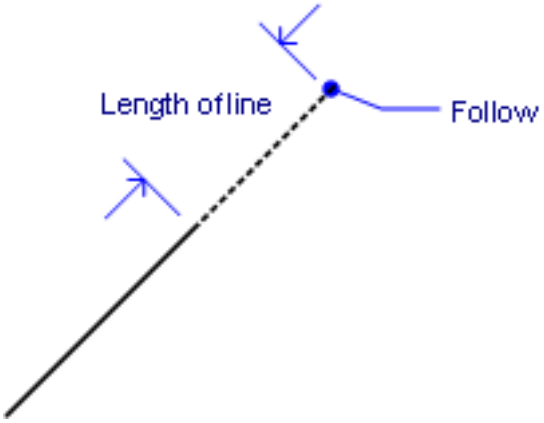


Figure 6.1249

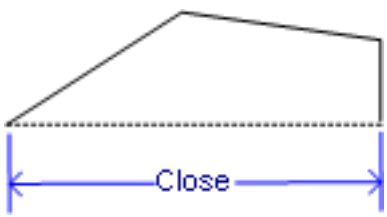


Figure 6.1250



6.20.32 -Linetype

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads, sets, and creates linetypes through prompts in the command bar.

Accessing the Command

command bar: **-linetype** transparent: **'-linetype** alias: **-lt** status bar: **ByLayer** : **-linetype Prompts you in the command bar:** Linetype [? to list/Create/Load/Set]: **(Enter an option, or press Enter to end the command)**

Command Options

Option	Description
? to list	Displays the Select Linetype File dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S 1. Choose a LIN file, such as **iso.lin** 2. Click **Open**. The programlists the names and details of the file's linetype definitions in the Prompt History window, like this: **BORDER Border __ __ . __ __ . __ __ . __ __ . __ __ . 12.700, -6.350, 12.700, -6.350, 0, -6.350**
Create	Creates new linetype definitions; prompts you: 1. **Name for new linetype** - **(enter a name, such as legaleBorder.)** 2. The program opens the Create or Append Linetype File dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou 3. Select a file or enter a new name 4. Click **Open**. The programchecks if the name exists in the file: **One moment... Checking existing linetypes for "legalBorder."** 5. **Linetype description** - **(enter a description of the linetype, such as"Legal border".)** 6. **Linetype definition (positive numbers for lines, negative for spaces)** - **(enter the code that defines the linetypes)**: • **Dashes** - shown by positive numbers, such as 0.5. • **Spaces** - shown by negative numbers, such as 0.25. • **Dots** - shown by zero (0). Separate numbers with commas, such as 0.5, -.25, 0. 7. Press **Enter** to finish the definition. The program adds the definition to the LIN file, and then reports: **Linetype "new" was defined in "C:\...default.lin"**

Load	Loads a linetype definition from a LIN file. Prompts you: 1. **Enter linetype to load - (specify the name of the linetype, such as 'border'.)** 2. The program displays the Select Linetype File dialog box, so choose a LIN file. 3. Click **Open**. The program reports, "Linetype 'BORDER' loaded."
Set	Sets a loaded linetype as the current (working) linetype; prompts you: **? to list linetypes in drawing/Linetype to set as current <BORDER> - (enter an option)**: • **?** -lists the linetype definitions in the drawing. • Enter the name of a linetype • Press **Enter** to accept the default

Status Bar Menu

Right-click **ByLayer** (or other linetype name) on the status bar for the shortcut menu: (\vref {fig:C:/help/enUS/V18/CmdRef/sou (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image524.jpg})

Shortcut Menu	Description
ByLayer, ByBlock, Continuous	Chooses the default linetype.When the drawing contains additional linetypes, their names are listed here.
Properties	Opens the Linetypes section of the Drawing Explorer; see Linetype (6.20.33) command.

Figure 6.1251

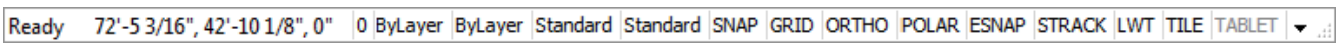
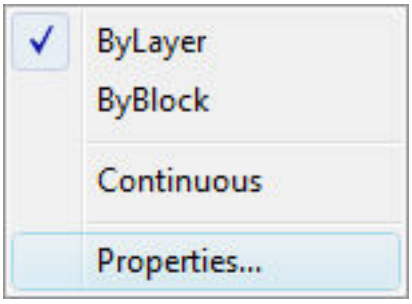


Figure 6.1252



Related Commands

Linetype (6.20.33) - sets and loads linetypes through the Drawing Explorer. LWeight (6.20.41) - specifies lineweights. Settings (6.27.23) - sets linetype parameters. Layer (6.20.2) - assigns linetypes to layers. Properties (6.24.49) - change linetypes through a palette. Change (6.9.10) and ChProp (6.9.18) - change linetypes through the command bar.

Figure 6.1253

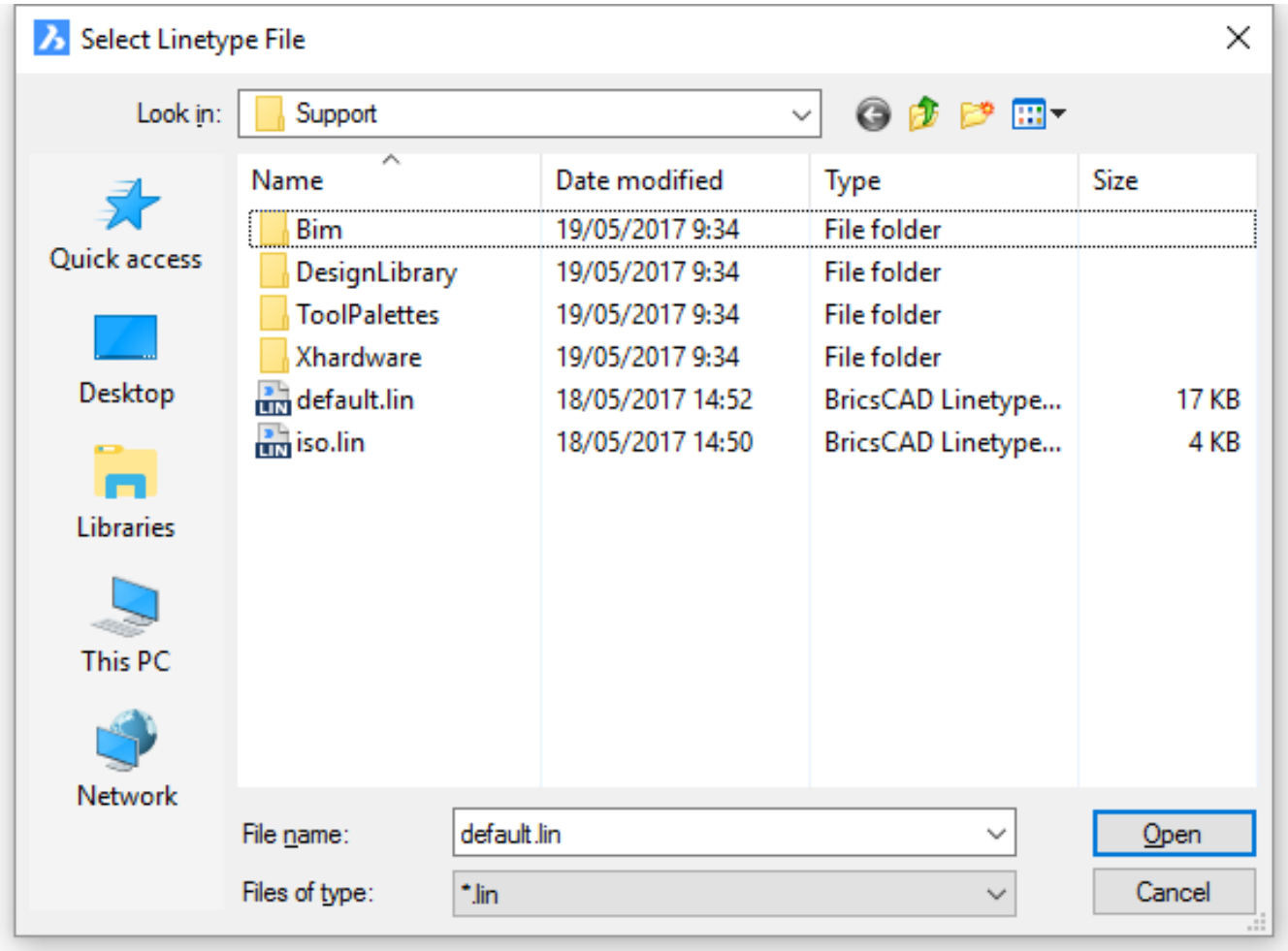


Figure 6.1254

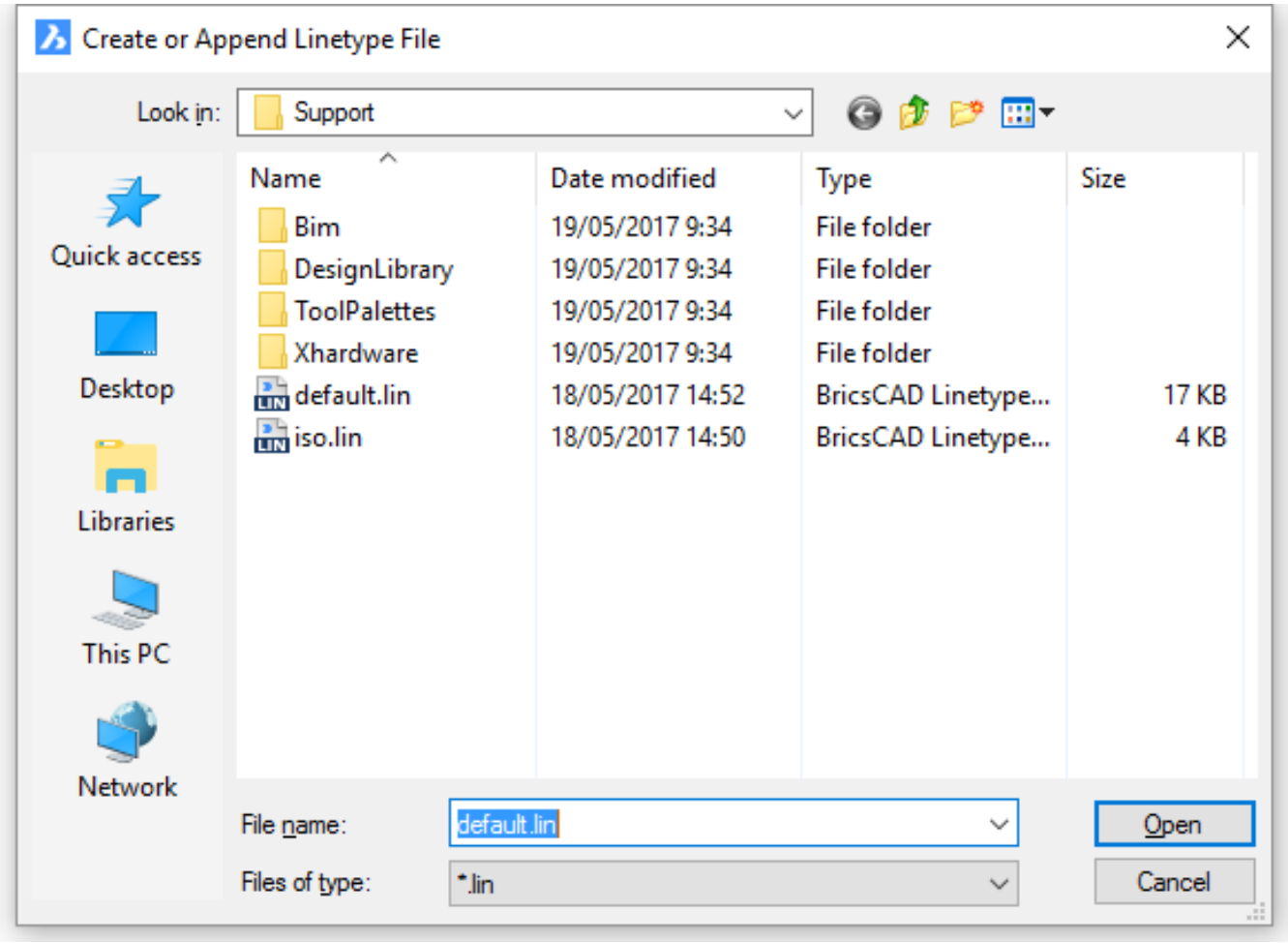


Figure 6.1255

Shape Classic Pro Platinum Mechanical BIM

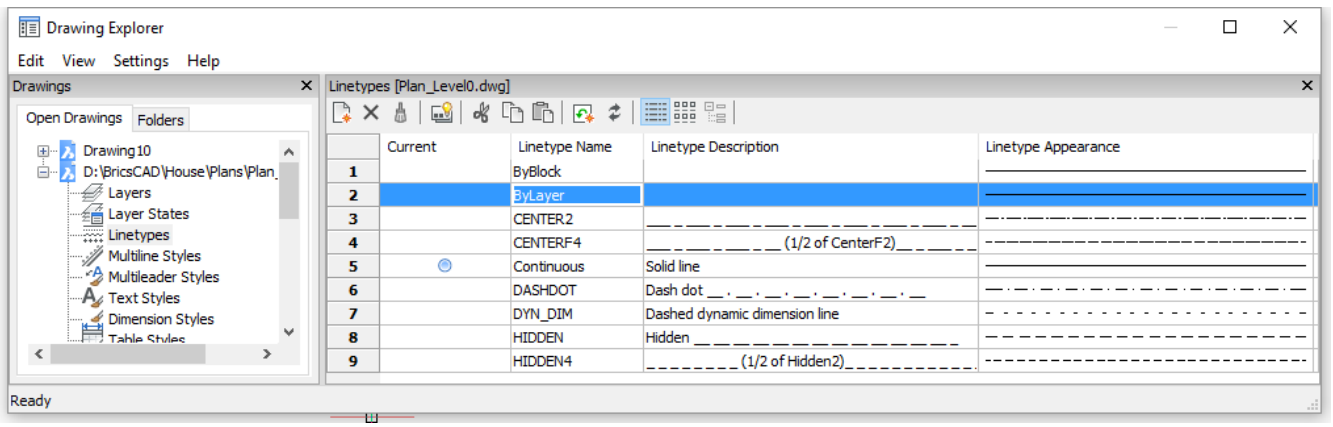
6.20.33 Linetype

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads, sets, and creates linetypes through the Drawing Explorer.

Accessing the Command

command bar: **linetype** transparent: **'linetype** aliases: **lt ddltype expltypes** menu bar: **Tools | Drawing Explorer | Linetypes** status bar: **Linetype**field (”ByLayer”) : **Linetype** Displays the Linetypes node of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DrawingExplorerLi Add linetypes or modify their properties, and then click **X**. Press **F1** to access help. New drawings contain at least these linetype definitions: Continuous, ByLayer, and ByBlock.Any other linetype must be loaded into the drawing before it can be used: click the **New** button to load the linetype. To start new drawings with all linetypes loaded, create and save the drawing as a DWT template file.

Figure 6.1256



Command Options

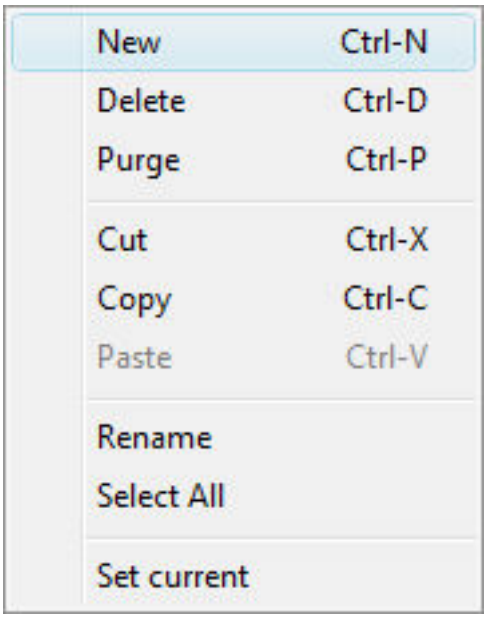
Option	Description
New	Loads additional linetype definitions into the drawing. Displays the Load Linetypes dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L (To add linetypes from a different LIN file, click the **File** button.) 1. Select one or more linetypes. 2. Click **OK**. The linetype(s) are added to the Drawing Explorer's list.
Delete	Erases linetype definitions from the drawing. The following linetype definitions cannot be deleted: • Continuous • ByLayer • ByBlock • Any linetype in use
Purge	Removes all unused linetype definitions from the drawing. The following linetypes cannot be purged: • Continuous • ByLayer • ByBlock • Any linetype in use

Shortcut Menu Options

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image199.jpg}) The keystrokes shown below work only inside the Drawing Explorer:

Option	Keystroke	Description
New	Ctrl+N	Adds a linetype definition by displaying the Load Linetypes dialog box.
Delete	Ctrl+D	Erases the selected linetypes, except for the ones listed above.
Purge	Ctrl+P	Purges unused linetypes from the current drawing, except for the ones listed above.
Copy	Ctrl+C	Copies the selected linetype to the Clipboard. (The Cut option does not work with linetypes.)
Paste	Ctrl+V	Pastes the linetype definition into this or another drawing.
Rename		Renames the selected linetype.
Select All		Selects all linetype definitions.
Set Current		Sets the selected linetype as current.

Figure 6.1257



Status Bar Menu

Right-click ByLayer (or other linetype name) on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image792.png}) (\vref {fig:C:/Dev/Task-2448/images/static

Shortcut Menu	Description
ByLayer, ByBlock, Continuous	Chooses the default linetype.When the drawing contains additional linetypes, their names are listed here.
Properties	Opens the Linetypes section of the Drawing Explorer.

Figure 6.1258

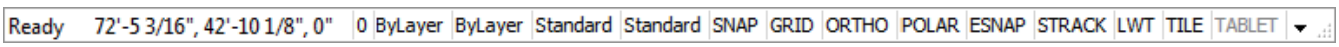
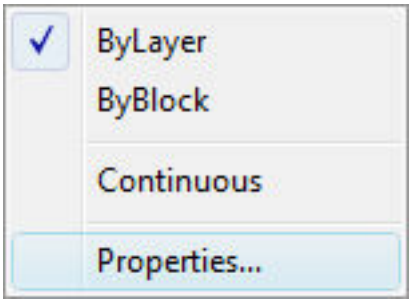


Figure 6.1259



Related Commands

-Linetype (6.20.32) - sets and loads linetypes through the command bar. LWeight (6.20.41) - specifies lineweights. Settings (6.27.23) - sets linetype parameters. Layer (6.20.2) - assigns linetypes to layers. Properties (6.24.49) - change linetypes through a palette. Change (6.9.10) and ChProp (6.9.18) - change linetypes through the command bar. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.1260

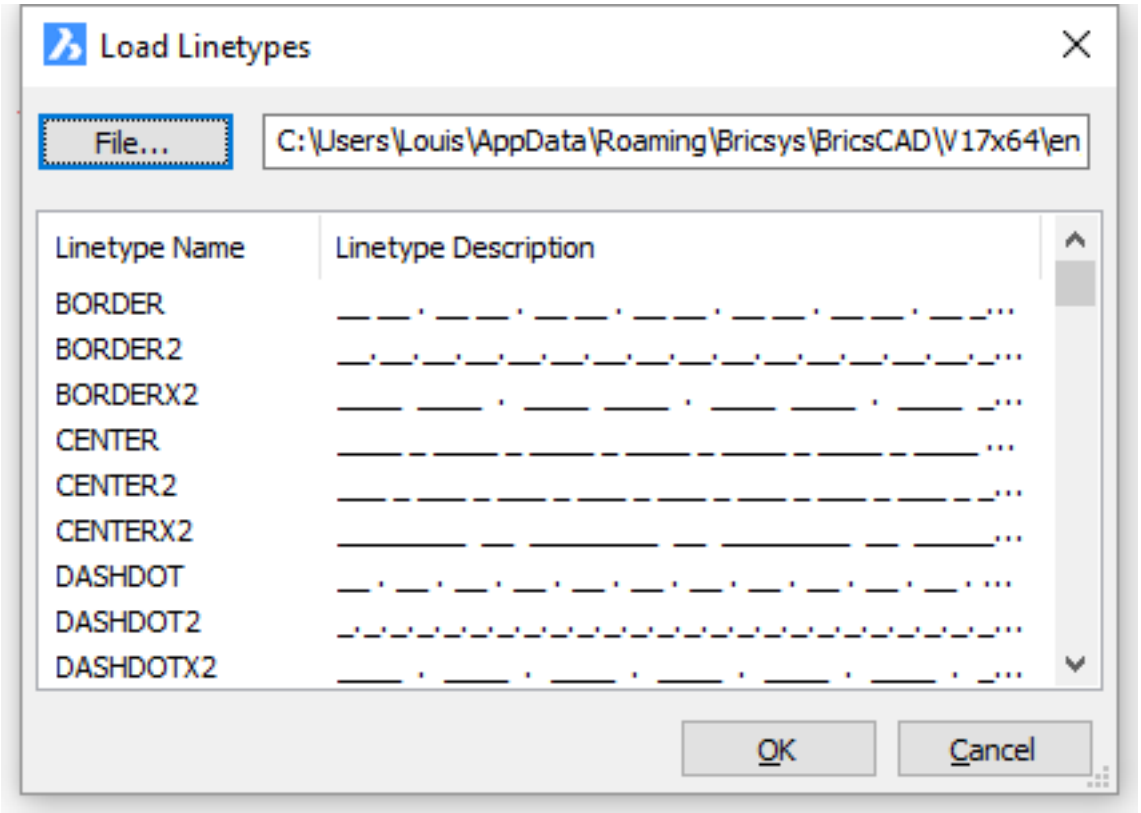
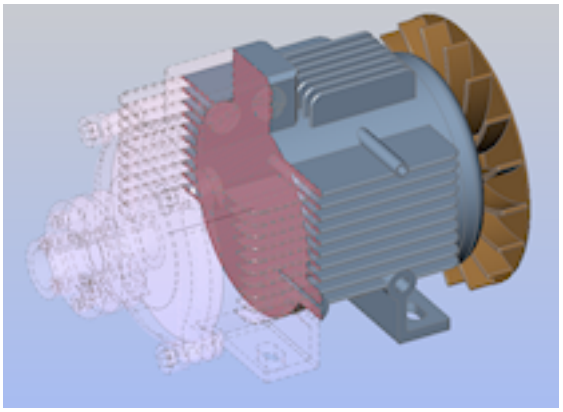


Figure 6.1261



Figure 6.1262



6.20.34 LiveSection

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the Live Section property of a section plane, which exposes the insides of 3D models. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/OrthographicLeft.png})
Notes:

- Before you can use this command, you have to create at least one section plane in the drawing with the SectionPlane (6.27.12) command.
- It is recommended to use the **Clip Display** property instead of Live Section. The Clip Display status can be set for multiple section entities simultaneously.

Accessing the Command


command bar: **livesection** : **LiveSection Prompts you in the command bar:** Select a section entity: **(Click a section entity in the drawing)** When the Live Section property of a section plane was off, it is turned on -- and vice versa.

Command Options

Option	Description
Select a section entity	Selects the section entity for which the plane should be toggled.If the section plane is displayed, then this command turns it, and vice versa.

Related Commands

ClipDisplay (6.9.24) - toggles the Clip Display property of a section plane or a BIM section entity. BimSection (6.7.27) - creates a BIM section entity. BimSectionOpen (6.7.28) - opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing. BimSectionUpdate (6.7.29) - exports or updates the result of a BIM section in a drawing. Section (6.27.11) - creates section planes of 3D solids; the results are region entities. SectionPlane (6.27.12) - creates a section entity that creates sections of 3D solids. SectionPlaneSettings (6.27.13) - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog. SectionPlaneToBlock (6.27.14) - saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

Figure 6.1263



6.20.35 List

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Lists the properties of selected entities. **Note:** This command is largely superseded by the Properties (6.24.49) command.

Accessing the Command

 command bar: **list** aliases: **li, ls** menu bar: **Tools | List Entities** ribbon:**Tools | Inquiry | List Entity Info** toolbar: **Inquiry |**  : **List Prompts you in the command bar:** SOrt/Tracking/<Select entities to list>: **(Choose one or more entities, or enter an option.)** This command returns long lists of data, so press F2 to switch to the Prompt History window. Whenlisting gets too long, press Esc to end it.

Command Options

Option	Description
SOrt	Switches from sequential to sort mode; prompts you: **SEquential/Tracking/<Select entities to list>** - **(enter an option)**: • **SEquential** - lists entities in the order by which you selected them • **Tracking** - specifies the number of lines of text to display before 'Press ENTER to continue' appears; prompts you: **Number of command lines to track <256>** - **(enter a number)** • **Select entities to lis**t - choose one or more entities, and then press **Enter** When you select more one entity, this option lists them sorted by entity. For example, select a line, an arc, and another line. This option lists them as line 1, line 2, and arc 1.

A typical List listing looks like this: **------ Line 0 ---------- Handle: B9 Current space: Model Layer: 0 Color: 256(BYLAYER) Linetype: ByLayer From point: X= -36.0000 Y= 16.5000 Z= 0.0000 To point: X= -24.5000 Y= 11.5000 Z= 0.0000 Length: 12.5399 Angle in UCS XY plane: 337 Delta values: X= 11.5000 Y= -5.0000 Z= 0.0000**

Related Commands

Area (6.4.13) - reports the area, circumference, and perimeter of entities. DbList (6.10.4) - reports parameters of every entity in the drawing. Dist (6.10.31) - reports the 2D and 3D distances between two points. Id (6.17.1) - reports the x,y,z coordinates of picked points. MassProp (6.21.4) - reports the properties of solid models and regions. Properties (6.24.49) - lists the area and other parameters of entities in a palette.

Figure 6.1264



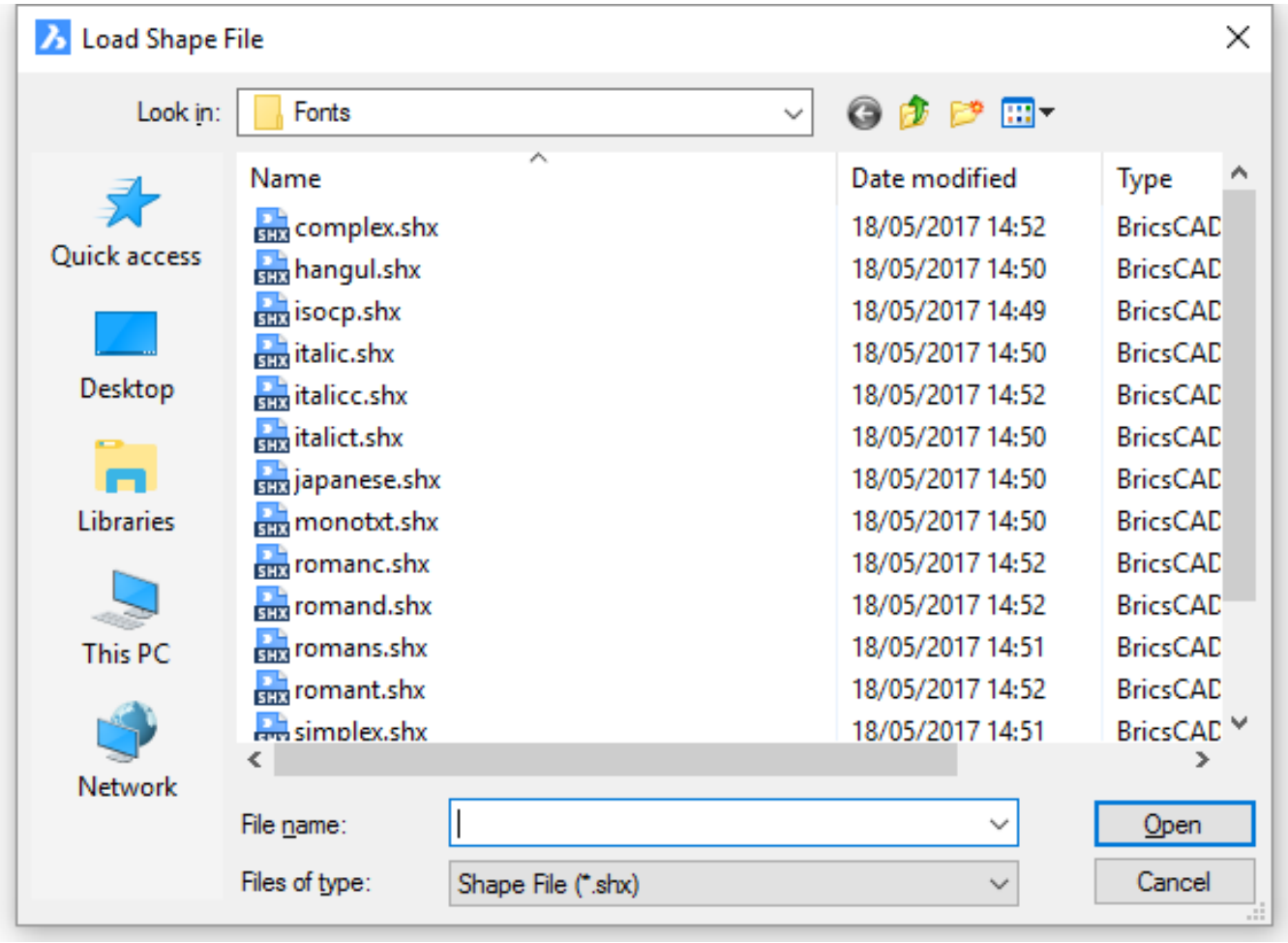
6.20.36 Load

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads SHX shape files into the current drawing. **Note:**This command works only with **shape** SHX files, and not with**font** SHX files. Use the Text and MText commands to work with font SHX files.

Accessing the Command

command bar: **load** : **Load** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cmd
Choose an SHX file, and then click **Open**.Shapes are placed with the Shape (6.27.30) command. SHX files are compiled SHP shape files, which are used to define fonts, shapes, andcomplex linetypes.See the **Customizing BricsCAD** reference guide on creating custom shapes. Shapes are an early version of highly efficient blocks, but are difficult to create. This command remains in BricsCAD for compatibility reasons.

Figure 6.1265



Command Options

No options.

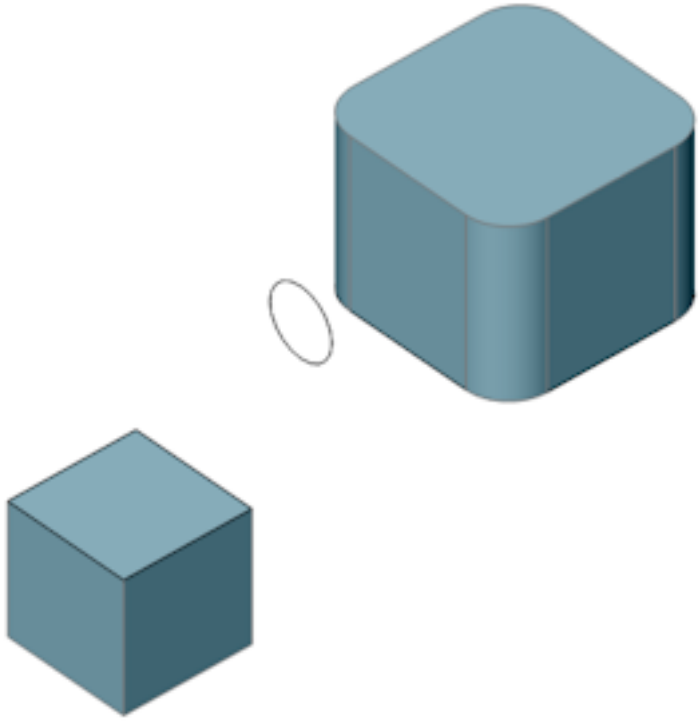
Related Commands

Shape (6.27.30) - inserts shapes into drawings. Insert (6.17.13) - inserts blocks into drawings.

Figure 6.1266



Figure 6.1267



6.20.37 Loft

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Creates 3D solids or 3D surfaces passing through two or more cross sections, as shown below. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/LoftSolidFacesStart.p
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/LoftSolidFacesEnd.png})

Figure 6.1268



Accessing the Command


command bar: **loft** menu bar: **Model | 3D Solids | Loft** ribbon:**Solid | Create | Loft** toolbar: **3D Modeling |**  : **Loft Prompts you in the command bar:** Select cross sections in lofting order [MOde]: **(Select a 2D entity or a planar face of a 3D solid; or choose the Mode option)** Select cross sections in lofting order [MOde]: **(Select a 2D entity or a planar face of a 3D solid; or choose the Mode option)** The first part of the lofted 3D solid is created. Select cross sections in lofting order [MOde]: **(Select more 2D entities or planar faces; or choose the Mode option; or press Enter to stop selecting cross sections)** The the lofted 3D solid is created. 3 cross sections selected Enter an option [Guides/Cross sections only/Settings] <Cross sections only>: **(Press Enterto end the command, or else choose an option)** Examples of cross-sections (left) and resulting 3D surfaces (right) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/LoftProfiles.png}) (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/LoftSolids.png}) Examples of 3D faces used as cross-sections (left) and resulting 3D solid (right): (\vref {fig:C:/Dev/Task-2448/images/staticbri (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/LoftSolidFacesEnd.png}) The program uses cross sections and optionally guidelines to determine the shape of the loft. Cross-sections (at left, in red) specify the shape of the loft at the two ends, and possibly in the mid-sections. The program draws the loft by the most expedient method (right). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/LoftSolidEdgeGuideIntermediate.pn Guidelines specify the shape of the loft between cross-sections. In the figure below, guidelines force the loft along a curve: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/LoftSolidEdgeGuideEnd.png}

Figure 6.1269

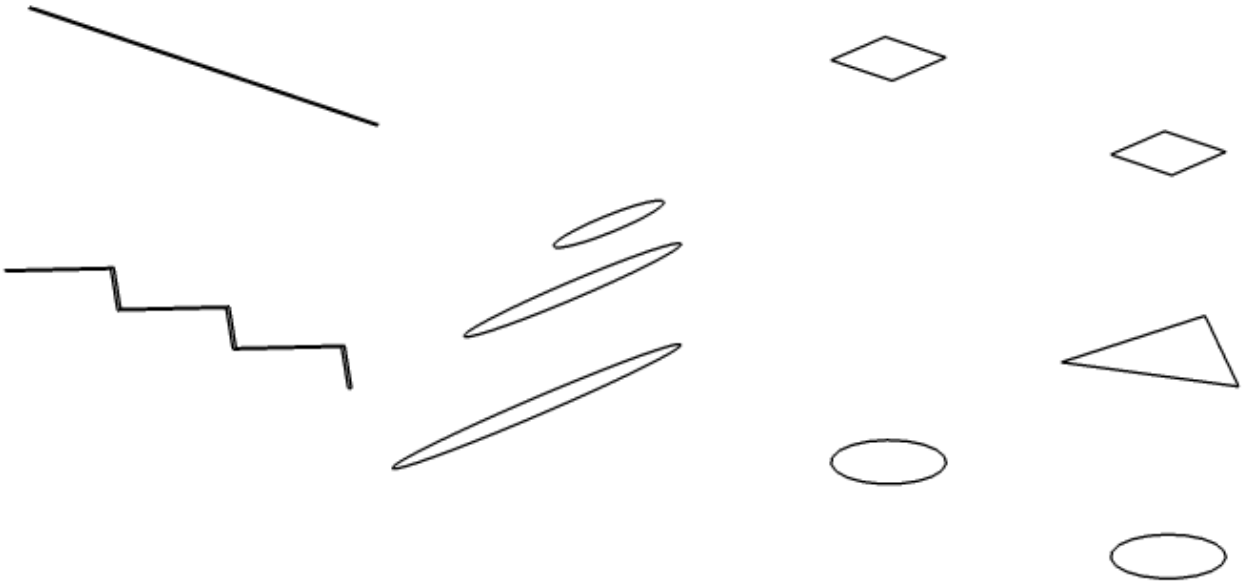


Figure 6.1270

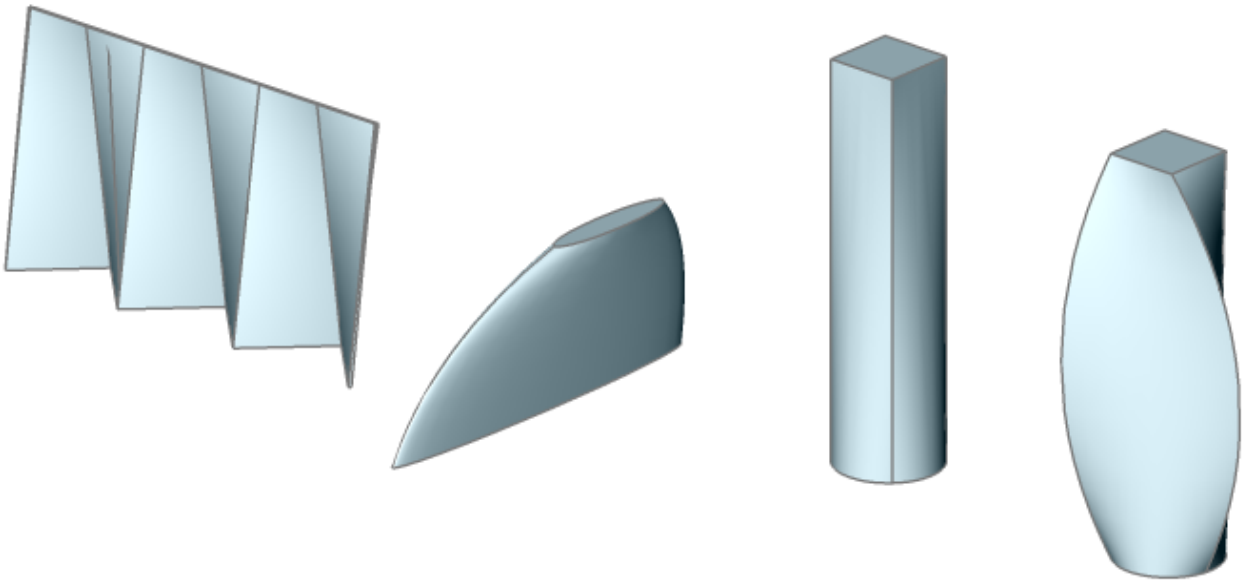


Figure 6.1271

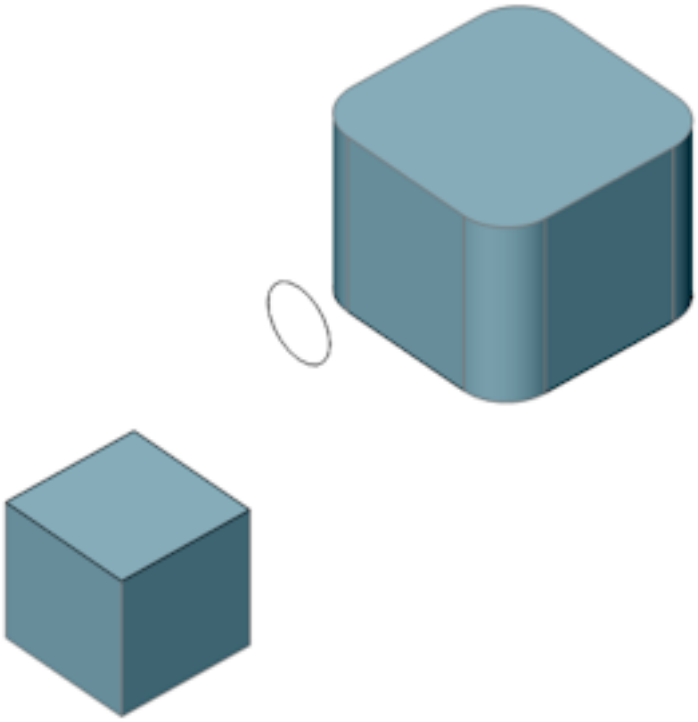


Figure 6.1272



Figure 6.1273

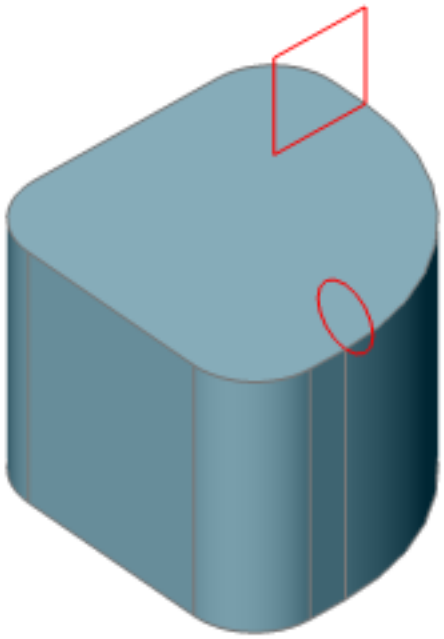


Figure 6.1274

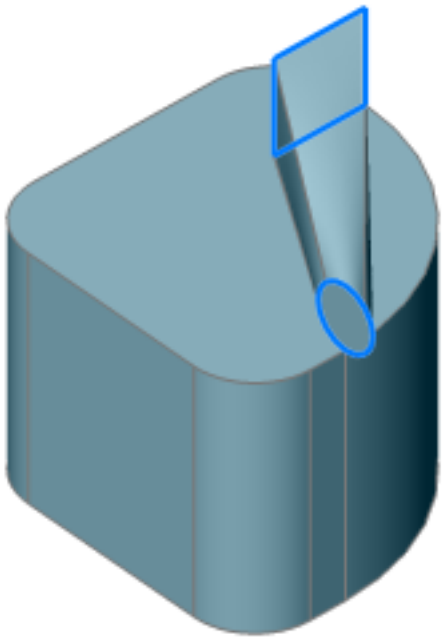
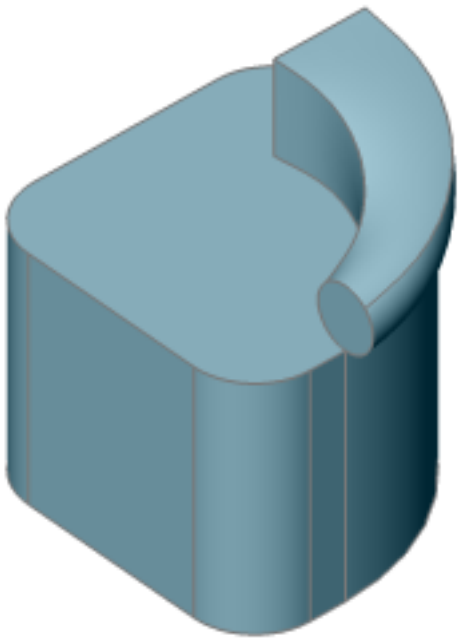


Figure 6.1275



Command Options

Option	Description
Select cross sections in lofting order	Select the entities that represent the cross-section profiles of the loft. Pick any • 2D entity • Planar (flat) face of any 3D solid • Press **Enter** to end entity selection The pick order is important, so keep in mind which one you picked first and so on.
Mode	Determines if the resulting loft is a solid (left) or a surface (right); prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L Surface.png}) **Choose type of created entity [SOlid/SUrface] <SOlid>** - **(enter an option)**: • **SOlid** - Creates lofts as solids • **SUrface** - Creates lofts as surfaces
Cross Sections only	Uses only cross section entities to create the loft, and no guides. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L (WhenDELOBJ = 1 or 2, the selected 2D entities are deleted.)
Guides	Uses both cross-sections (shown in red below) and guide lines (blue) between the selected cross sections to create the loft; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L 1. **Select guide curves** - **(select a planar entity (a line, arc, elliptical arc, splines, single-segment 2D polyline, or edge of a 3D solid).)** 2. **Select guide curves** - **(select more linear entities or else press Enter to carry on.)** (When DELOBJ = 2, the selected guide entities are deleted.)

Settings	Sets variables that affect how the loft is constructed; prompts you: **Current option is Smooth. Enter an option [Ruled/Smooth fit/Normal to/Draft angles] <eXit>** - **(enter an option)**: • **Ruled** - draws straight surfaces between the cross sections; features sharp edges at each cross-section (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • **Smooth fit**- draws smooth surfaces between cross-sections. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • **Normal to** - draws surfaces normal to the cross-sections; prompts you: **Specify normal to [Start/End/Both/All/eXit] <eXit>** - **(enter an option)**: **Start** - loft is normal at the starting cross-section (shown at left, below) **End** - loft is normal at the ending cross-section (middle image) **Both** -loft is normal at the starting and ending cross-sections (at right) **All** - loft is normal at the all cross-sections (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ • **Draft Angles** - specifies the angle at starting and ending cross-sections; prompts you: **Specify [1A to start angle/1M to start magnitude/2A to end angle/2M to end magnitude/eXit] <eXit>** - **(enter an option, or press Enter to return to the previous promtp)**: **Angle -** specifies the angle at which the loft starts from a cross-section **Magnitude** - sets the relative distance of the surface from the cross section in the direction of the draft angle before the surface starts to bend toward the next cross section. **1A** - angle at the starting cross-section **1M** - magnitude at the starting cross-section **1A** - angle at the ending cross-section **2M** - magnitude at the ending cross-section In the figure below,1A = 45° and 2A = 135° (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/Cm In the figure below, both loft angles = 90 degrees (at left), and 0 degrees (right): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L In the figure below, both loft angles = 180 degrees (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L

Tutorial: Making Beautiful Shapes from Lofts

Click here (1.6.28) to watch

Figure 6.1276

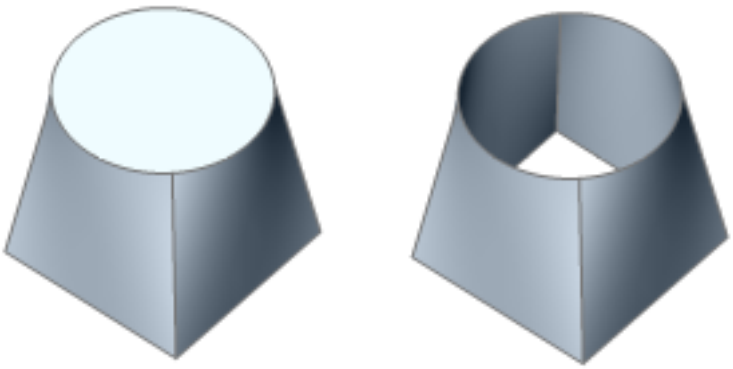
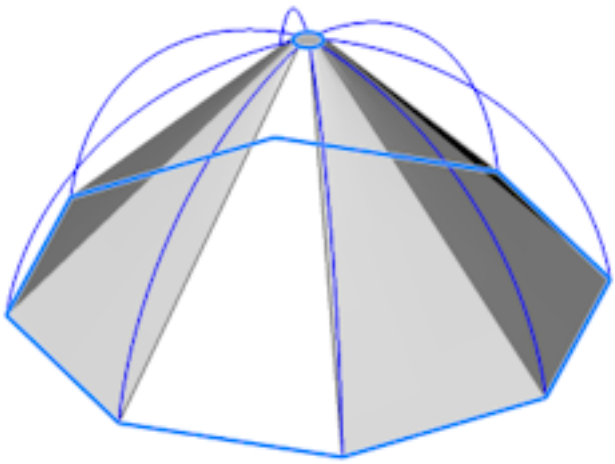


Figure 6.1277



Related Commands

Extrude (6.13.21) - extrudes 2D entities to 3D solids. dmExtrude (6.12.13) - creates a 3D solids by extruding closed 2D entities or regions. dmRevolve (6.12.22) - creates 3D solids by revolution of closed 2D entities or regions about an axis. dmTwist (6.12.32) - modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis. Revolve (6.26.30) - revolves 2D closed entities to turn them into 3D solids. RevSurf (6.26.32) - creates a 3D mesh surface by revolving a linear entity around a line. Sweep (6.27.60) -creates 3D solids or 3D surfaces by sweeping profiles (open or closed 2D entities or 3D subentities) along an open or closed path.

Figure 6.1278

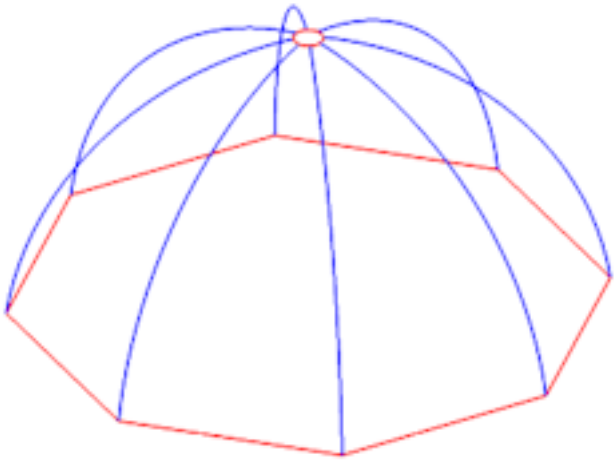


Figure 6.1279

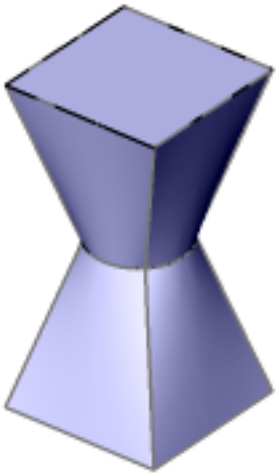


Figure 6.1280

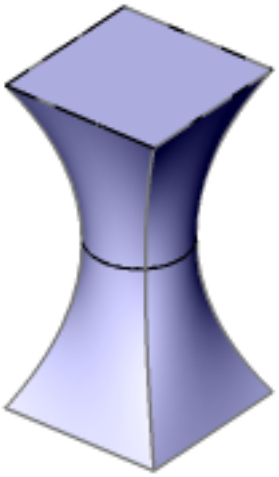


Figure 6.1281

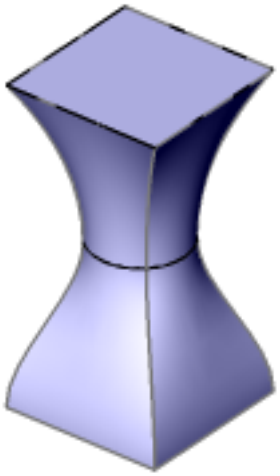


Figure 6.1282

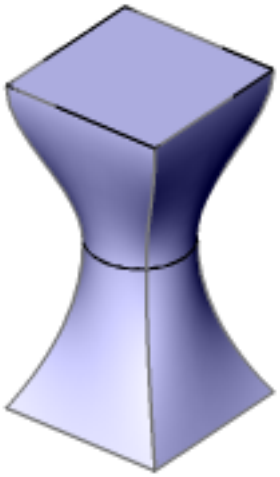


Figure 6.1283

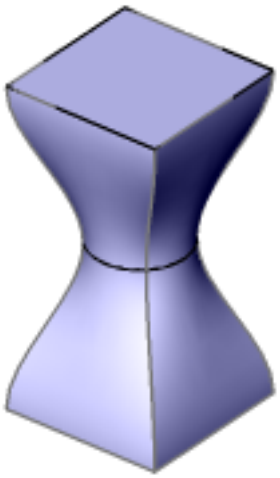


Figure 6.1284

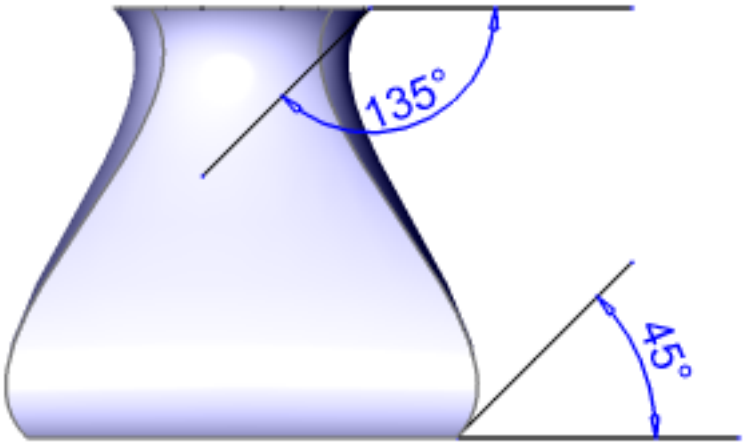


Figure 6.1285

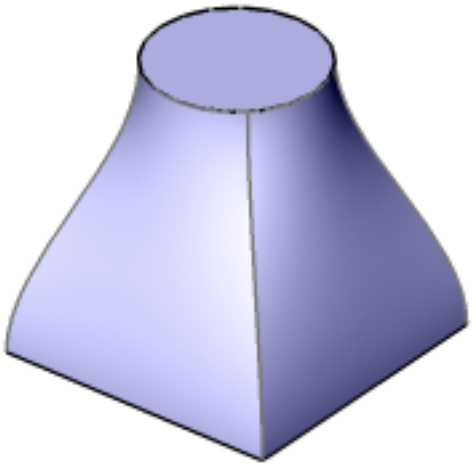


Figure 6.1286

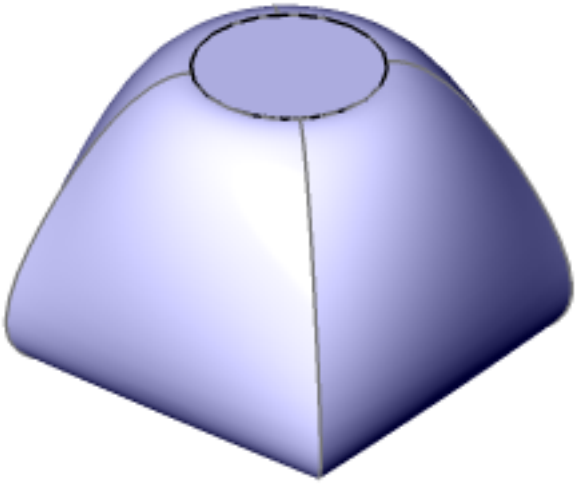


Figure 6.1287

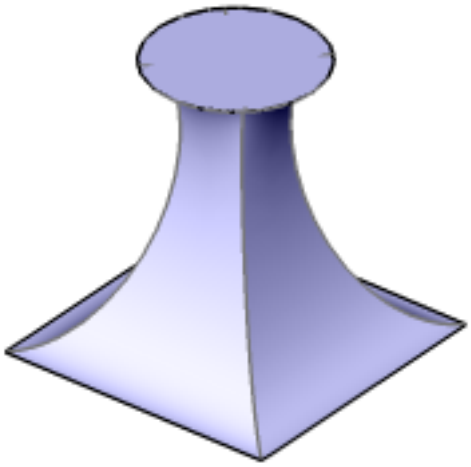


Figure 6.1288



6.20.38 LogFileOff

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Turns off log file recording. **Note:** Log files record all prompts from the program and all of your keyboard input. It does not record mouse actions or other non-keyboard activities.

Accessing the Command

command bar: **logfileoff** : **LogFileOff** No prompts at the command bar. The resulting log file is found in the folder specified by the LogFilePath system variable.The default name of the file is **drawingname_year-month-date_hour-minutes-seconds.log**, such as **drawing1_2029-08-31_08-32-46.log**.You can open the resulting log file in Notepad or another text editor.

Command Options

No options.

Related Commands

LogFileOn (6.20.40) - turns on log file recording. Settings (6.27.23) - sets up log file parameters in the Log Files section of the Settings dialog box. CopyHist (6.9.54) - copies command history from the Text window.

Figure 6.1289



6.20.39 LookFrom

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Controls the display of the Look From widget.



Accessing the Command

command bar: **lookfrom** alias:**navvcube** menu bar:**View | LookFrom** : **LookFrom Prompts you in the command bar:** LookFrom [ON/OFf/Settings] <ON>: **(Choose an option, or press Enter to accept the default)**

Command Options

Option	Description
ON	Displays the Look From widget, which islocated in the position determined by the NAVVIEWCUBELOCATION variable. The default location is the top right corner of the drawing area.  Move the cursor over the Look  From control to activate it:
OFF	Switches off Look From widget.
Settings	Displays the Look From control settings in the Settings dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L

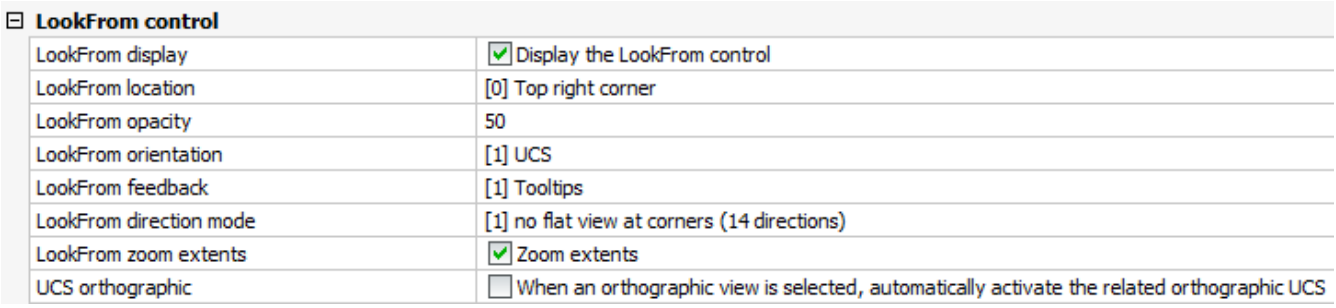
Procedures

- Setting the Look From Control system variables and user preferences (1.16.8)
- Setting the Look From mode (1.16.8)
- Using the Look From control in Isometric mode (1.16.8)
- Using the Look From control in Twist mode (1.16.8)

Related Commands

No related commands

Figure 6.1290



LookFrom control	
LookFrom display	☑ Display the LookFrom control
LookFrom location	[0] Top right corner
LookFrom opacity	50
LookFrom orientation	[1] UCS
LookFrom feedback	[1] Tooltips
LookFrom direction mode	[1] no flat view at corners (14 directions)
LookFrom zoom extents	☑ Zoom extents
UCS orthographic	☐ When an orthographic view is selected, automatically activate the related orthographic UCS

Figure 6.1291



6.20.40 LogFileOn

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Turns on log file recording. **Note:** Log files record all prompts from the program and all of your keyboard input. It does not record mouse actions or other non-keyboard activities.

Accessing the Command

command bar: **logfileon** : **LogFileOn** No prompts at the command bar. BricsCAD records all command text to the LOG file located in the folder specified by the LogFilePath system variable.The default name of the file is **drawingname_year-month-date_hour-minutes-seconds.log**, such as **drawing1_2029-08-31_08-32-46.log**. Use the LogfilePath variable to remind yourself of the location of the log file.

Command Options

No options.

Related Commands

LogFileOff (6.20.38) - turns off log file recording. Settings (6.27.23) - sets up log file parameters in the Log Files section of the Settings dialog box.

Figure 6.1292



6.20.41 LWeight

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Controls the display of lineweights in the drawing, through the Settings dialog box.

Accessing the Command

command bar: **lweight** toolbar: **Entity Properties** |(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cm status bar: **LWT** : **LWeight** Displays the Lineweights section of the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SettingsLineWeights.png}) Change settings, and then click **X**. Lineweights are be specified by the following commands:

- The Layer (6.20.2) command
- The droplist in the Entity Properties toolbar
- The droplist in the Properties (6.24.49) palette

Figure 6.1293

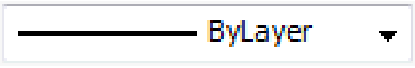
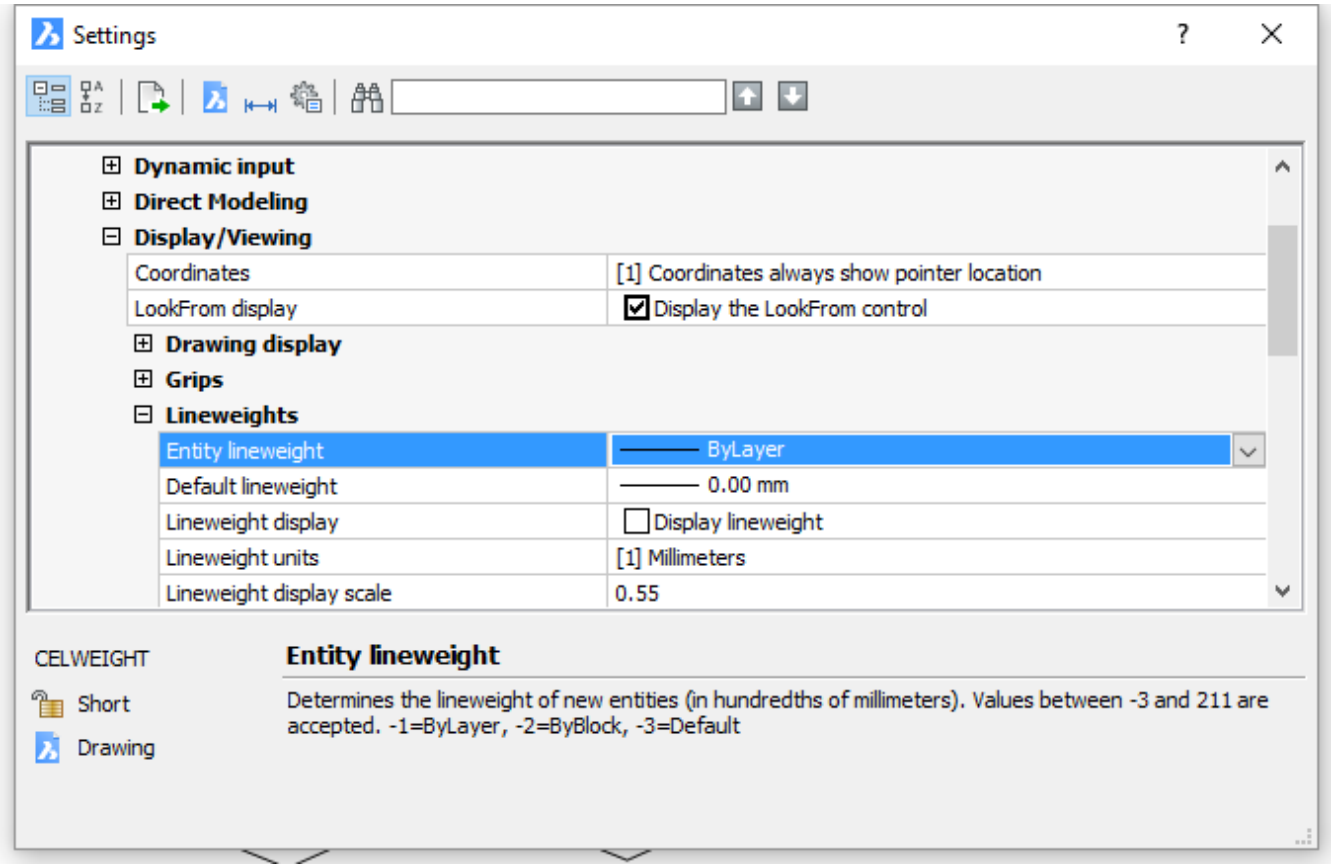


Figure 6.1294



Command Options

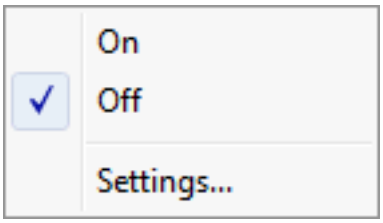
Option	Description
Entity Lineweight	Specifies the lineweight for entities drawn from now on; choose a preset lineweight from the droplist. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i This sets the current lineweight in the Entity Properties droplist and Properties palette.Lineweights cannot be customized, unlike linetypes.
Default Lineweight	Specifies the default lineweight for all new drawings; choose a preset lineweight from the droplist.
Lineweight Display	Toggles the display of lineweights: • **On** - displays lineweights • **Off** - does not display lineweights; all entities have the same lineweight
Lineweight Units	Specifies the unit of measurement: • **Millimeters** - measures the width in millimeters • **Inches** - measures the width in inches
Lineweight Display Scale	Specifies the relative scale of lineweights; applies to lineweights in Model tab only

Status Bar Menu

Right-click **LWT** on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image791.png})

Shortcut Menu	Description
On	Turns on lineweights

Figure 6.1296



Related Commands

Settings (6.27.23) - sets lineweight parameters. Layer (6.20.2) - assigns lineweights to layers. Properties (6.24.49) - change lineweight through a palette. Change (6.9.10) and ChProp (6.9.18) - change lineweight through the command bar. Plot (6.24.26) and Print (6.24.46) - optionally plot entities with lineweights.

Figure 6.1297



6.21 M

6.21.1 Mail

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Attaches the current drawing to a new message with your computer's default email client.

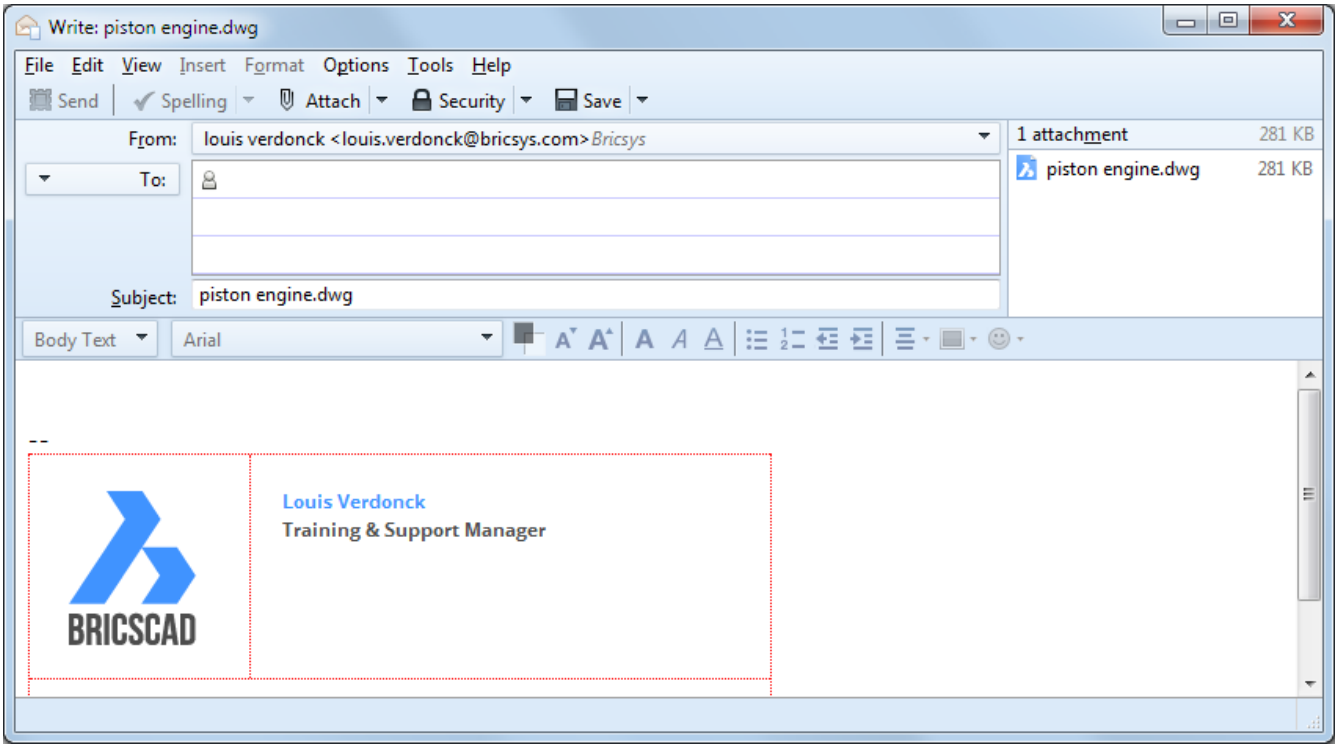
Figure 6.1298



Accessing the Command

command bar: **mail** menu bar: **File | Send Mail** : **Mail** Launches the email client software, starts a new message, and attaches the current drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/MailDi
Complete the message, and then click **Send**.

Figure 6.1299



Command Options

No options.

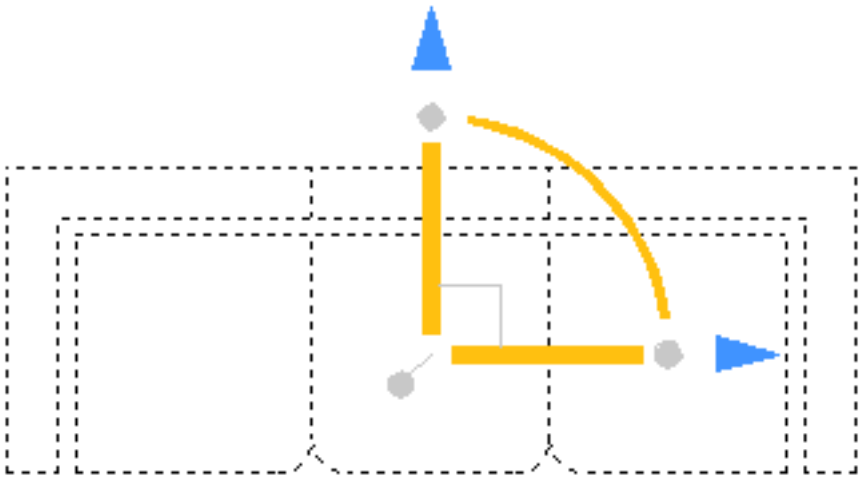
Related Command

eTransmit - packages drawing files and their support files for transmission by email and other methods.

Figure 6.1300



Figure 6.1301



6.21.2 Manipulate

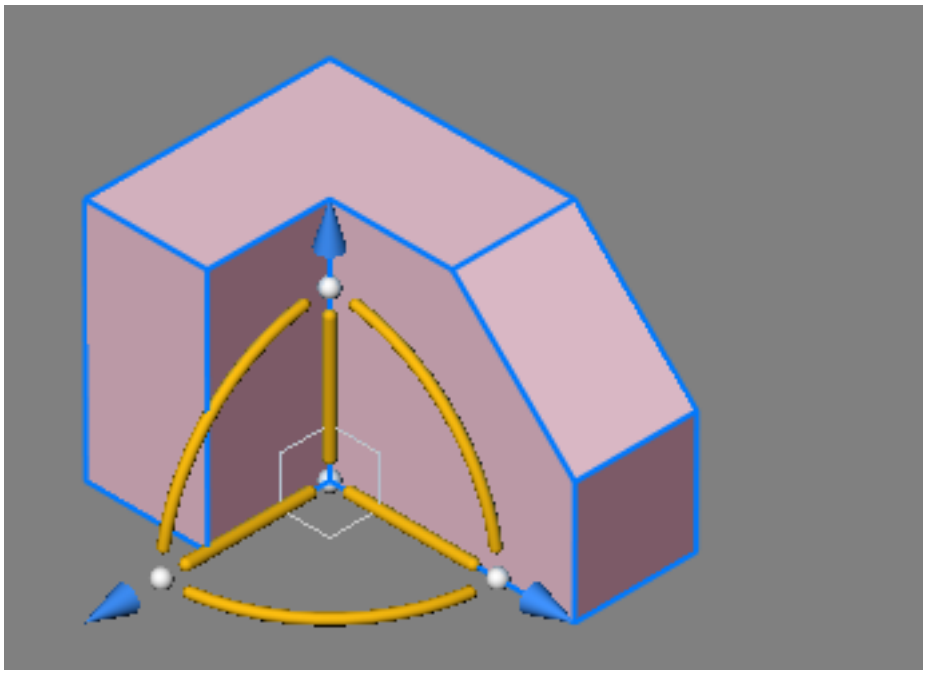
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Launches the manipulator widget to rotate, move, copy, mirror, and/or scale 2D entities and 3D models. **Note:** The manipulator widget appears automatically any time you select an entity, when the MANIPULATOR variable is set to 1 or 2. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/ManipulatorStart2D.png})

Accessing the Command


command bar: **manipulate** quad: **Modify | Manipulate** : **Manipulate Prompts you in the command bar:** Select entities: **(Select one or more entities)** Entities in set: 1 Select entities: **(Select more entities, or else press Enter)** The manipulator widget appears near the selected entities: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en The manipulator widget stays visible until you press **Esc**. **Note:**You don't need to use the Manipulate command to turn on the widget; it appears automatically with the MANIPULATOR variable:

- **0**- manipulator widget is displayed only when you enter the Manipulate command
- **1**- manipulator widget is displayed automatically each time you select an entity
- **2**- widget is displayed automatically when you hold down the left mouse buttonlonger than the value of the MANIPULATORDURATION variable (default = 250 milliseconds, a quarter-second)

Figure 6.1302



Command Options

Option	Description
Drag an axis	Moves or copies the selection along the axis being dragged: 1. Place the cursor over one of the axes. 2. **To move the entity** - **(click, and then move the cursor to move the selection along the axis.)** **To copy the entity**- **(hold down the Ctrl key and click, then move the cursor to copy the entity along the axis.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Manipu To complete the move or copy operation, do one of the following actions: • Pick a point in the drawing • If dynamic input (1.15.9) is on, type a distance into the distance field • Eyeball the distance with the ruler (see figure above), and then click

Drag a plane	Moves or copies a plane. Place the cursor over one of the plane swatches. The swatch under the cursor highlights. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Manipu Do one of the following: • Click when the swatch of the desired plane highlights. The selection moves along with the cursor in the selected plane. • Pick a point • IfDynamic Input (1.15.9) is ON, type a distance in the dynamic entry fields. Hit the TAB-key to switch between the entry fields; press Enter to confirm the current values.
Drag an arrowhead	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Manipu Mirrors, moves or scales the selection. Right click when the cursor is over the manipulator to display a context menu. Choose **Arrowhead act as** , then select the desired action: 3D Scale or 3D Mirror. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/M To mirror* the selection: Place the cursor over an arrowhead. The Mirror swatch displays in a plane through the selected axis and perpendicular to the mirror plane. The selection will be mirrored about the plane that is perpendicular to the axis of the selected arrowhead. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Manipu Click when the desired mirror plane is selected. The selection is mirrored dynamically when you move the cursor. Do one of the following: • Pick a point. • If Dynamic Input (1.15.9) is on, type a distance in the dynamic entry field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/M To scale* the selection: • Place the cursor over an arrowhead. The arrowheads are doubled with a large arrowhead. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/M • Click to start scaling the selection; you are prompted in the command bar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/M *** MANIP_SCALE *** New scale factor [Reference]: Do one of the following: • Pick a point. • Type a scale factor at the command prompt. • Type R ; prompts you: **Enter reference length - (use entity snaps to pick two points in the model.) New length [Reference] - (type the desired distance between the picked points.)**

Drag a rotation arc	Rotates* the selection about one of the manipulator axes. Place the cursor over one of the three rotation arcs. The rotation arc under the cursor expands and the rotation axis displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/Manipu Click when the desired rotation axis displays. The selection rotates dynamically when you move the cursor. Do one of the following: • Pick a point. • If Dynamic Input (1.15.9) is on, type a distance in the dynamic entry field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/M

* To copy the selection set, keeping the original entities, do one of the following:

- Press and hold the Ctrl-key when starting the action.
- When dragging an axis or plane:
  - Type c then right click to create one or more copies. Right click to stop.
  - Type r then right click to create multiple copies at equal intervals. Click to define the interval, then move the cursor and click when the desired number of copie displays.

Procedures

- To define the manipulator settings (1.23.10)
- The manipulator layout (1.23.10)
- To control the location and orientation of the manipulator (1.23.10)
- To relocate the manipulator (1.23.10)

Related Commands

Copy (6.9.50) - copies entities. dmMove (6.12.17) - moves solids, faces or edges of a solid, or inserts using a vector. dmRotate (6.12.24) - rotates faces of a solid around an axis. Move (6.21.30) - moves entities. Rotate (6.26.34) - rotates entities in 2D. Rotate3D (6.26.35) - rotates entities about an axis in 3D space. Scale (6.27.6) - resizes entities.

Figure 6.1303

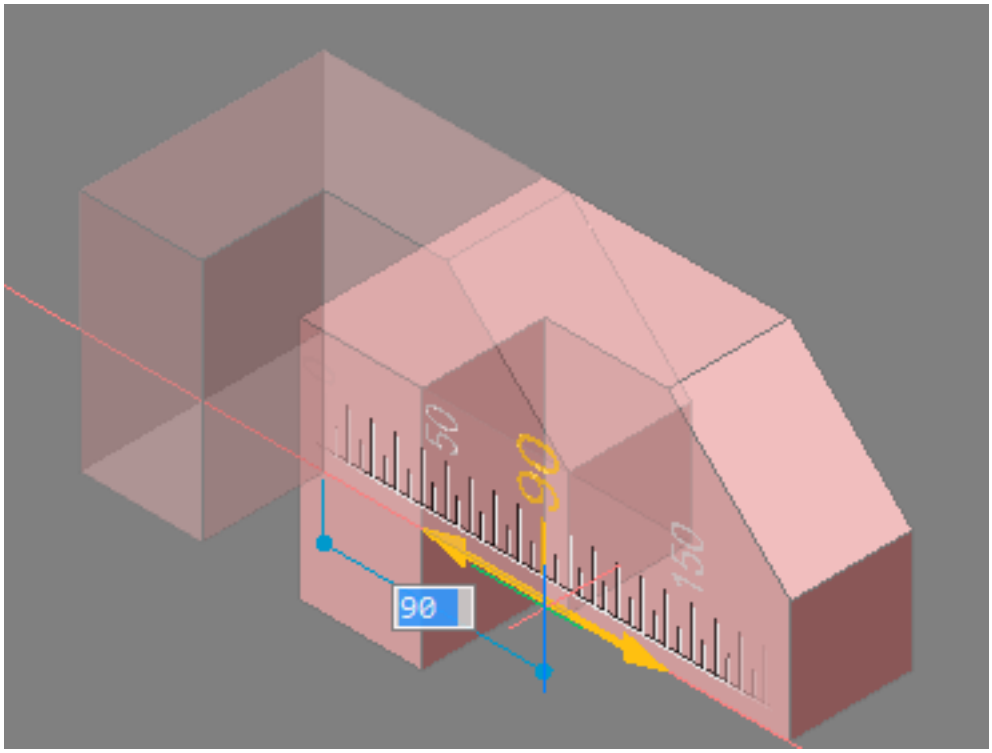


Figure 6.1304

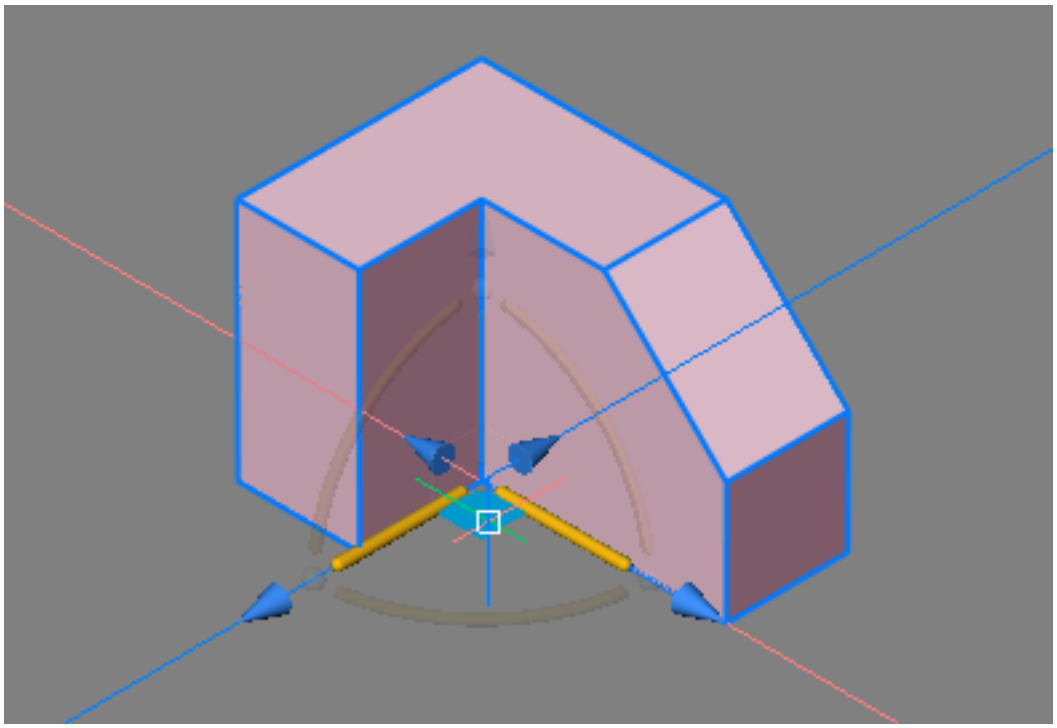


Figure 6.1305

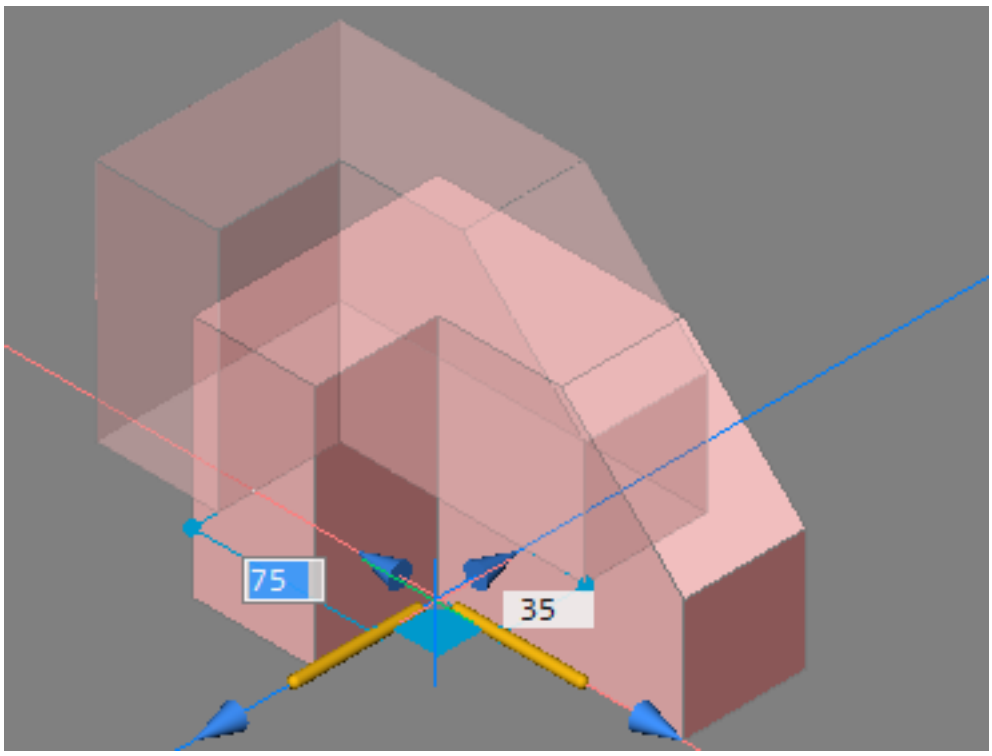


Figure 6.1306

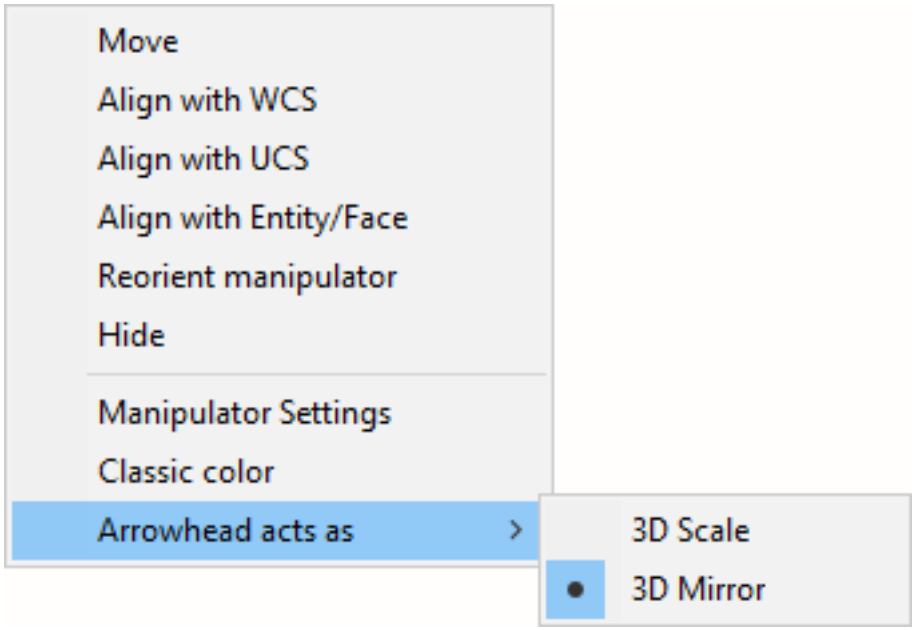


Figure 6.1307

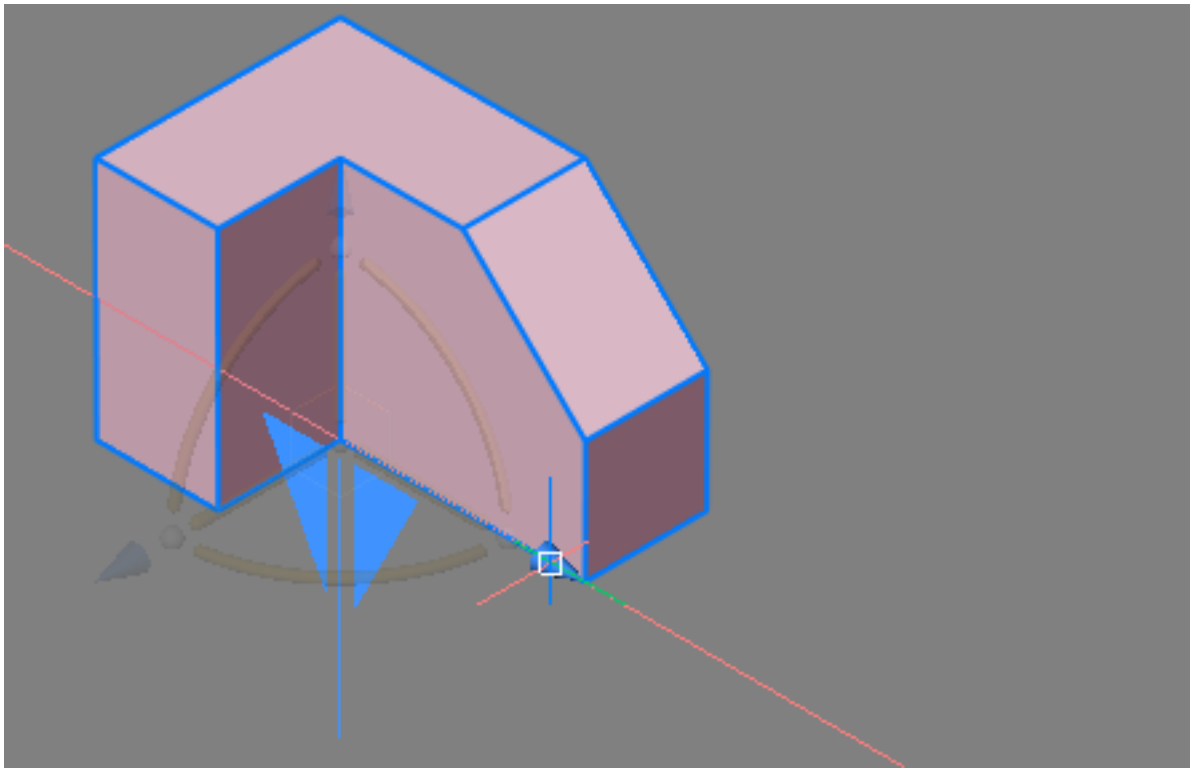


Figure 6.1308

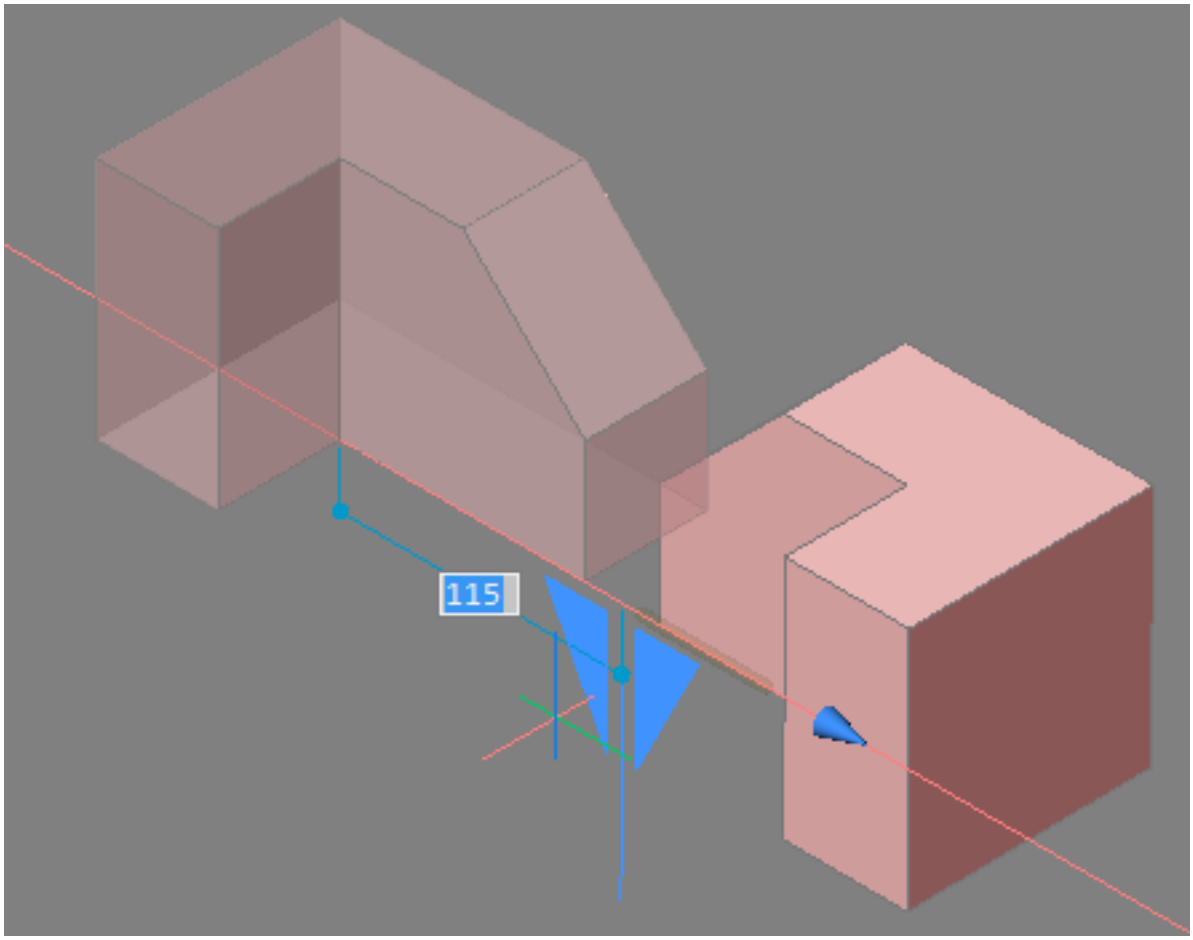


Figure 6.1309

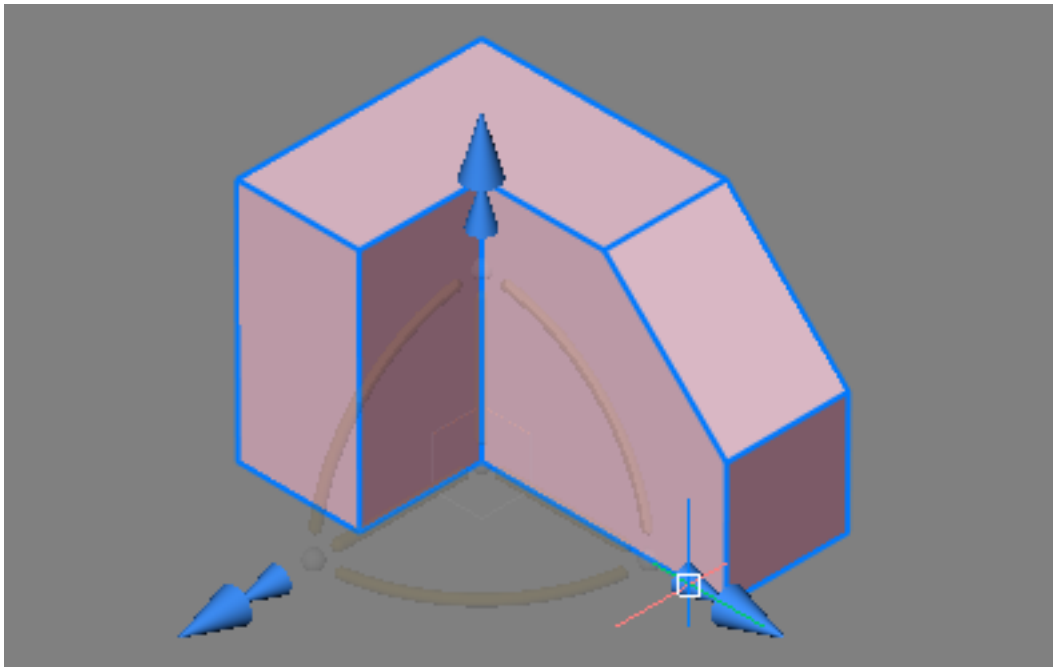


Figure 6.1310

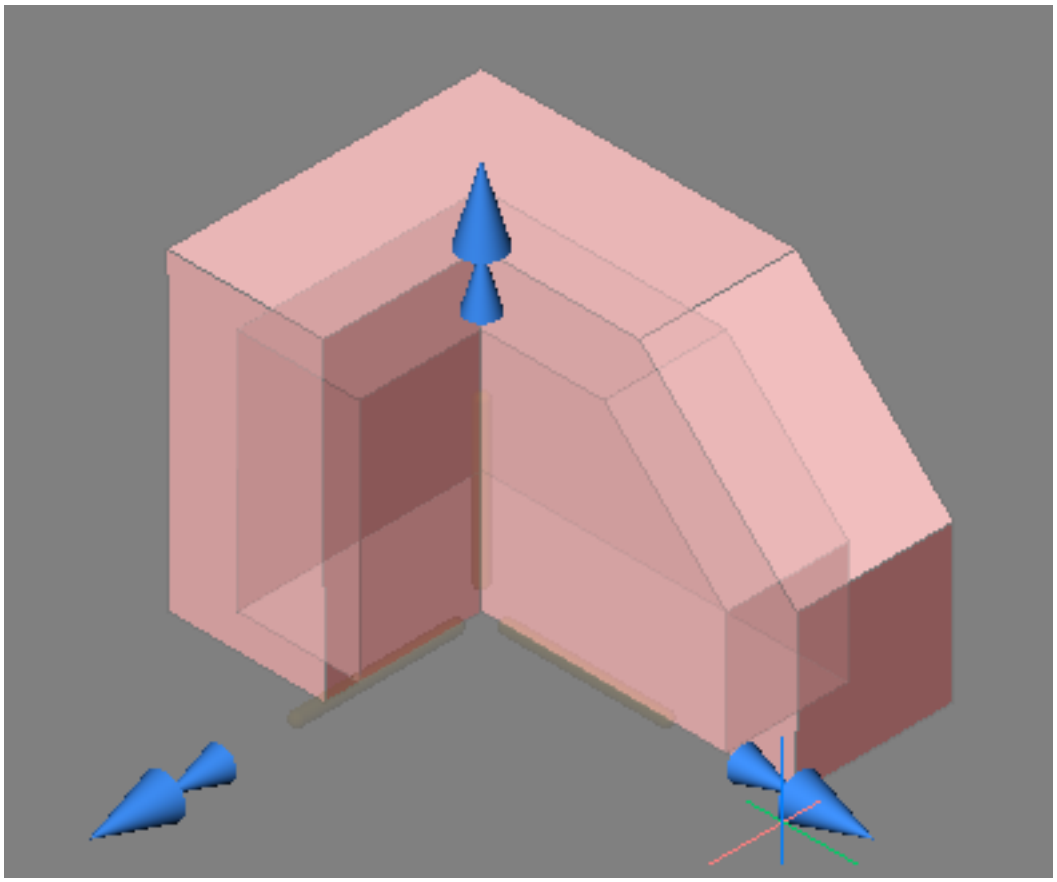


Figure 6.1311

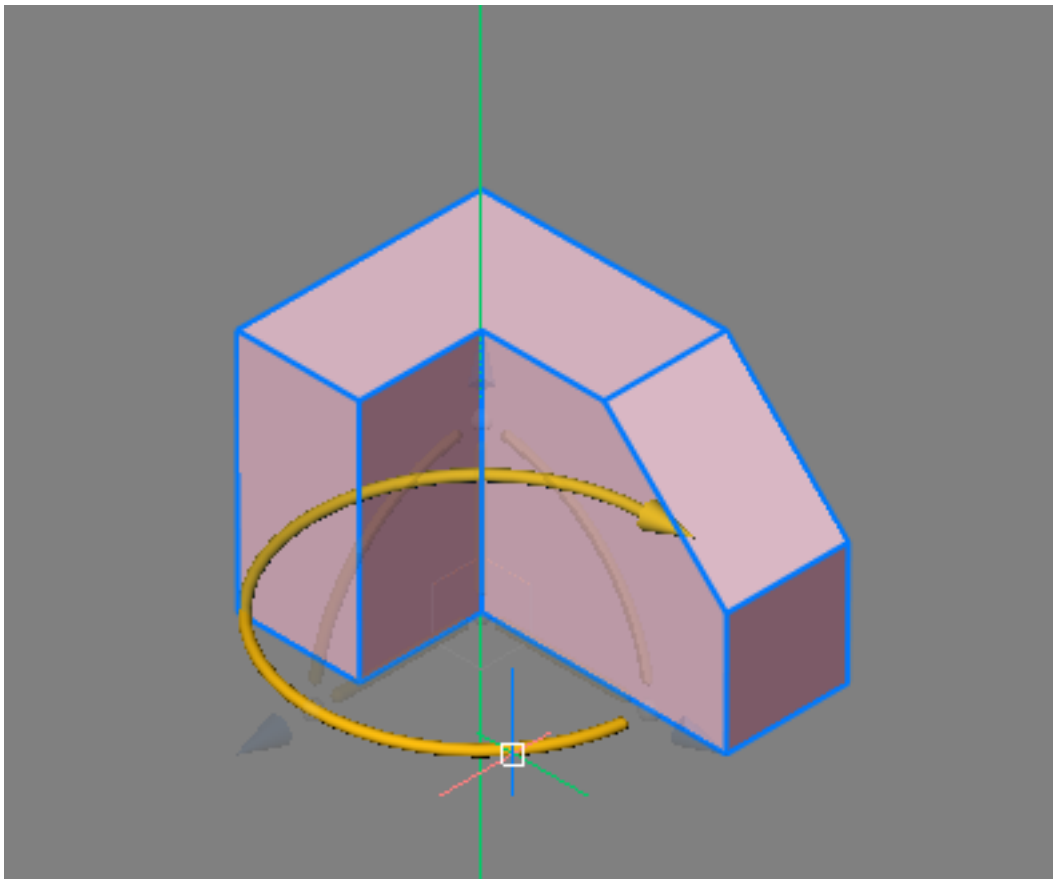


Figure 6.1312

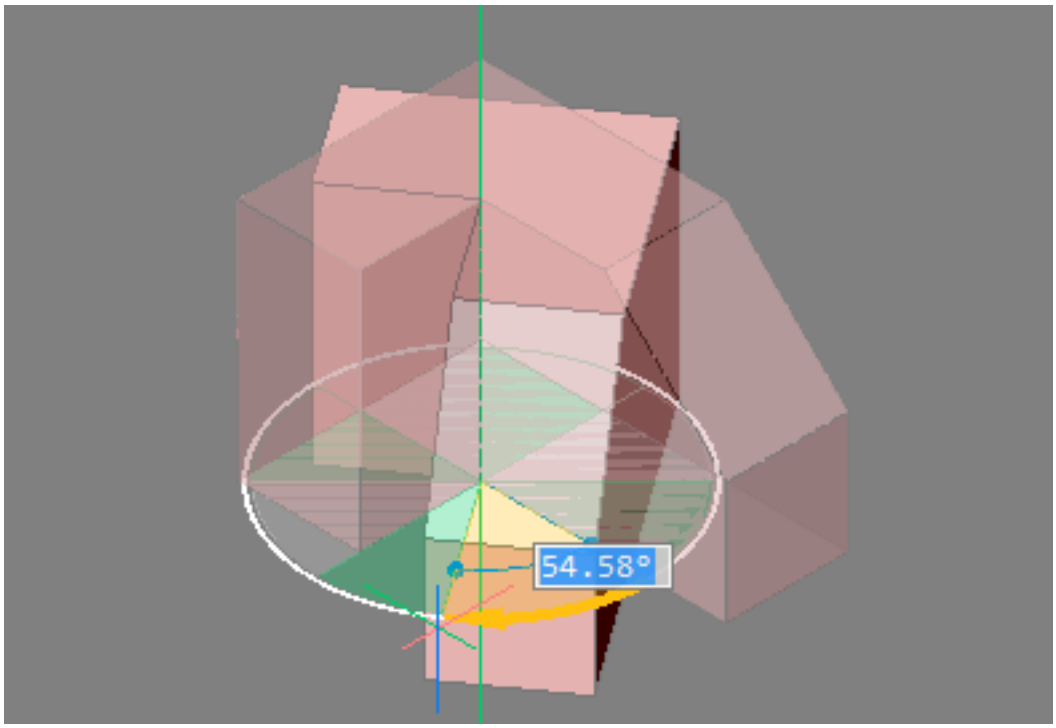


Figure 6.1313



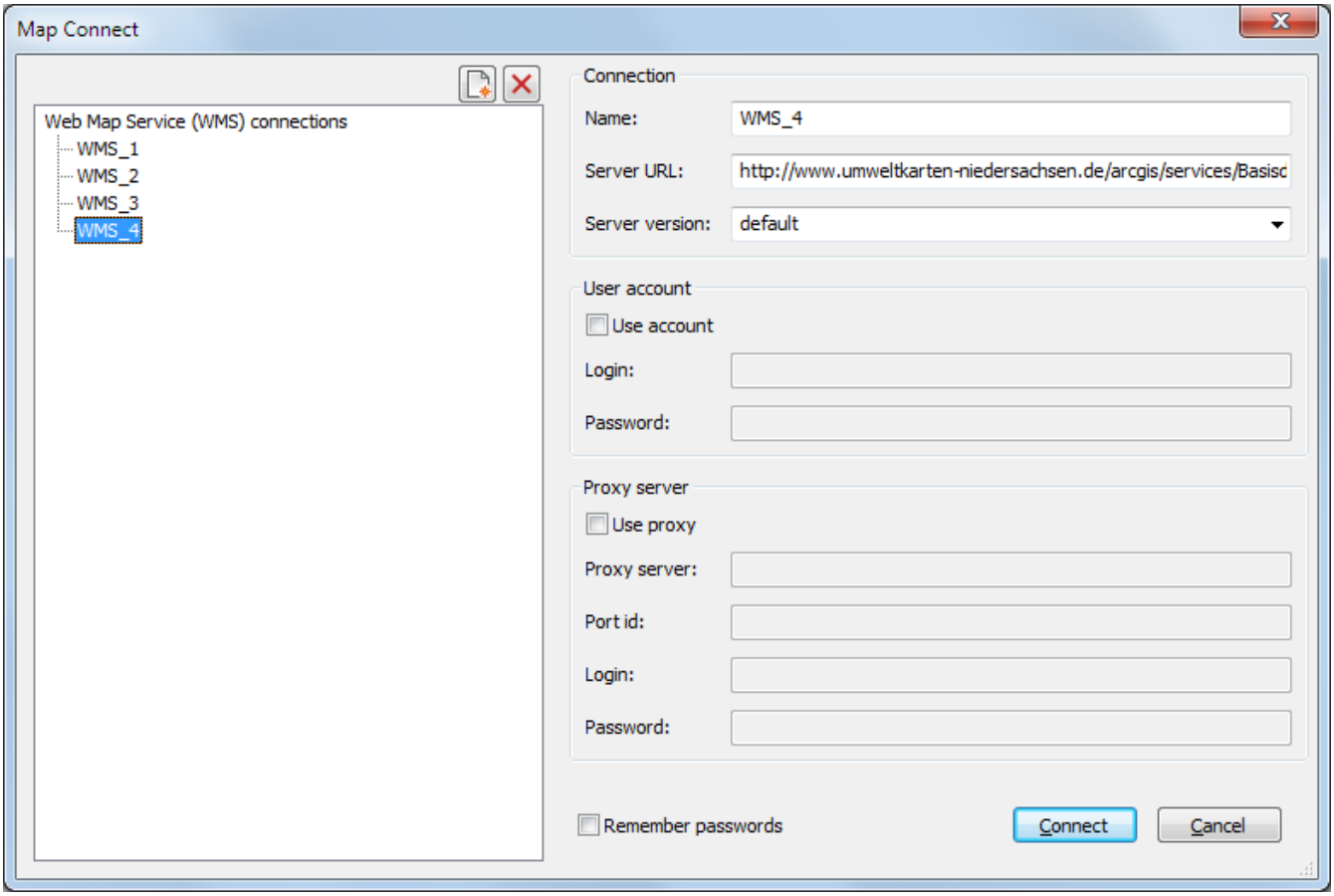
6.21.3 MapConnect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Allows to set up a connection with a Web Map Service through a dialog box. **Note:** Before adding map data to a drawing with this command, you must run the GeographicLocation (6.15.15)command to define the geographic location of the drawing.

Accessing the Command

command bar: **mapconnect** : **MapConnect** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

Figure 6.1314



Command Options

Option	**Description**
Name	(Optional) Names the connection. Type a name to replace the default WMS<x> name.The ten most recently used connections are saved.
Server URL	Specifies the URL of a Web Map Service.
Server version	Specify a server version, or else select one from the droplist.

User Account Option	**Description**
Use Account	• **Off** - no login or password needed • **On** - the Web Map Service need a valid login and password to access it
Login	Specifies the login information
Password	Specifies the password

Proxy Server	**Description**
Use Proxy	• **Off** - a proxy server is not being used • **On** - specify a proxy server to connect to the Web Map Server
Proxy server	Specifies the URL of the proxy server
Port Id	Specifies the port to use for the connection
Login	Specifies the login for the proxy
Password	Specifies the password for the proxy lognin

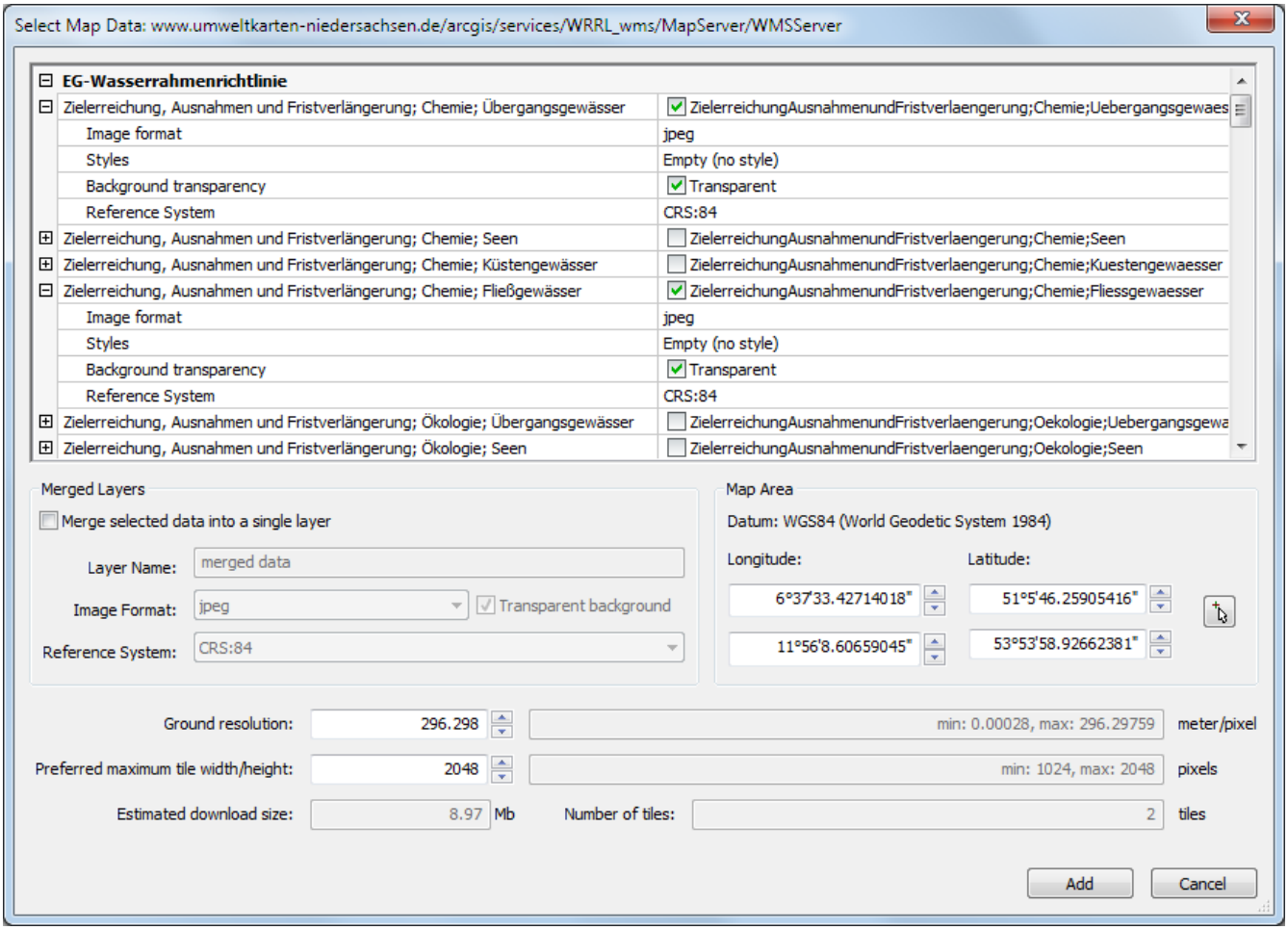
Remember passwords	• **Off** - the program does not remember your passwords for greater security • **On** - passwords will be remembered for future use
Connect	Connects to the map server and then displays the Select Map Data dialog box

A dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SelectMapCor
Do the following:

1. Select one or more data layers.
2. Specify the desired ground resolution and the preferred maximum dimensions of a tile.
3. An estimate of the download size displays in the **Estimated download size** field.
4. Press the **Add** button.

The program stores downloaded map tiles as regular image files in the folder in which the drawing is saved. The definition of the supported Coordinate Reference Systems (CRS) is stored in the new **geodatabase.xml** file located in the folder pointed to by the ROAMABLEROOTPREFIX variable, such as**C:\Users\John\AppData\Roaming\Bricsys\BricsCAD\V** The**geodatabase.xml**files lists cities and their WGS84 geo coordinates used by the **Choose Location**function of the Geographic Location dialog box.

Figure 6.1315



Related Commands

Geographiclocation (6.15.15) - sets the geographic location of the drawing. Image (6.17.3) - inserts images in drawings through the Drawing Explorer.

Figure 6.1316



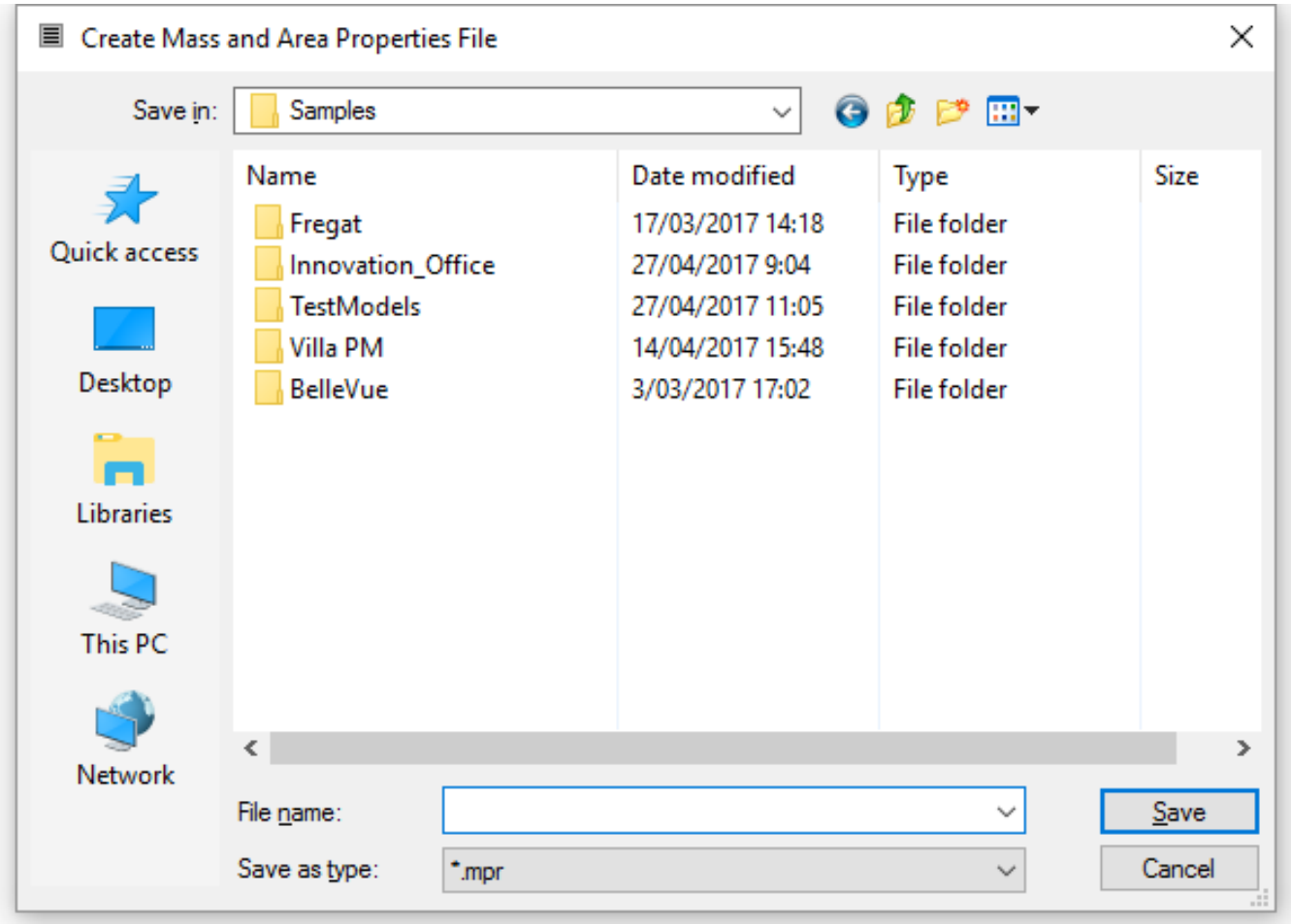
6.21.4 MassProp

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for ”mass properties”).

Accessing the Command

command bar: **massprop** menu bar: **Tools | Inquiry | Mass Properties** ribbon:**Tools | Inquiry | Mass Properties** toolbar: **Inquiry |** : **MassProp Prompts you in the command bar:** Select entities: **(Choose one or more entities; the program filters out any that are not ACIS objects)** Entities in set: 1 Select entities: **(Choose more ACIS objects, or else press Enter to produce the report)** Displays report in the command bar; press F2 to view report in the Prompt History (6.29.13) window: The command continues: Write analysis to a file? **(Type Y or N)** When you enter Y, displays dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/MassPropSaveDial Name the file, and then click Save.The report is stored in an MPR file, which can be opened in any text editor.MPR is short for ”mass properties report.”

Figure 6.1317



Command Options

Option	Description
Select entities	Specifies the entities on which to report properties.Choose 3D solid models and 2D region entities; all other entities are ignored by the program.
Write analysis to a file?	Writes the report to a text file.Type Y or N: • **Yes** - saves the report to a file; opens the Mass and Area Properties File dialog box: 1. Enter a file name 2. Click **Save**. The report is stored in an MPR file, which can be opened with any text editor • **No** - does not save the report to a file

Related Commands

Box (6.6.17) - creates 3D boxes as solid models. Cone (6.9.63) - creates 3D cones as solid models. Cylinder (6.9.60) - creates 3D cylinders as solid models. Dish (6.10.30) - creates the lower half of 3D balls as solid models. Dome (6.10.34) - creates the upper half of 3D balls as solid models. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. Region (6.26.18) - creates 2D regions from 2D entities. Revolve (6.26.30) - revolves 2D entities into 3D solid models. Section (6.27.11) - creates 2D regions from 3D models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Sphere (6.27.64) - creates 3D balls as solid models. Subtract (6.27.56) - removes one solid model from a second. Torus (6.29.22) - creates 3D donuts as solid models. Union (6.30.6) - joins two or more solid models into a single entity. Wedge (6.33.7) - creates 3D wedges as solid models.

Figure 6.1318



6.21.5 MatBrowserClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes the materials browser.

Accessing the Command

command bar: **matbrowserclose** context menu: right-click the ribbon or any toolbar, then choose **Render Materials** : **MatBrowserClose**

Command Options

No options.

Related Commands

MatBrowserOpen (6.21.6) - opens the materials browser to shows available materials for high-resolution renderings. -ToolPanel (6.29.21) - toggles the display of all tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Figure 6.1319



6.21.6 MatBrowserOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Render Materials panel to access materials used for renderings. **Note:** Most rendering materials are not installed with the program, but must first be downloaded using this panel.

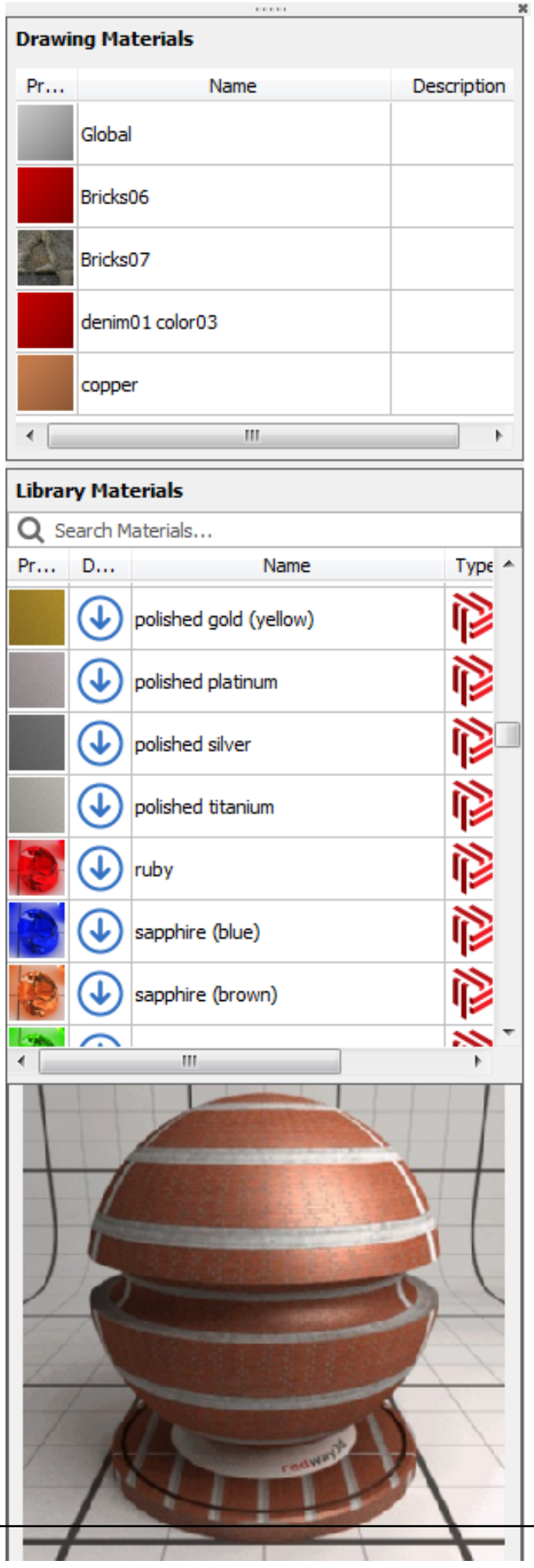
Accessing the Command

command bar: **matbrowseropen** aliases: **matb, matlib** context menu: right-click the ribbon or any toolbar, then choose **Render Material** : **MatBrowserOpen** Displays a panel with lists of materials:

- **Drawing Materials** pane (top panel) - lists materials used in the drawing
- **Library Materials** (middle pane) - list materials available to be downloaded from the master library
- **Preview** (lowest pane) - previews the selected material

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/matbrowseropen-panel.png})

Figure 6.1320



Command Options

Option	Description
Drawing Materials pane	Lists the material definitions contained in the current drawing. Toadd materials from the external library to the Drawing Materials list (and to this drawing): • Drag a material from the Library Materials list onto an entity in the drawing • Select one or more materials in the Library Materials list, right-click, and then choose **Add Material(s) to Drawing** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M • **Add material(s) to library** - copies the selected material(s) to the materials library in the folder defined by the RenderMaterialPathpreference. • **Change row height** - sizes the rows bewteen**Small**, **Medium**, and**Large** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so
Library Materials pane: Search Field	Seaches for materials by name: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M As you type text in the search field, materials that contain the text in their names are listed. The material list is updated as you type. Click the **X**button to clear the search field.

Library Materials Pane	Click the **Download**, **Name**, and otherheaders to sort the lists alphabetically. Click again to reverse the sort order. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Right click a material to display a context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M • **Add material(s) to drawing** - copies the selected material(s) to the current drawing. • **Delete material(s)** - removes the selected material(s) from the materials library • **Cancel download** - stops an ongoing material definition download • **Change row height** - sets the row height to**Small** , **Medium** or **Large** in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou Download column uses the bluearrow to indicate materials that have not yet been downloaded: double click to download the material.Downloaded materials are saved in the folder which is defined by the RenderMaterialPathpreference, such asC:\ProgramData\Bricsys\RenderMaterials. To assign a material to an entity, drag its name from this panel onto an entity The definition is added to the current drawing and the Drawing Materials list. When you drag a material that has not yet been downloaded onto an entity, then the material appears red in color. Once the download is complete, the material appears correctly.

Procedures

- To open or close a panel (1.12.7)
- To dock a panel (1.12.7)
- To dock a panel with respect to another docked panel (1.12.7)

Related Commands

MaterialMap (6.21.10) - allows to adjust how a texture is mapped to regions, faces and faces of a 3D solid. Mat-BrowserClose (6.21.5) - closes the materials browser. -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Figure 6.1321

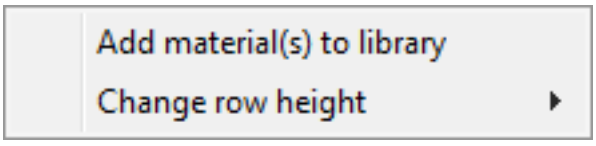


Figure 6.1322

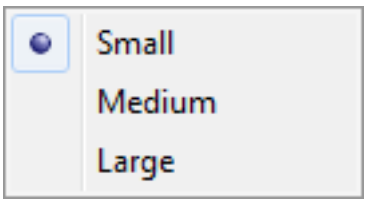


Figure 6.1323



Figure 6.1324

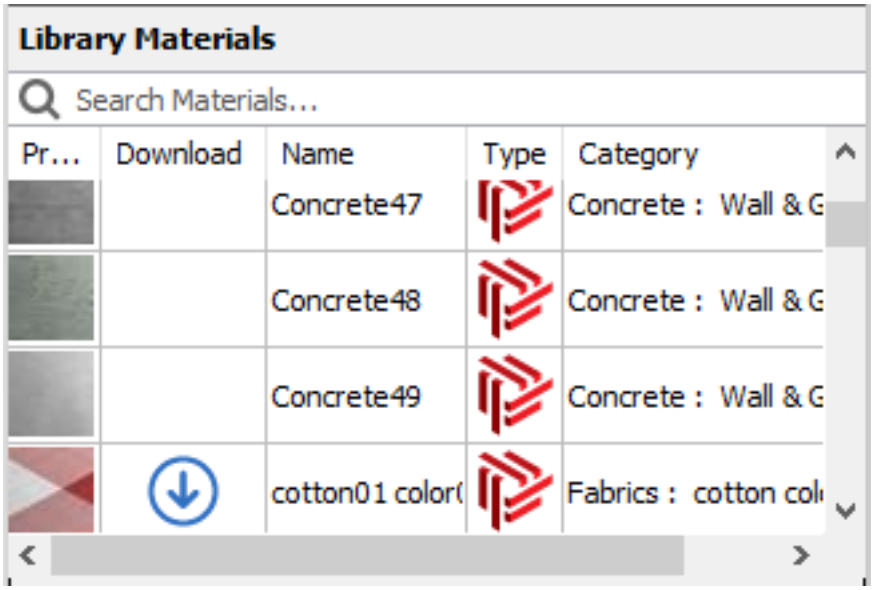


Figure 6.1325

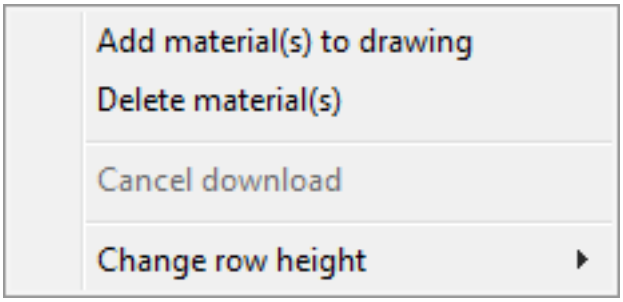


Figure 6.1326

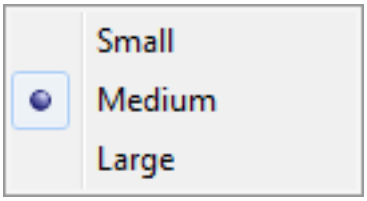


Figure 6.1327



Figure 6.1328



6.21.7 MatchPerspective

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Matches the current model space view with a background image after you select three or more point pairs. **Note:** This command operates only in perspective mode. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so

Accessing the Command

 command bar: **matchperspective** menu bar: **View | Match Perspective** ribbon:**View | View | Viewpoint | Match Perspective** toolbar: **View |**  To turn on Perspective viewing mode:

- Set the **Perspective** variable to on
- In the **View** toolbar, click**Perspective**()
- In the **View** menu, click**Perspective**
- In the ribbon's**View** panel and **View** tab, click**Perspective**()

Figure 6.1329



: **MatchPerspective Prompts you in the command bar:** Enter Model Point: **(Specify a point on the 3D model)** Enter Image Point or [Undo]: **(Click the corresponding point in the image)** Enter Model Point or [Undo]: **(Specify another point on the 3D model)** Enter Image Point or [Undo]: **(Click the corresponding point in the image)** Enter Model Point or [Undo]: **(Specify yet another point on the 3D model)** Enter Image Point or [Undo]: **(Click the corresponding point in the image)** Enter Model Point or [Undo] <Match>: **(Specify more point pairs, or else press Enter to match the view)** As you pick pairs of matching points between the model and image, they are connected by green lines. After you press **Enter**, the model matches the perspective of the background image.Optionally use the View (6.31.1) command to save the view. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/MatchPerspe

Command Options

Option	Description
Enter Model Point	Specify a point on the 3D model; it can be helpful to use entity snaps for precision
Enter Image Point	Click the corresponding point in the image
Undo	Removes the previous point pair.
Match	Matches the model space view with the background image.

Related Commands

Background (6.6.1) - sets the model space background for the current viewport and saved views. Camera (6.9.2) - Creates perspective views. -View (6.31.2) - Creates and sets named views through the command bar. View (6.31.1) - Creates and sets named views through the Drawing Explorer.

Figure 6.1330



6.21.8 MatchProp

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Copies the properties and styles from one entity and then applies them to one or more other entities, with control over which properties are copied (short for ”match properties”).

Accessing the Command


command bar: **matchprop** transparent: **'matchprop** alias: **ma** ribbon:**Home | Edit | Match** toolbar: **Standard |**  : **MatchProp Prompts you in the command bar:** Select entity to copy properties from: **(Choose one entity.)** Active Settings: 'Color' 'Layer' 'Linetype' 'Linetype Scale' 'Lineweight' 'Transparency' 'Thickness' 'Plot Style' 'Dimension' 'Polyline' 'Material' 'Text' 'Shadow Display' 'Hatch' 'Multileader' Select entities to copy properties to or [Settings]: **(Choose one or more entities, or type S) Select entities to copy properties to** or [Settings]: **(Choose additional entities, or press Enter to exit the command)**

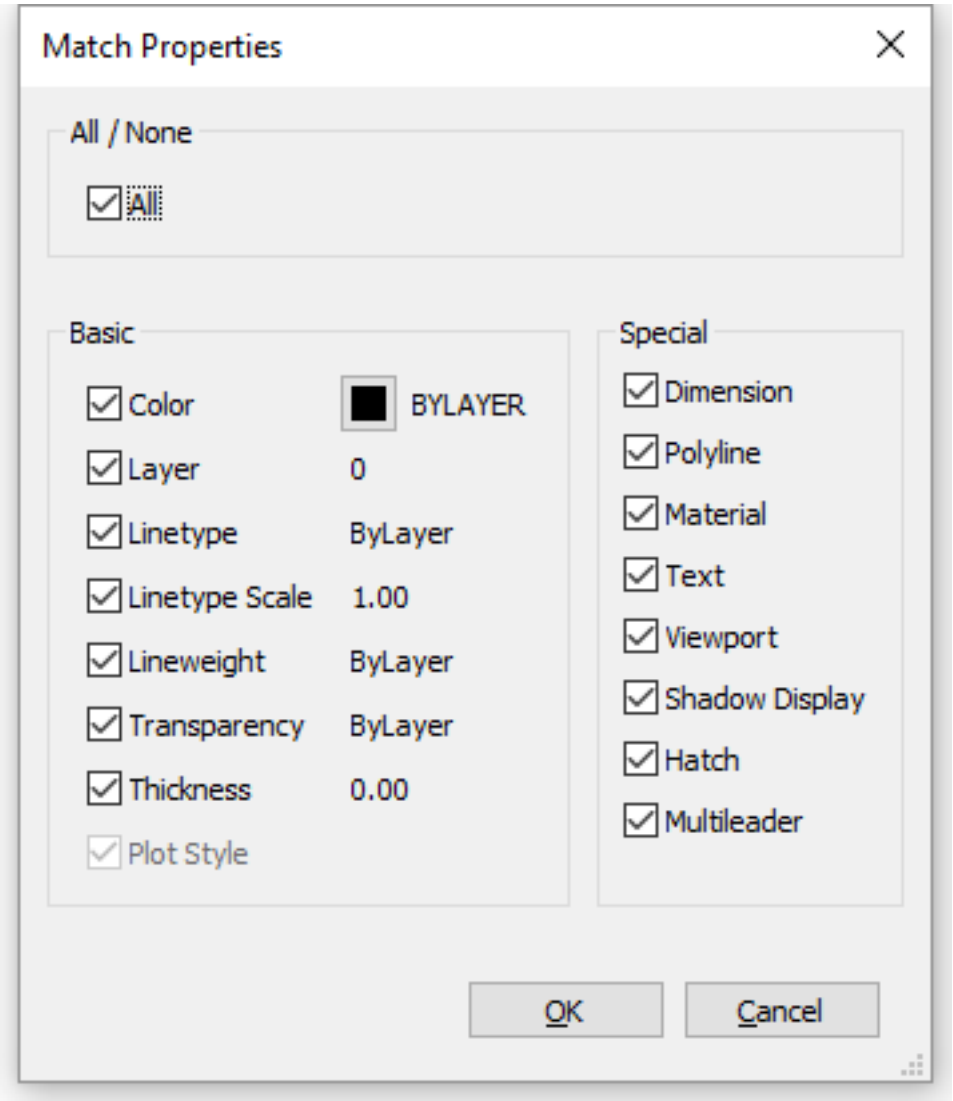
Command Options

Option	Description
Select entity to copy properties from	Specifies the entity whose properties will be copied. The currently active settings display in the command bar, such as 'Color' 'Layer' 'Linetype' 'Linetype Scale' 'Lineweight' 'Transparency' 'Thickness' 'Plot Style' 'Dimension' 'Polyline' 'Material' 'Text' 'Shadow Display' 'Hatch' 'Multileader'
Select entities to copy properties to	Specifies the entities whose properties are to be changed.
Settings	Specifies which properties are to be copied. Thedialog box lists applicable properties, along with the current value of each one. The Special section lists the styles that will be copied: **Dimensions** - dimension style and annotative property of dimensions, leaders, and tolerances **Hatch** - all hatch properties and annotative scaling **Materials** - adds or removes materials, depending on whether the source entities has a material definition or not **Multileaders** - multileader style and annotative scaling **Polylines** - width and linetype generation settings, but other settings are ignored, such as taper and spline **Shadow Displays** - all properties **Tables** - table style **Text** - text style and annotative property of text and mtext **Viewports** - viewport-specific properties, such as on or off, snap and grid status, and scale factor, but not other properties, such as clipping or frozen layer states (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M 1. Choose some or all properties 2. Click **OK** 3. Select entities at the prompt: **Select entities to copy properties to or [Settings] - (select entities.)**

Procedures

- To copy properties between entities (1.23.3)

Figure 6.1331



Related Commands

Copy (6.9.50) - copies entities. Properties (6.24.49) - changes properties assigned to entities. Layer (6.20.2) - assigns properties to layers. Color (6.9.33) - assigns the working color. Linetype (6.20.33) - assigns the working linetype. LWeight (6.20.41) - assigns the working lineweight.

Figure 6.1332



6.21.9 MaterialAssign

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Applies the material specified by the **CMaterial** variable to 3D entities that you select.

Accessing the Command


Keyboard:**materialassign** Transparent: **'materialassign : MaterialAssign Prompts you:** Select object to assign material to:**(Select one or more 3D entities)** Select object to assign material to or [Undo]:**(Select more 3D entities, or type u to undo the material assignment)** Select object to assign material to or [Undo]:**(Hold down the Alt key to pick a material)** Select material from object:**(Pick another 3D entity with the material you wish to copy)** Select object to assign material to or [Undo]:**(Select more 3D entities, or type u to undo the material assignment,or else press Enter to end the command)**

Command Options

Option	Description
Select object to assign material to	To pre-designate a material, enter its name into the **CMaterial** variable before starting this command. Select one or more 3D objects to which you want the current material assigned.Notice the paintbrush cursor; use it to ”paint” entities with the material.  To see the change in material, ensure that the visual style is set to ”Rendering” with the Properties panel Visual Style setting.
Undo	Removes the material you assigned to the previously-selected entities.
Select material from object	Hold down the **Alt** key to copy the material from another 3D entity; notice the dropper cursor.  When the program next prompts you ”Select material from object,” pick another 3D entity with the material you wish to copy.

Related Command

Materials (6.21.11) - creates and edit rendering materials.

Figure 6.1333



6.21.10 MaterialMap

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Adjusts the placement of texture images onto 3D solids, 3D faces, wide polylines, and polygon meshes to create a more realistic look, such as matching a brick pattern to the wall of a house.

Accessing the Command

 command bar: **materialmap** transparent: **'materialmap** alias: **setuv** menu bar: **View | Rendering | Material Map** toolbar: **Rendering |** : **MaterialMap Prompts you in the command bar:** Select an option [Box/Planar/Spherical/Cylindri mapping to (Y)/Reset mapping)] <Box>: **(Choose an option)** Select faces or entities: (**Select one or more entities)** Entities in set: 1 Select faces or entities: (Se**lect more entities, or press Enter to end the command)**

Command Options

Option	Description

Select faces or entities	Specifies the entities whose material maps are to be adjusted.The entities you select are included in the yellow bounding box. When you select entities, blue grips and the manipulator (6.21.2) widget appear: • Drag blue grips to change the scale of the material, horizontally and vertically • Click different parts of the manipulator to move, scale, and rotate the material (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M **Scale the mapped image** - **(drag one of the blue arrow grips to scale the texture in the X-, Y- or Z direction.) Move the mapped image**- **(follow these steps)**: 1. Place the cursor over the one of the tripod axes 2. Click when the second arrowhead appears at the other end of the axis 3. Move the cursor to adjust the position of the pattern; in the image below the red Y-axis is selected. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou **Rotate the mapping image** - **(move the cursor over one of the rotation handles and then click when the handle expands.)** 1. The rotation axis in the color of the selected handle displays. The material map rotates dynamically around the selected axis. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou Z.png}) 2. Click a point to define the rotation angle.
Box	Wraps the texture around the six faces of a the bounding box of the selected entity or entities; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/r **Edit the mapping or [reseT/sWitch mapping mode] <Accept current mapping>** - **(choose an option)**: • **Enter** - accepts the material mapping, and returns to the previous prompt • **reseT**- restores the default mapping; reverses all previously applied adjustments. • **sWitch mapping mode** - switches to a different mapping mode; prompts you: **Select an option [Box/Planar/Spherical/Cylindrical/Copy mapping to (Y)/Reset mapping)] <Box>- (select an option.)**

Planar	Aligns the texture to a single face.To select a face of a 3D solid, set SELECTIONPREVIEW (1.13.11) = 2 or 3 or check the **Select Faces** option of the SELECTIONMODES (1.13.11) variable. 1. Place the cursor over the 3D solid face and then click when the face highlights. Repeatedly hit the **Tab** key to select an obscured face (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Prompts you: **Edit the mapping or [reseT/sWitch mapping mode] <Accept current mapping>** - **(choose an option.)** • **Enter** - accept the material mapping • **reseT**- restores the default mapping; reverses all previously applied adjustments. • **sWitch mapping mode** - switches to a different mapping mode; prompts you: **Select an option [Box/Planar/Spherical/Cylindrical/Copy mapping to (Y)/Reset mapping)] <Box>- (select an option.)**
Spherical	Aligns the texture to a spherical shape. The top and bottom edges of the image are compressed to a point at the north and south poles of the sphere.Spherical mapping of a brick pattern (left) compared to box mapping (right). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Prompts you: **Edit the mapping or [reseT/sWitch mapping mode] <Accept current mapping>** - **(choose an option)**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M • **Enter** - accepts the material mapping, and returns to the previous prompt • **reseT**- restores the default mapping; reverses all previously applied adjustments. • **sWitch mapping mode** - switches to a different mapping mode; prompts you: **Select an option [Box/Planar/Spherical/Cylindrical/Copy mapping to (Y)/Reset mapping)] <Box>- (select an option.)**

Cylindrical	Aligns the texture to a cylinder shape. The vertical edges of the image are wrapped together. The height of the image is scaled along the axis of the cylindrical shape.Cylindrical mapping of a wood pattern (left) compared to a box mapping (right). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Prompts you: **Edit the mapping or [reseT/sWitch mapping mode] <Accept current mapping>** - **(choose an option)**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M • **Enter** - accepts the material mapping, and returns to the previous prompt • **reseT**- restores the default mapping; reverses all previously applied adjustments. • **sWitch mapping mode** - switches to a different mapping mode; prompts you: **Select an option [Box/Planar/Spherical/Cylindrical/Copy mapping to (Y)/Reset mapping)] <Box>- (select an option.)**
copY mapping to	Applies mapping from the source entity or face to the selected entities, thus duplicating the mapping, including any adjustments, to other entities. All entities share the same mapping origin, axes and scale. As a result the texture image is extended seamlessly from one entity to another.Prompts you: 1. **Select face or entity** - **(select the source entity.)** 2. **Select the entities to copy the mapping to: Select faces or entities**- **(select the target entity.)** 3. **Entities in set: 1 Select faces or entities** - **(select more target entities or press Enter.)**
reseT mapping	Restores the default mapping; reverses all previously applied adjustments.

Related Commands

MatBrowserOpen (6.21.6) - opens the materials browser, which shows an overview of available high-resolution render materials. Materials (6.21.11) - creates materials and edits their properties through the Drawing Explorer.

Figure 6.1334

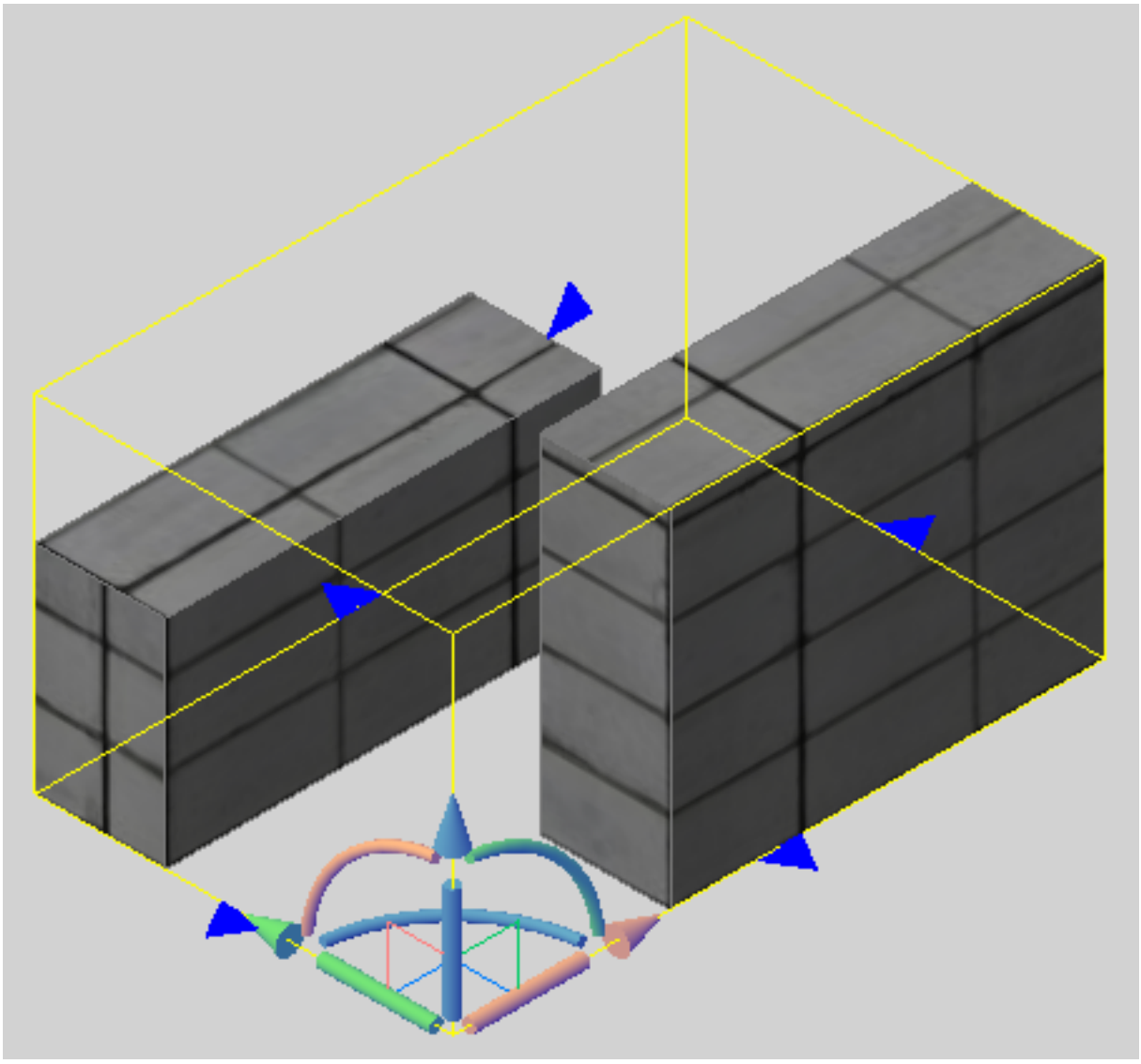


Figure 6.1335

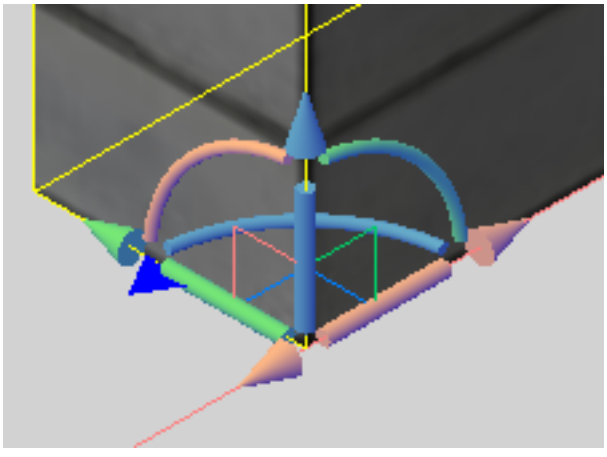


Figure 6.1336

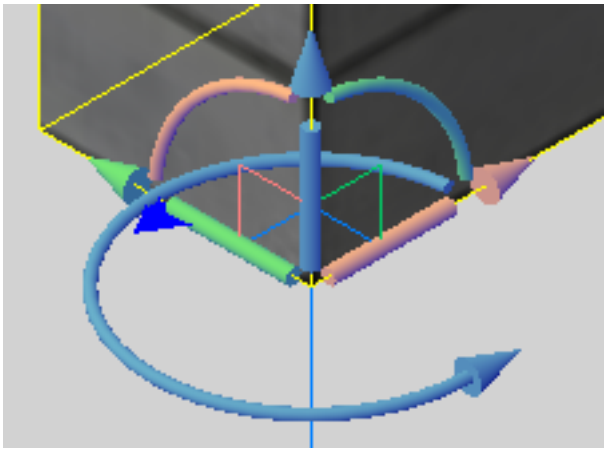


Figure 6.1337

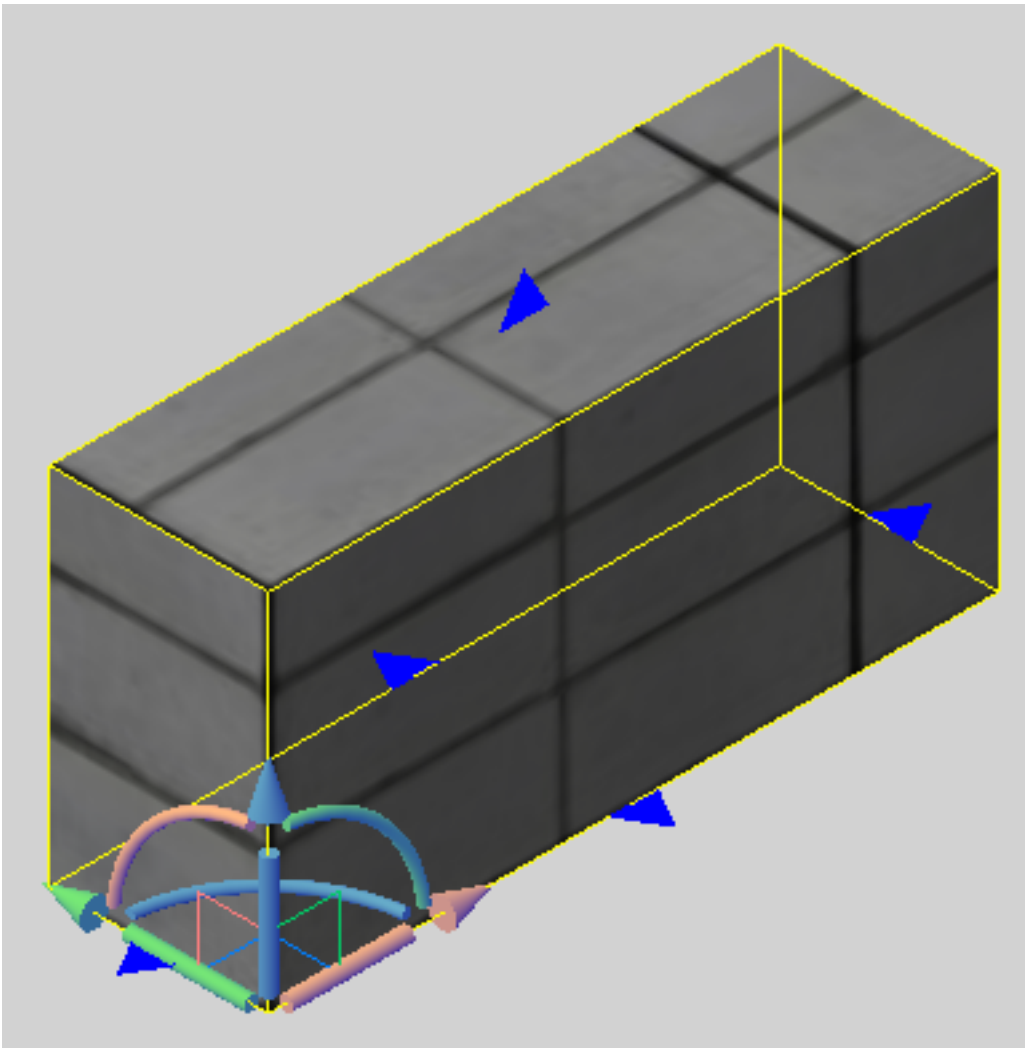


Figure 6.1338

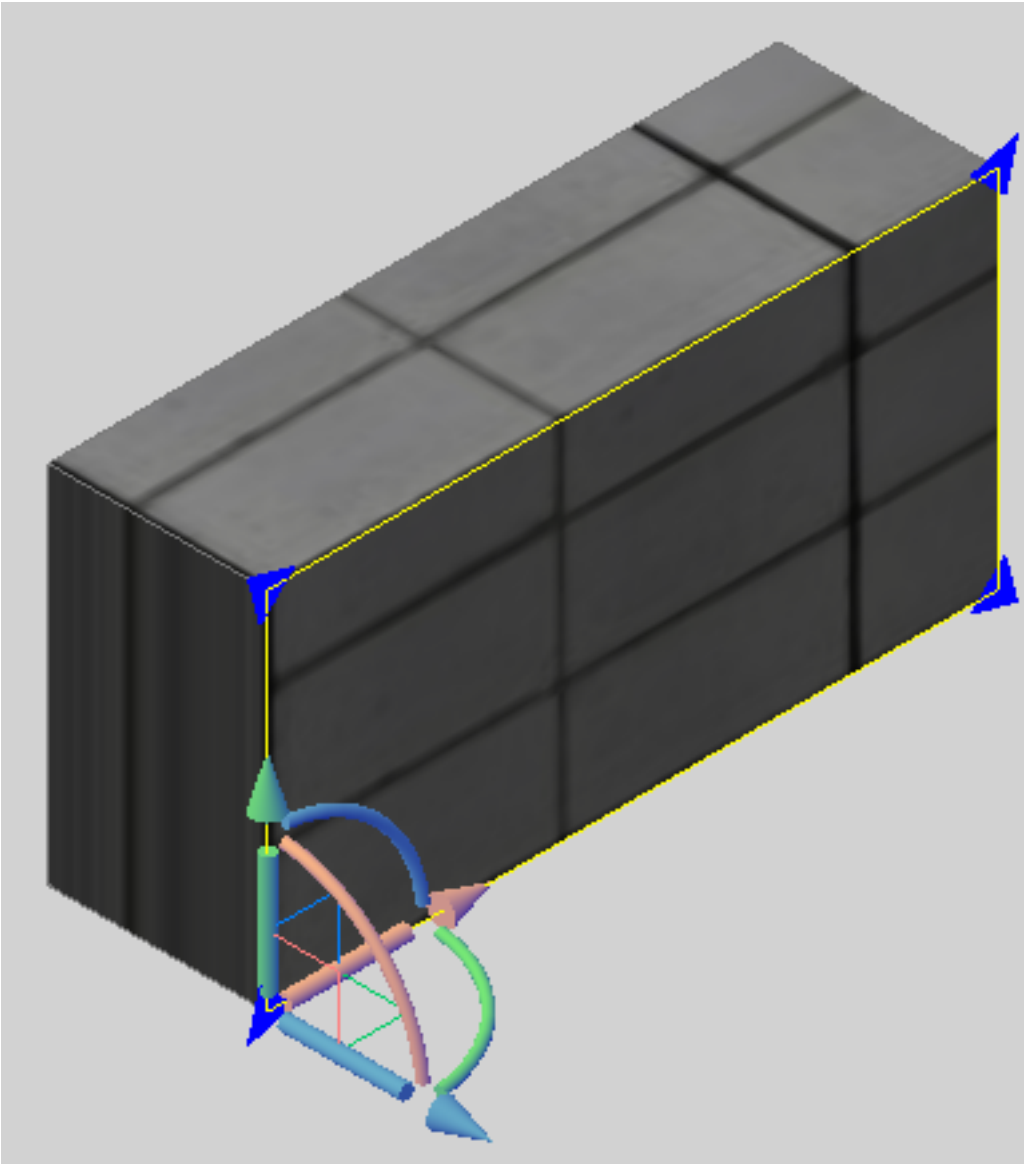


Figure 6.1339

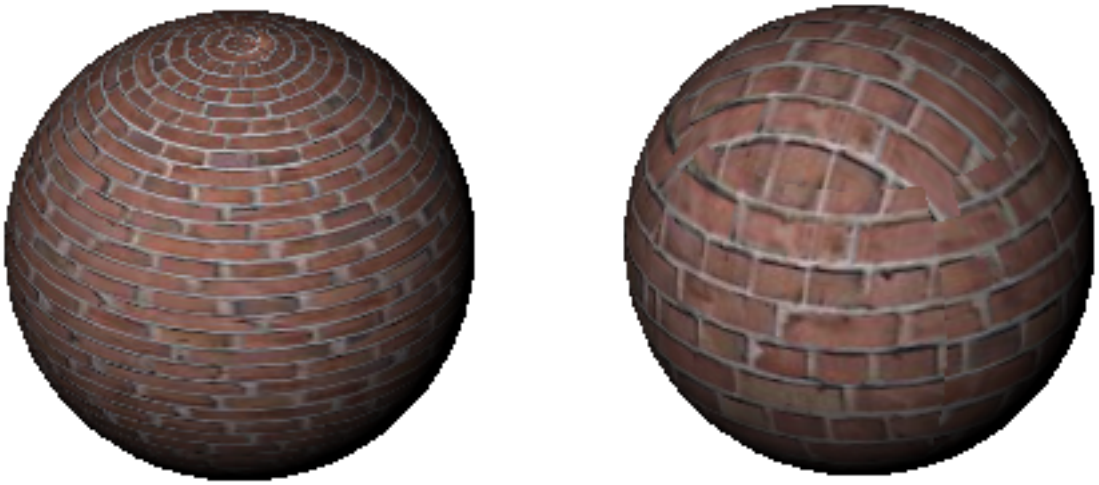


Figure 6.1340

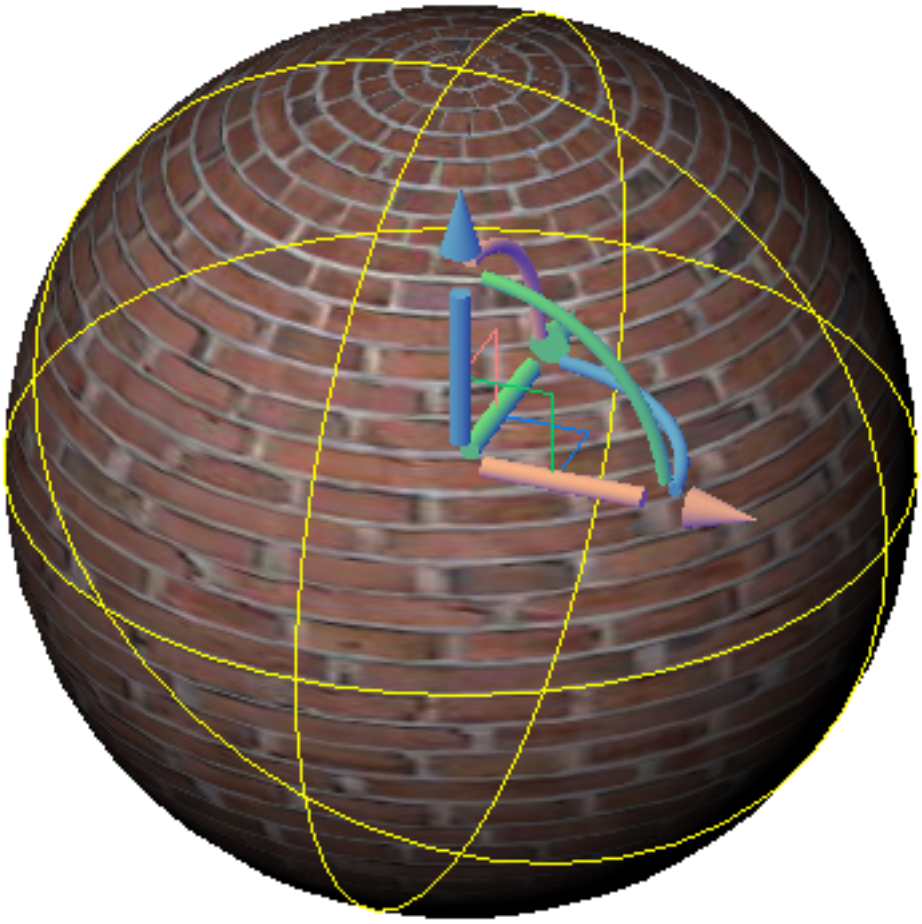


Figure 6.1341

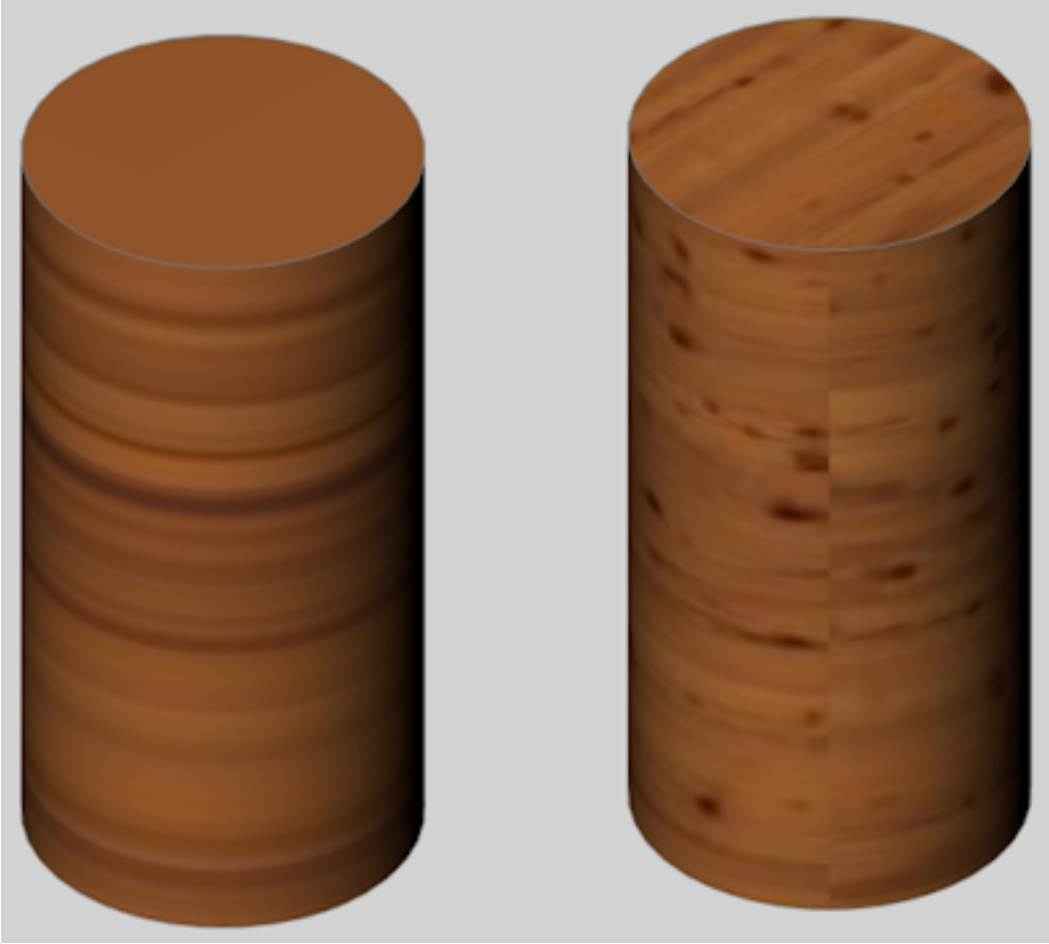


Figure 6.1342

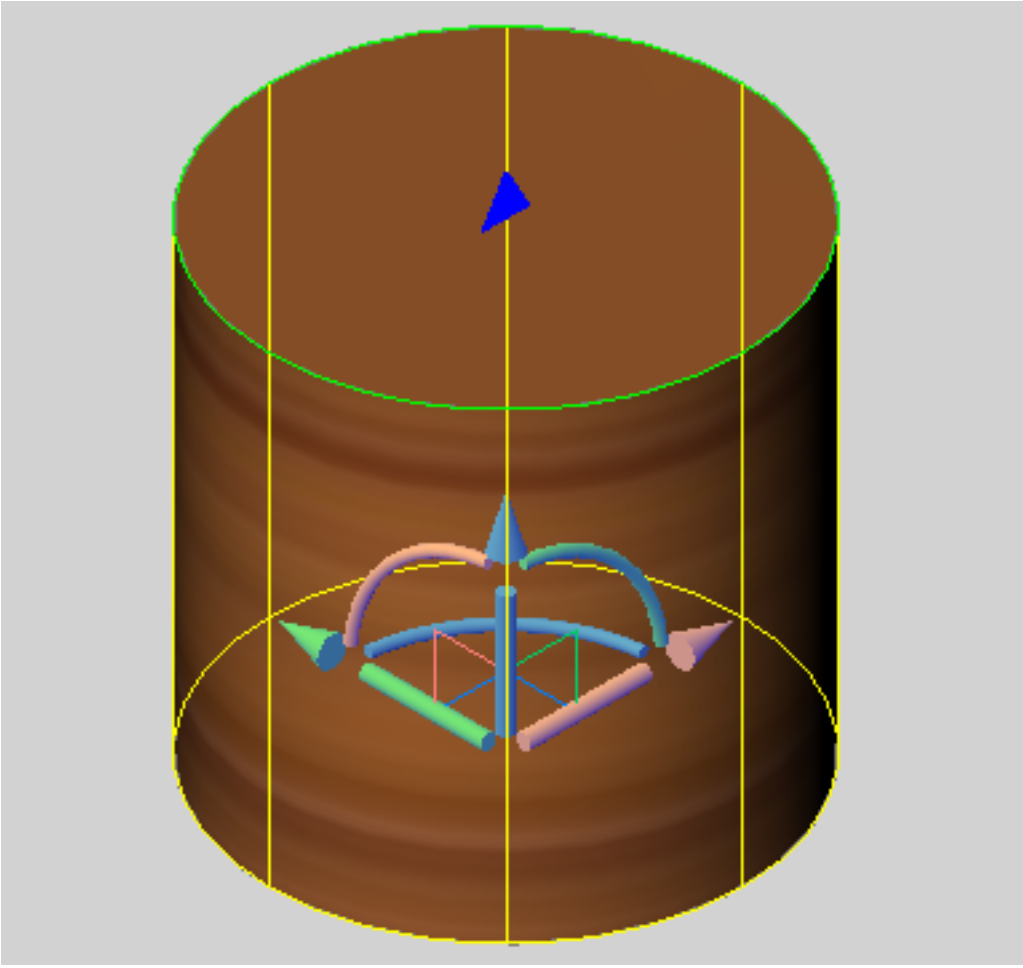


Figure 6.1343



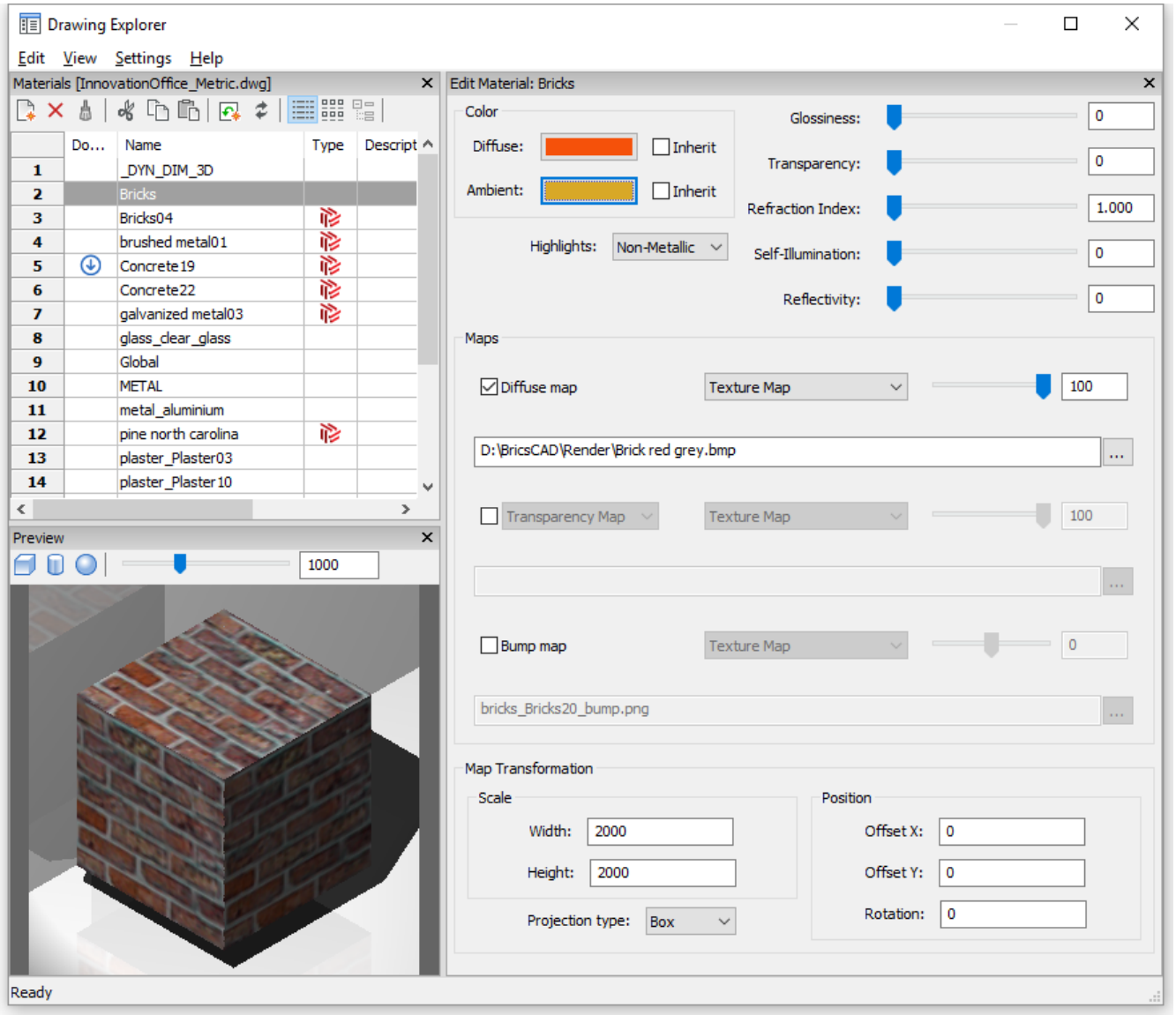
6.21.11 Materials

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and edits materials used for renderings through the Drawing Explorer dialog box. **Note:**Materials provided by RedWay ( )cannot be edited initially and so their settings are greyed-out, except for **Map Transformation**.To edit these materials, right-click a Redway material, and then choose **Convert to regular material** from the context menu.

Accessing the Command

 command bar: **materials** aliases:**mat, finish, rmat** menu bar: **Tools | Drawing Explorer | Materials View | Rendering | Materials** ribbon:**Home | Settings | Drawing Explorer | Materials View | Materials | Materials** toolbar: **Rendering|** : **Materials** Displays the Materials section of the Drawing Explorer (1.14.2) dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DrawingExplorerMaterials.png})

Figure 6.1344



Command Options

Materials Toolbar Option	Description
New	Creates a new material: 1. Click **New**. 2. Name the material. 3. (Optional) Fill out the **Description** field.
Delete	Deletes selected material(s).The Global material cannot be deleted.

Preview Options	Description
Cube	Displays a preview of the selected material applied to a cube.
===ERROR WITH DOWNLOADED IMAGE=== Cylinder	Displays a preview of the selected material applied to a cylinder.
Sphere	Displays a preview of the selected material applied to a sphere.
Light Intensity	Sets the light intensity in the Preview window: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M • Drag the slider • Click before or after the slider

Edit Material Option	Description
Name	Type a name for the material. The material name must be unique.The Global material cannot be renamed.
Description	Type a description for the material. The material description is optional.

Color Option	Description
Diffuse Ambient	Sets the diffuse and ambient color: • Check the **Inherit** option to apply the entity color • Click the colored tile, then choose the color in the **Color** dialog box
Highlights	Sets the highlight property of the material: 1. Click the Highlights droplist 2. Choose either **Non-Metallic** or **Metallic**
Glossiness Transparency Refraction Index Self-Illumination Reflectivity	Defines a property of the material surface: • Drag the slider • Click before or after the slider • Type the desired value in the settings field **Glossiness** - range = 0 - 100 **Transparency** - range = 0 - 100 **Refraction Index** - range = 1.00 - 3.00 **Self-Illumination** - range = 0 - 100 **Reflectivity** - range = 0 - 100

Maps Option	Description

Maps

Texture maps add detail to a surface, which is not included in the 3D model itself.To apply a texture map:

1. Turn on the **Diffuse, Transparency** and/or**Bump Map** options.
2. Do one of the following:
   - Select **Wood** or **Marble** from the **Texture Map** option button
   - Choose **Texture Map**, then press the **Browse** button () and select a texture bitmap file (*).
3. Set the **Blend factor** for the texture map:
   - Diffuse and Transparency maps: range = 0 to 100
   - Bump map: range = -1000 to 1000
   - Use the slider or type the factor in the settings field.
   - The **Blend factor** sets the intensity of the texture map.

(*) The TextureMapPath (1.13.11) user preference defines the search path for texture map images. In the BricsCAD program folder exist three subfolders under **Textures** , each containing a number of texture files of the same name. Images in folder **1** have a size of 256 x 256 pixels, folder **2** contains images of 512 x 512 pixels, images in folder **3** have a size of 1024 x 1024 pixels. If the **Diffuse map** setting of a material uses the image name only (not path), you can control the quality of a rendered image by setting the **TextureMapPath** user preference to folder **1** , **2** or **3** .

Diffuse Map

Diffuse maps apply a texture to the surface of a material such as wood grains, bricks or tiles.Drag the slider to blend the texture map with the Color (6.9.33) settings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M

Transparency Map (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/Transparency Map.png})

The selected **Texture Map** can be applied as **Transparency Map** or as **Cutout Map**. In the examples below, this image is used as transparency map and



cutout map: If **Transparency Map** is selected, the selected **Texture Map** image defines a transparency map. The image is recommended to be a grayscale image. White pixels will be invisible, black pixels will be opaque and gray pixels will be transparent.

- Colored images are converted to grayscale images in the background.
- The meaning of black and white is the inverse of **Cutout Map** textures.
- The alpha channel of the image is ignored.

Blend factor: the transparency values in the texture will approach the scalar **Transparency** setting as the blend factor approaches 0. This means that if the **Blend factor** is 100, then the transparency value will controlled completely by the **Texture Map** image. If the **Blend factor** is 0, the transparency texture will be completely ignored and the scalar **Transparency** value will be used. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
If **Cutout Map** is selected, then the selected transparency map defines cutouts: parts of the objects will be fully visible and parts not visible at all, depending on the pixels of the selected image. The image is recommended to be a bitonal black-and-white image, without grays. White pixels are visible, and black pixels are invisible. The **Cutout Map Inverted** option inverts the effect: visible parts become invisible and vice versa.

- Gray scale and color images are converted to black-and-white images in the background.
- The meaning of black and white is the inverse of **Transparency Map** textures.
- The alpha channel of the image is ignored.
- (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou

Bump Map

Bump mapping is a technique for simulating bumps and wrinkles on the surface of an object. The result is an apparently bumpy surface although the surface of the underlying object is not modified. Blend factor range = -1000 / +1000 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
Left: Diffuse Map only; Right: Bump Map applied.

Map Transformation

Scale	The texture map image is applied at a size of 1 drawing unit multiplied by the **Width** and **Height** factors. E.g. if the **Width** and **Height** factors or both set to 10, the size of the texture image is 10 x 10 drawing units.
Position	
Offset	Texture maps maps are tiled starting from the origin of the WCS. To adjust the tiling you can define an Offset X and Offset Y. The offsets are expressed in drawing units.
Rotation	Sets the rotation of the texture maps. Type the desired rotation in the **Rotation** field.

Related Commands

MatBrowserOpen (6.21.6) - opens the materials browser, which shows an overview of available high-resolution render materials. MaterialMap (6.21.10) - allows to adjust how a texture is mapped to regions, faces and faces of a 3D solid. Render (6.26.22) - calculates a photorealistic image of the 3D model. Light (6.20.28) - creates a new light in the drawing. VisualStyles (6.31.13) - creates and controls visual style definitions in the Drawing Explorer. Explorer (6.13.14) - opens the Drawing Explorer dialog box.

Figure 6.1345

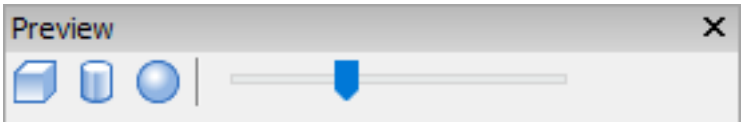


Figure 6.1346

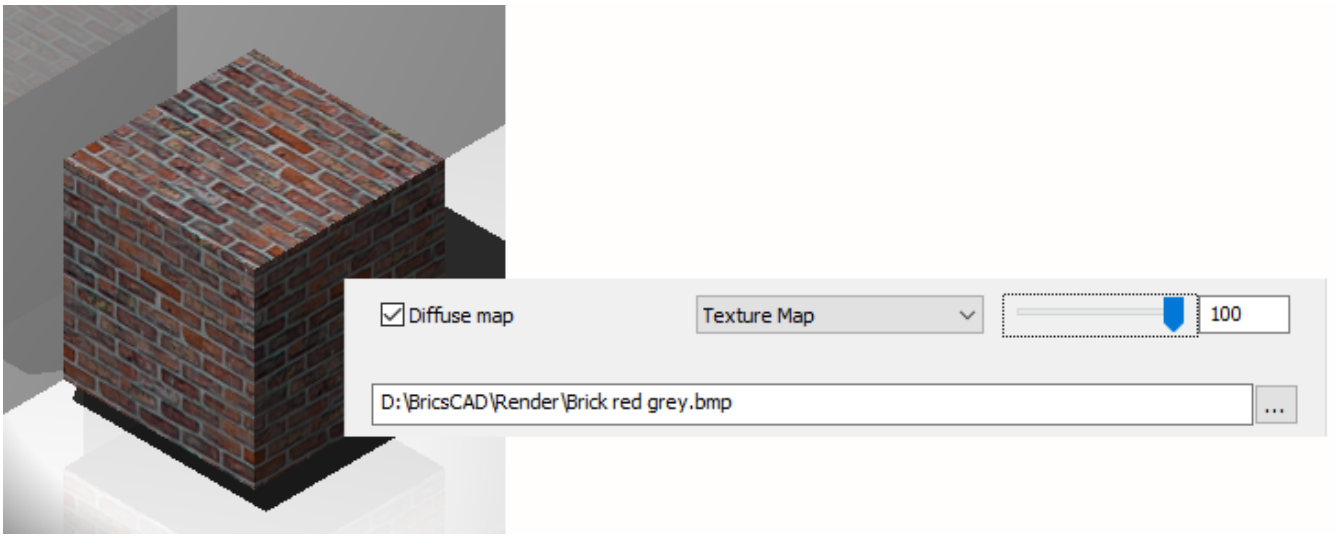


Figure 6.1347

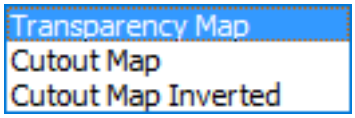


Figure 6.1348

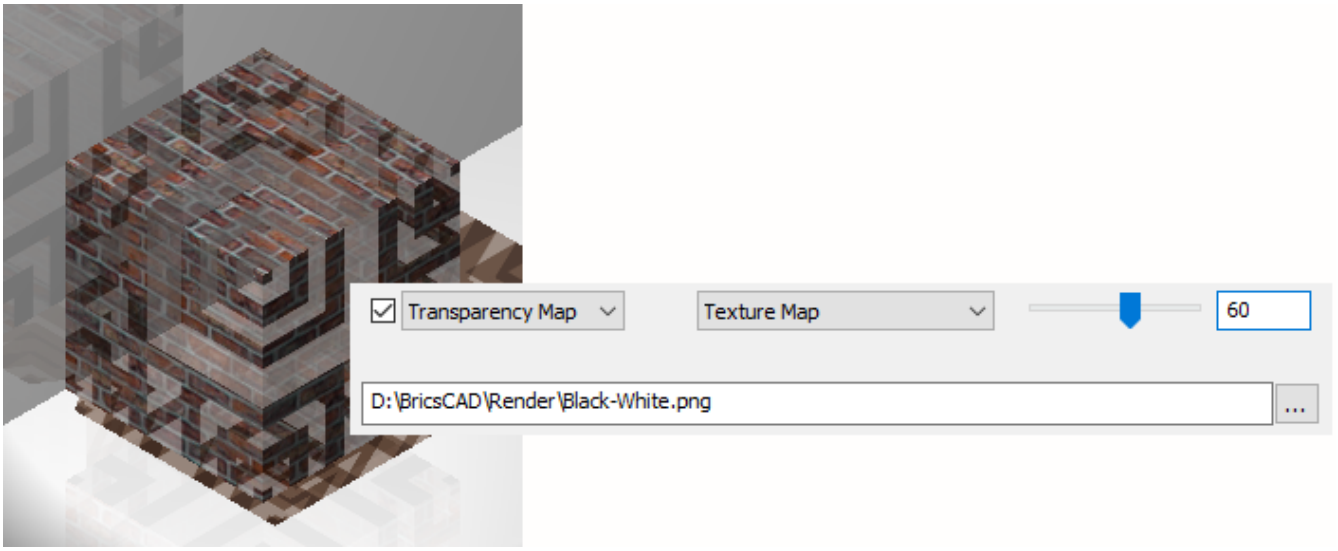


Figure 6.1349

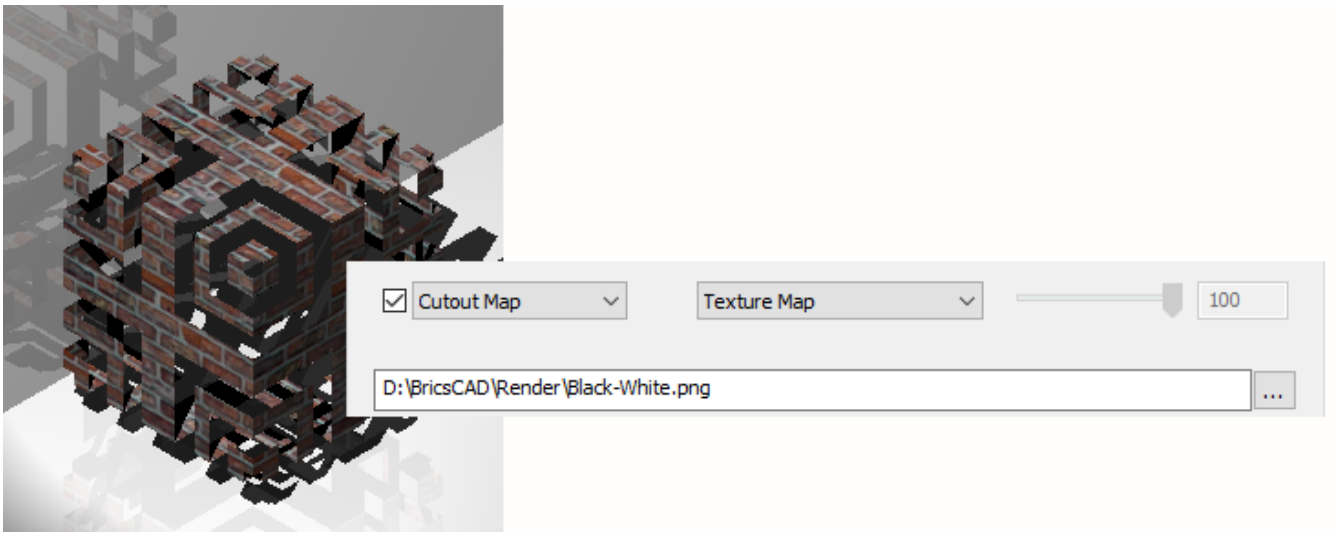


Figure 6.1350

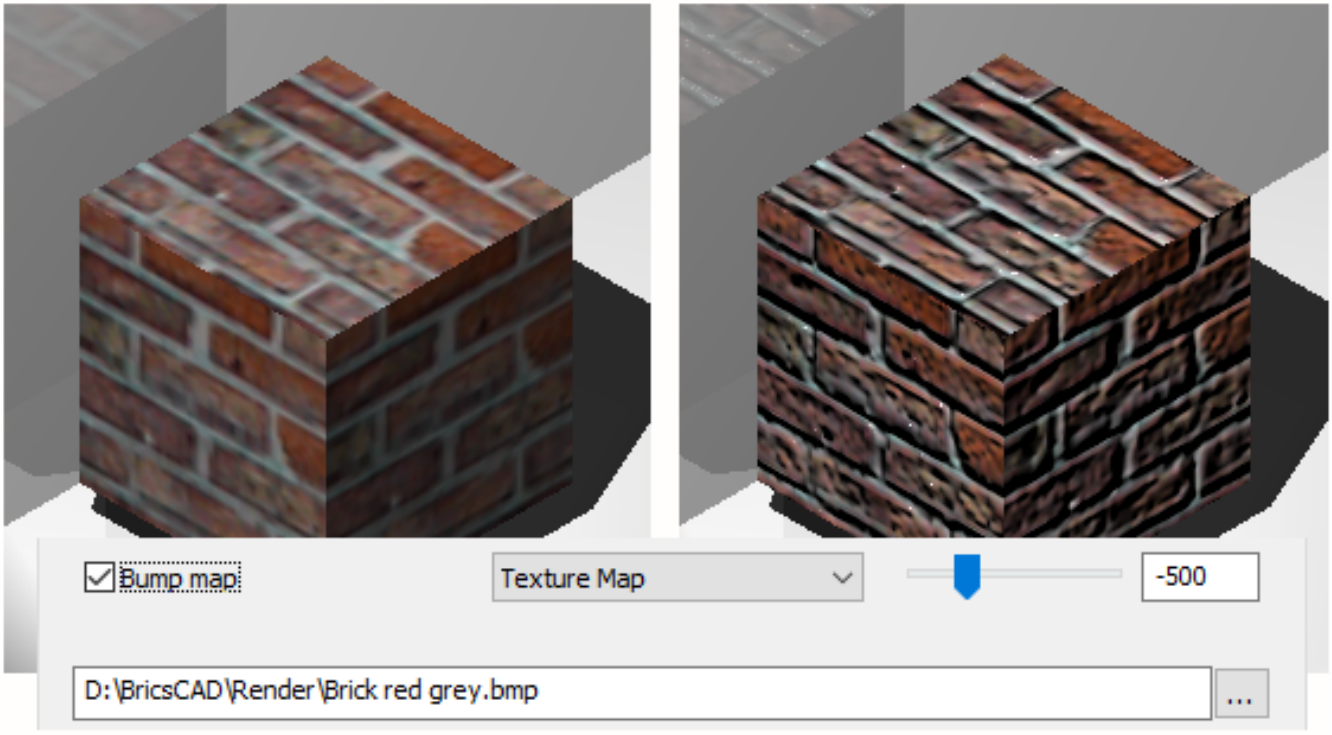


Figure 6.1351



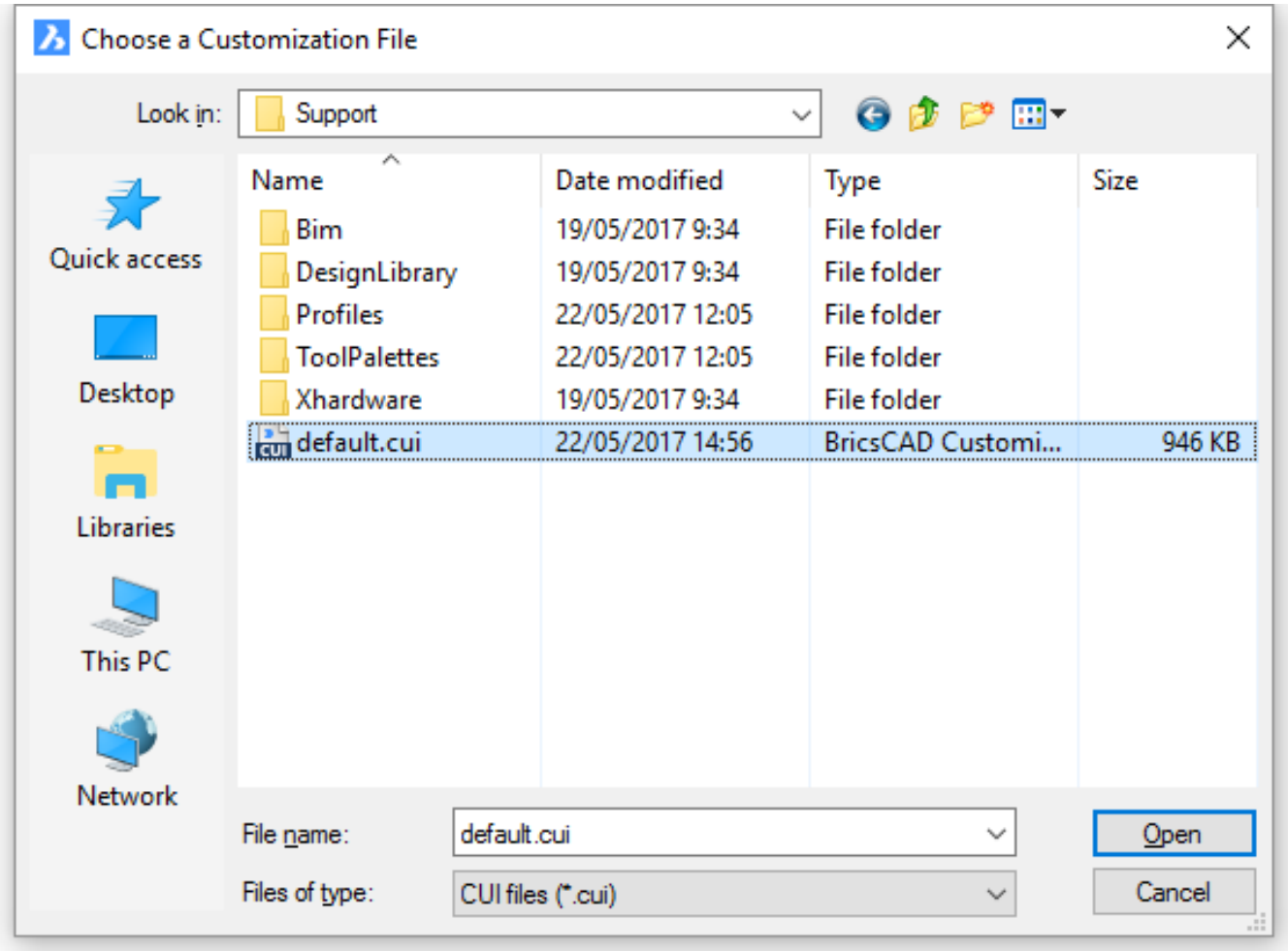
6.21.12 Menu

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads a variety of menu files into the program that modify its user interface.

Accessing the Command

 command bar: **menu** menu bar: **Tools | Load menu** ribbon:**Tools | Customization | Load Menu** toolbar: **Tools** |  : **Menu** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
Choose a menu file, and then click **Open**.

Figure 6.1352



Command Options

Option	Description
File name	Specifies the name of the menu file.
File type	Specifies the format of menu file: • **CUI** - compatible with AutoCAD 2007 and later • **ICM** - compatible with IntelliCAD menu files • **MNU** - compatible with AutoCAD 2006 and earlier • **MNS** - LISP source code used with MNU files
Open	Opens the menu file. **Caution**: loading a menu file can replace the user interface of BricsCAD unexpectedly.

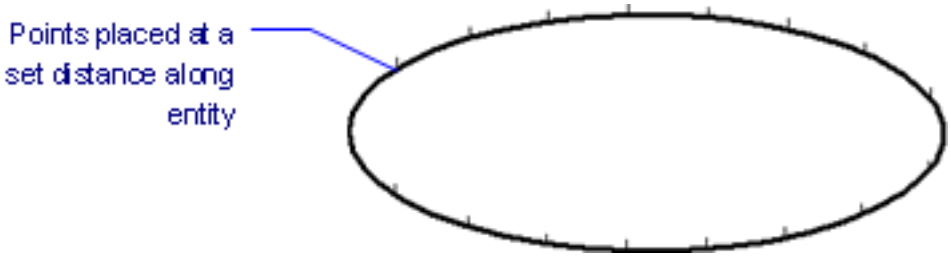
Related Commands

CuiLoad (6.9.56) and CuiUnload (6.9.57) - loads and unloads menu files. Customize (6.9.58) - customizes the user interface. Menu (6.21.12) - loads menu files into BricsCAD's user interface. MenuLoad (6.21.14) - loads CUI (user interface customization), MNU (menu), MNS (LISP code), and ICM (IntelliCAD menu) files. MenuUnload (6.21.15) - unloads menu files.

Figure 6.1353



Figure 6.1354



6.21.13 Measure

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places points or blocks along entities at equally-spaced distance that you specify. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image311.png})

Accessing the Command

 command bar: **measure** menu bar: **Modify | Measure** ribbon:**Draw | Draw | Points | Measure** toolbar: **Modify | Measure/Divide |**  : **Measure Prompts you in the command bar:** Select entity to measure: **(Choose a line, polyline, spline, arc, circle, helix, or ellipse)** Segment length [insert Block]: **(Enter a number, or type B)** Note: Points are usually invisible, so use the PdMode (6.24.31)variable to make them larger.

Command Options

Option	Description
Select entity to measure	Chooses a single entity: a line, polyline, spline, helix, arc, circle, or ellipse.
Segment length	Specifies the distance between segments; the segment is measured beginning at the pick point.
Insert block	Uses blocks in place of points; prompts you: 1. **Name of block to insert [? to list]** - **(specify the name of the block; the block must exist in the drawing.) ? to list** - lists the name of the blocks in the drawing. 2. **Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-Align blocks>** - **(toggles whether blocks are aligned with the entity)**: • **Yes** - rotates blocks to match the entity alignment • **No** - does not rotate blocks; they maintain their orientation

Procedures

- To measure an entity using points (1.23.39)
- To place blocks a specified interval along an entity (1.23.39)

Related Commands

Divide (6.10.33) - places evenly-spaced points along entities. Point (6.24.31) - places points in drawings. DdPType (6.10.18) - specifies the look and size of point entities through the Settings dialog box

Figure 6.1355



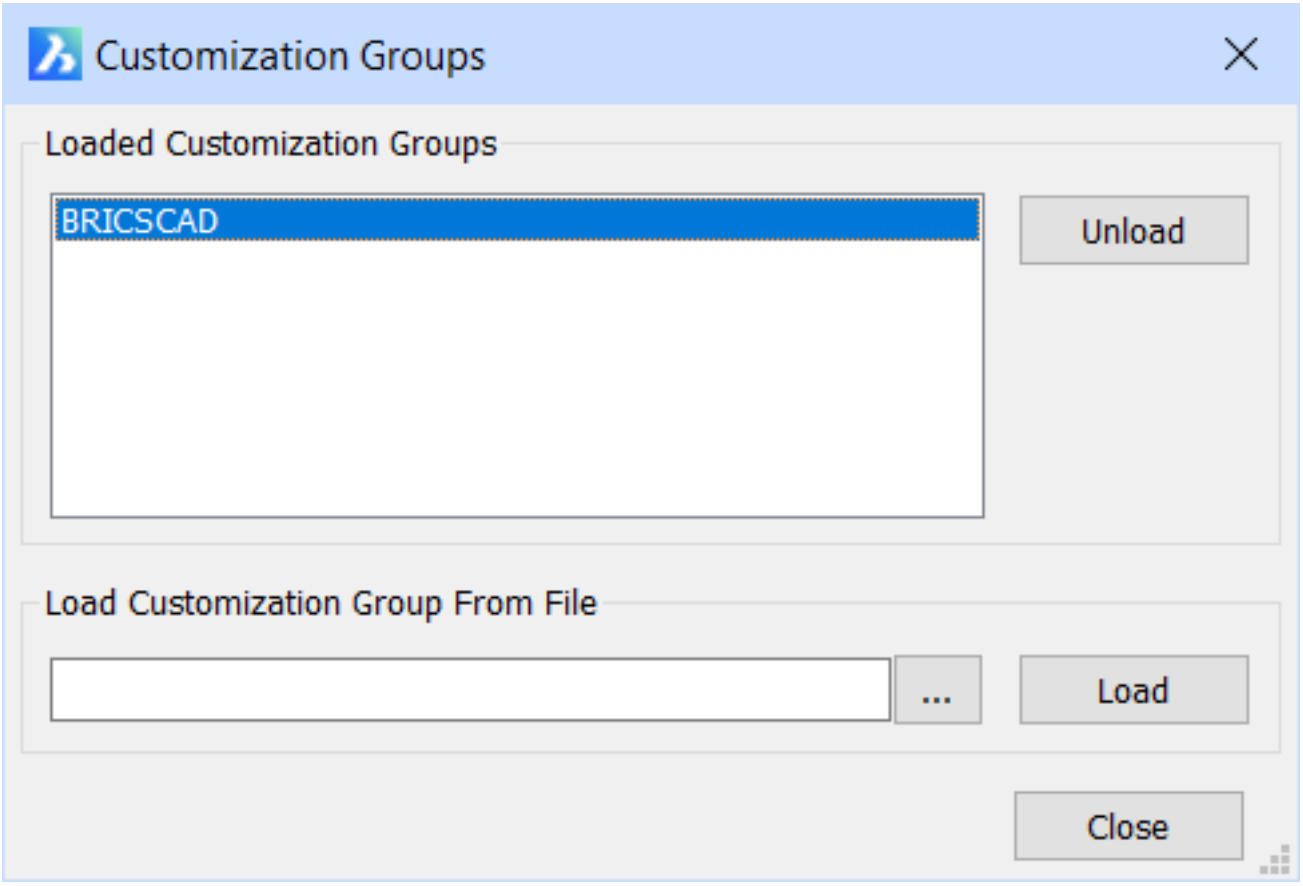
6.21.14 MenuLoad

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads CUIX and CUI (user interface customization), MNU (menu), MNS (LISP code), and ICM (IntelliCAD menu) files into the program.

Accessing the Command

command bar: **menuload** transparent:'**menuload** aliases: **menuunload (6.21.14), cuiunload, cuiload** menu bar:**Tools | MenuLoad** : **MenuLoad** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comm dlg.png})

Figure 6.1356



Command Options

Option	Description
Loaded Customization Groups	Lists the names of customization files loaded in BricsCAD.
Unload	Unloads the selected customization group.**Caution**: unloading 'BricsCAD' causes most user interface elements to disappear.
Browse (...)	Displays the Choose a Customization File dialog box.Choose a CUIX, CUI, MNU, MNS, or ICM file, and then click **Open**.
Load	Loads the opened customization file. **Caution:** loading a customization file changes the program's user interface.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Commands (6.9.37) - lists the names of internal and external commands. CuiUnload (6.9.57) - unloads menu files. Customize (6.9.58) - customizes the user interface. MenuUnload (6.21.15) - loads and unloads menu files. Netload (6.22.3) - loads a .NET application. VbaLoad (6.32.2) and -VbaLoad (6.32.3) - loads VBA projects. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaMan (6.32.4) - manages VBA projects. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.1357



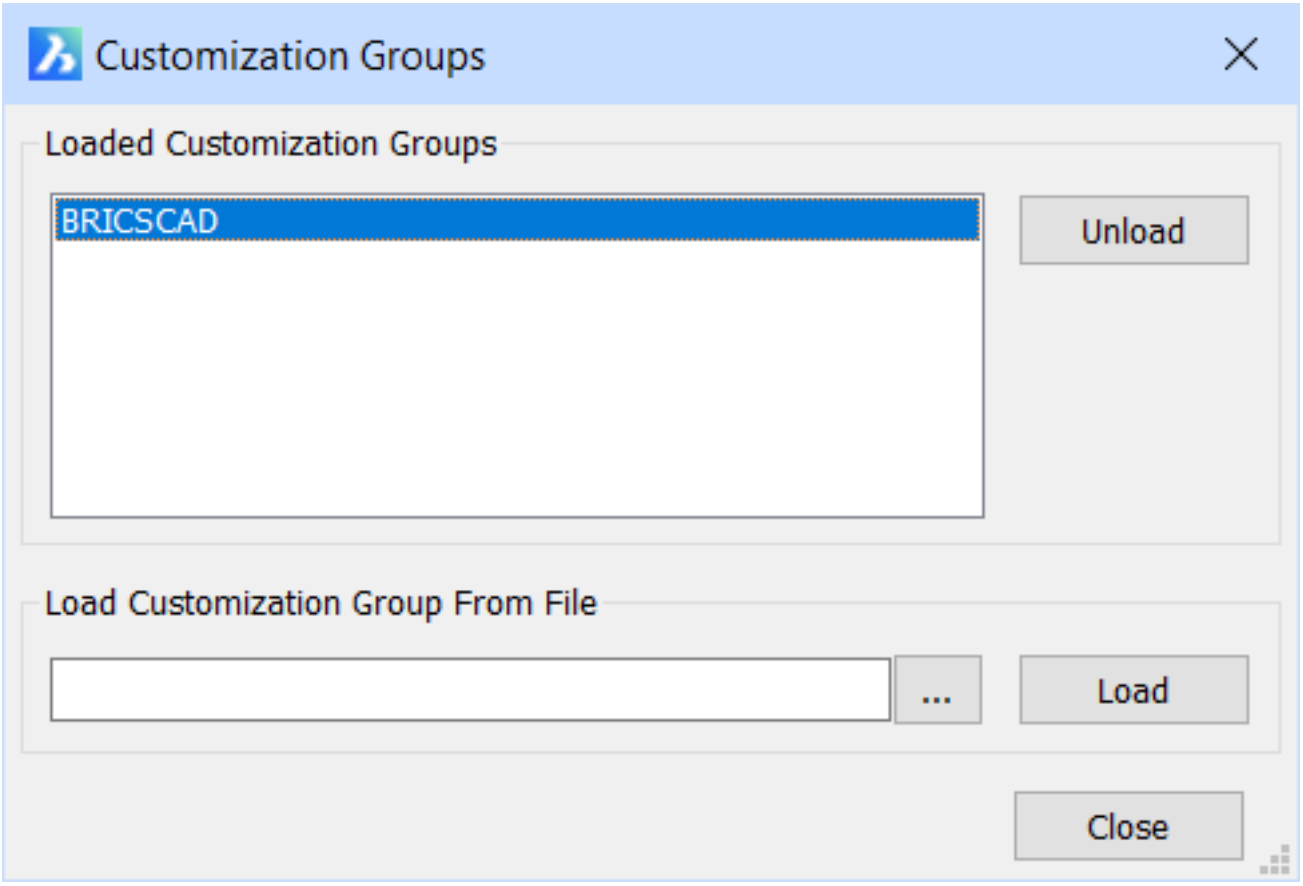
6.21.15 MenuUnload

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads CUIX and CUI (user interface customization), MNU (menu), MNS (LISP code), and ICM (IntelliCAD menu) files into the program.

Accessing the Command

command bar: **menuunload** transparent:**'menuunload** aliases: **menuload, cuiunload, cuiload** menu bar:**Tools | MenuLoad** : **MenuUnload** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comm dlg.png})

Figure 6.1358



Command Options

Option	Description
Loaded Customization Groups	Lists the names of customization files loaded in BricsCAD.
Unload	Unloads the selected customization group.**Caution**: unloading 'BricsCAD' causes most user interface elements to disappear.
Browse (...)	Displays the Choose a Customization File dialog box.Choose a CUIX, CUI, MNU, MNS, or ICM file, and then click **Open**.
Load	Loads the opened customization file. **Caution:** loading a customization file changes the program's user interface.

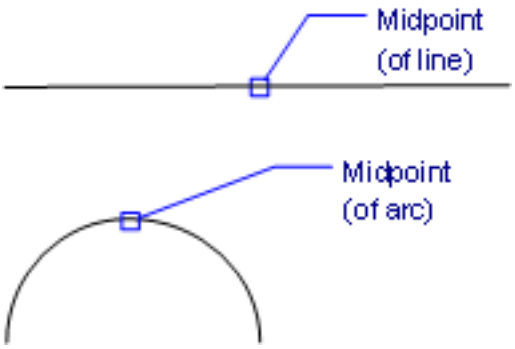
Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Commands (6.9.37) - lists the names of internal and external commands. CuiUnload (6.9.57) - unloads menu files. Customize (6.9.58) - customizes the user interface. Menuload (6.21.15) - loads and unloads menu files. Netload (6.22.3) - loads a .NET application. VbaLoad (6.32.2) and -VbaLoad (6.32.3) - loads VBA projects. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaMan (6.32.4) - manages VBA projects. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.1359



Figure 6.1360



6.21.16 Midpoint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles midpoint entity snap; snaps to the middle of lines, arcs, and other open entities, and to the edges of 3D solids. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Accessing the Command

 command bar: **midpoint** transparent: **mid** ribbon:**Home | Entity Snaps | Snap to Midpoint** toolbar: **Entity Snaps |**  : **Midpoint** Displays no prompt in the command bar. When midpoint entity snap is on, the cursor snaps to the midpoints of entities.This entity snap works with 3D solid models.

Command Options

Option	Description
Midpoint	Entering the command toggles Midpoint entity snap. It snaps to the midpoint of entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode. Zmidpoint (6.35.4) - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge.

Figure 6.1361



6.21.17 MInsert

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Inserts a block as a rectangular array, a combination of the -Insert (6.17.14) and -Array (6.4.15) commands (short for "multiple insertion"); creates a MInsert Block entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image41.png})

Accessing the Command

command bar: **minsert** transparent: **'minsert** : **MInsert Prompts you in the command bar:** ? to list blocks in drawing/~ to open the file dialog/<Block to insert>: **(Type the name of a block, press Enter to accept the name of the previously inserted block, or choose an option)** Insertion point for block or [Multiple blocks/Scale]: **(Pick a point to insert the block, or choose an option)** [Corner] <1.000000>: **(Pick a point, type a value to define the X scale factor, or press Enter to accept the default)** Y scale factor: < Equal to X scale (2.000000)>: **(Pick a point, type a value to define the Y scale factor, or press Enter to set the Y scale equal to the X scale)** Rotation angle for block <0>: **(Pick a point, specify a rotation angle , or press Enter to accept the default)** Number of rows in the array <1>: **(Enter the number of rows)** Number of columns <1>: **(Enter the number of columns)** Vertical distance between rows, or spacing rectangle: **(Specify the distance between rows, or pick two points)** Horizontal distance between columns: **(Specify the distance between columns)**

Command Options

Option	Description
Block to insert	Specifies the name of the block to be inserted.
? to list blocks in drawing	Type **?** to list the names of all blocks in the current drawing. Prompts you: **Block(s) to list [* for all]** - **(press Enter to list all blocks, or enter part of a name to view some blocks; you may use wildcards like ? and *.)** If necessary, press **F2** to view the full list in the Prompt Screen window. See TextScr (6.29.13) command.
~ to open the file dialog	Type **~** the Insert Block dialog box: 1. Choose a DWG file 2. Click **Open**
Multiple blocks	Allows you to insert the same block many times quickly by repeating the following prompt a the end of the command -- until you press **Enter** or **Esc**: **Insertion point for next block** - **(specify a point.)**
Scale	Allows to set the X and Y scale factor equally; prompts you: **Scale factor for block** - **(type a value.)**
Corner	Specifies the size of the block by picking a second corner to show the size. (The insertion point is the first corner.) Prompts you: **Select other corner for scale or enter X scale factor** - **(pick a second corner, or enter a number for the x scale factor.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Y scale factor	Specifies the scale factor of the block in the y direction: • Enter a number larger than 1 to make the block larger • Enter a number smaller than 1 to make it smaller • Enter a negative number to mirror the block • Press **Enter** to make the y scale = x scale
Rotation angle for block	Specifies the rotation angle of the block about its insertion point: • Enter a value • Pick a point to show the rotation angle
Number of rows in the array	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Specifies the number of rows: • Enter a number • For a linear array, enter 1 • To draw the array downwards, enter a negative number
Number of columns	Specifies the number of columns: • Enter a number • For a linear array, enter 1 • To draw the array to the left, enter a negative number.
Vertical distance between rows	Specifies the distance between rows of blocks: • Enter a number for the distance • Pick two points
Spacing rectangle	Specifies two points that indicate the spacing between rows and columns.
Horizontal distance between columns	Specifies the distances between columns of blocks. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • Enter a number for the distance • Pick two points

Procedures

- To insert multiple instances of a block in a rectangular array (1.22.6)

Related Commands

3dArray (6.3.2) - creates 3D arrays. Array (6.4.16) - creates 2D arrays. ArrayPath (6.4.21) - distributes entity copies evenly along a path into multiple rows, and levels. ArrayPolar (6.4.23) - distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. ArrayRect (6.4.25) - distributes entity copies into any number of rows, columns, and levels. Block (6.6.11) - creates blocks. Insert (6.17.13) - inserts blocks.

Figure 6.1362

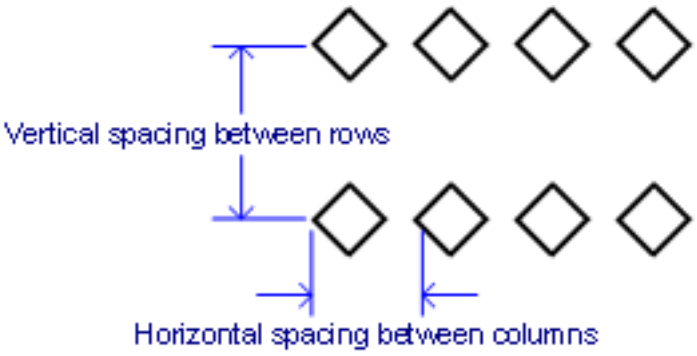


Figure 6.1363

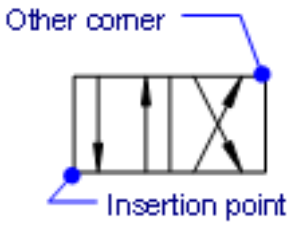


Figure 6.1364

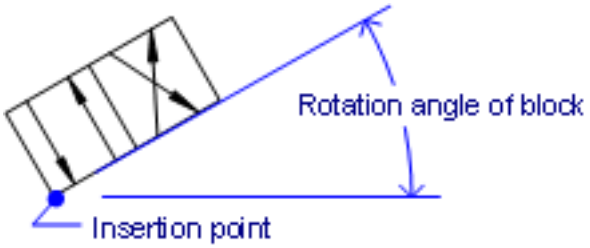


Figure 6.1365

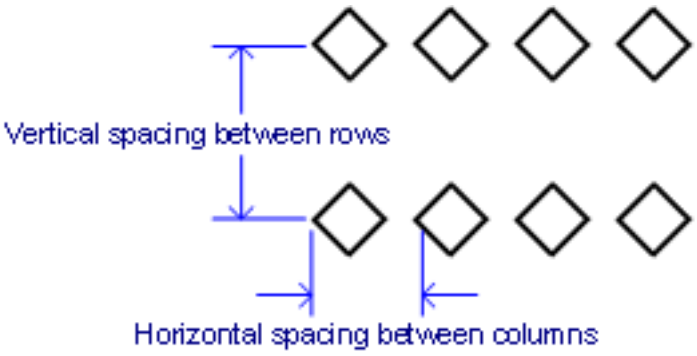


Figure 6.1366



6.21.18 Mirror

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Mirrors entities about a mirror line in a 2D plane.

Accessing the Command

 command bar: **mirror** alias: **mi** menu bar: **Modify | 2D Mirror** ribbon:**Change | Arrange | 2D MIrror** toolbar: **Modify |**  : **Mirror Prompts you in the command bar:** Select entities to mirror: **(Choose one or more entities)** Entities in set: 1 Select entities to mirror: **(Choose more entities, or press Enter to continue)** Start of mirror line: **(Pick a point, or enter x,y coordinates)** End of mirror line: **(Pick another point)** Delete the original entities? [Yes-delete entities/No-keep entities] <No-Keep entities>: **(Type Y or N)**

Command Options

Option	Description
Select entities to mirror	Chose one or more entities to be mirrored.
Start of mirror line	Specifies the start point for the line about which entities are mirrored.This line is temporary; as soon as you pick the second point, it disappears. To mirror vertically or horizontally, hold down the **Shift** key when specifying the second point or use Polar Tracking (1.15.16).
End of mirror line	Specifies the second point of the mirror line.
Delete the original entities?	Determines whether the original entities are erased: • **Y** - erases the original entities, leaving only the mirrored ones • **N** - does not erase the original entities, leaving them in place

Procedures

- Mirroring entities about a line (1.23.13)
- Mirroring entities about a plane (1.23.13)
- To mirror entities using grips (1.23.4)

Related Commands

Mirror3D (6.21.19) - mirrors entities in 3D space. InsertAligned (6.17.15) - inserts mirrored blocks.

Figure 6.1367



6.21.19 Mirror3D

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Mirrors entities about a mirror plane in 3D space.

Accessing the Command

command bar: **mirror3d** aliases: **3m, 3dmirror** menu bar: **Modify | 3D Mirror** ribbon: **Solid | Edit | Mirror | 3D Mirror** toolbar: **Modify |** : **Mirror3D Prompts you in the command bar:** Select entities: **(Choose one or more entities)** Entities in set: 1 Select entities: (**Choose more entities, or press Enter to continue>** Define mirror plane by [Entity/Last/View/Z axis/X-Y plane/Y-Z plane/Z-X plane/3 points] <3 points>: **(Enter an option)** Delete the original entities? [Yes-delete entities/No-keep entities] <No-Keep entities>: **(Type Y or N)**

Command Options

Option	Description
Select entities	Chooses one or more entities to mirror.
Entity	Defines the mirror plane by extrusion direction; prompts you: **Pick an entity that has an extrusion direction** - **(select entities with a thickness.)** This option does not work with flat entities or 3D solids.To give an entity an extrusion, use the Properties command's Thickness option.
Last	Reuses the last mirror plane; does not prompt you.
View	Defines the mirror plane as the current view plane; prompts you: **Point on view plane <0,0,0>** - **(specify the x,y,z coordinates of the view plane.)**
Zaxis	Defines the mirror plane by the z axis and a point on the view plane; prompts you: 1. **Point on view plane** - defines the view plane by a point. 2. **Point on the z-axis (normal) of the plane** - **(specify the height of the view plane.)**
XY	Specifies the mirror plane as the x,y plane; prompts you: **Point on XY plane <0,0,0>** - **(specify the location of the plane through x,y,z coordinates.)**
YZ	Specifies the mirror plane as the y,z plane; prompts you: **Point on YX plane <0,0,0>** - **(specify the location of the plane through x,y,z coordinates.)**
ZX	Specifies the mirror plane as the z,x plane; prompts you: **Point on ZX plane <0,0,0>** - **(specify the location of the plane through x,y,z coordinates.)**
3points	Defines the mirror plane by three points; prompts you: 1. **Second point on plane** - **(specify the second point; the first point is specified by the 3points prompt.)** 2. **Third point on plane** - **(specify the third point.)** Three points define a 2D plane in space.

Delete the original entities?	Determines whether the original entities are erased: • **Y** - erases the original entities, leaving only the mirrored ones. • **N** - does not erase the original entities, leaving all in place.

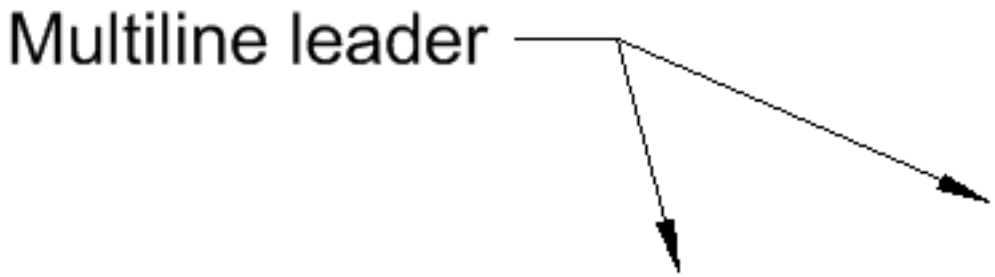
Related Commands

Mirror (6.21.18) - mirrors entities about a mirror line in a 2D plane. InsertAligned (6.17.15) - inserts mirrored blocks.

Figure 6.1368



Figure 6.1369



6.21.20 MLeader

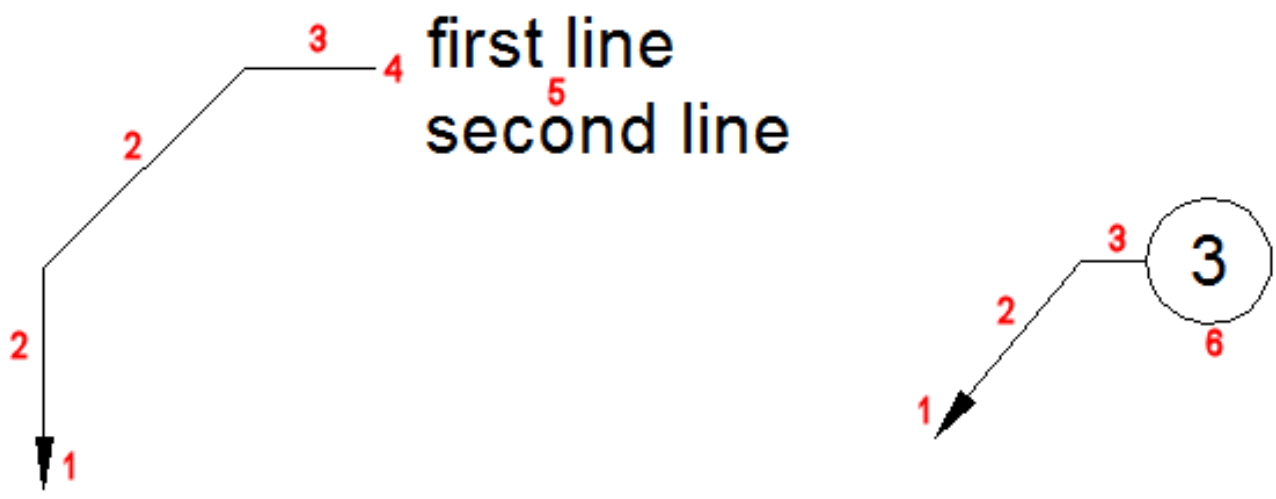
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates multileader entities using the current multileader style. **Notes:**A multileader entity consists of three components: (1) a leader line with an optional arrowhead, (2) a landing line, and (3) the content, such as text or a block.Multileaders support more than one leader line per content, as shown below. Leaders can be added, removed, and content from multiple leaders can be aligned and combined. The look of the mleader is determined by options during this command and by theMLeaderStyle (6.21.25)command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mleader-eg.png})

Accessing the Command

 command bar: **mleader** menu bar: **Dimension | Multileader** ribbon:**Draw | Leaders | Leaders | Multileader** toolbar: **Dimensions |**  : **MLeader When the draw order option is set to "Content first," prompts you in the command bar:** Specify first corner of text or [leader arrowHead first/leader Landing first/Options] <Options>:**(Specify the a corner point for the text or enter an option)** Specify the opposite corner: **(Specify the other corner for the mtext bounding box)** Specify leader arrowhead location: **(Pick the location of the arrowhead; the rest of the leader is drawn automatically)** The prompts appear in a different order when you choose the "leader Landing first" or "leader Arrowhead first" options. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ML

1. Arrowhead

Figure 6.1370



2. Segment
3. Landing
4. Landing gap
5. Mtext
6. Block

Command Options

Option	Description
leader Arrowhead first	Begins drawing the multileader from the arrowhead end; prompts you: 1. **Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>** - **(pick a point or choose an option.)** 2. **Specify leader landing location** - **(pick a point.)** 3. The program launches theMtext (6.21.35) command, in which you enter text. 4. Click **OK** to exit the MText editor, and finish the MLeader command.
leader Landing first	Begins drawing the multileader from the landing, the short horizontal line between the content and the rest of the leader; prompts you: 1. **Specify leader landing location or [leader arrowHead first/Content first/Options] <Options>** - **(pick a point or choose an option.)** 2. **Specify leader arrowhead location** - **(pick a point.)** 3. The program launches theMtext (6.21.35) command, in which you enter text. 4. Click **OK** to exit the MText editor, and finish the MLeader command.
Content First	Begins drawing the multileader from the content end. The prompts shown here are for mtext: 1. **Specify first corner of text or [leader arrowHead first/leader Landing first/Options] <Options>** - **(pick a point for the mtext bounding box, or choose an option.)** 2. **Specify the opposite corner** - **(pick a point.)** 3. The program launches theMtext (6.21.35) command, in which you enter text; click **OK** to exit the MText editor, and the following prompt appears: 4. **Specify leader arrowhead location** - **(pick a point.)**

Options	Sets overrides to the current multileader style; prompts you: **Enter an option [Leader type/leader lAnding/Content type/Maxpoints/First angle/Second angle/eXit options] <eXit options>** - **(enter an option)**: • **Leader type** - sets the leader type: straight, spline or none; prompts you: **Select a leader type [Straight/sPline/None] <Straight>** - **(choose an option)**: **Straight** - the leader is drawn from straight segments **sPline** - a spline is drawn, using the pick points as the control points **None**- no leader is drawn • **leader lAnding** - specifies whether to draw the leader landing, or not; prompts you: **Use landing [Yes/No] <Yes>** - **(enter an option.)** • **Content type** - specifies the content type of the multileader; prompts you: **Select a content type [Block/Mtext/None] <Mtext>** - **(choose an option.) Block** - uses a block; prompts you for the name of the block **Mtext** - uses mtext; displays the MText editor **None** - draws no content • **Maxpoints** - specifies the maximum number of bends in the leader line; prompts you: **Enter the maximum points for leader line <2>** - **(enter a number between 2 and2147483647.)** • **First angle** - constrains the first leader segment; prompts you: **Enter first angle constraint <0>** - **(enter an angle, such as 15 The angle of the first leader segment is constrained to this value, or to a multiple of this angle, such as 0, 15, 30, 45, and so on. Enter 0 for unconstrained leader line.)** • **Second angle** - constrains the second leader segment; prompts you: **Enter second angle constraint <0>** - **(enter an angle, such as 45 The angle of the second leader segment is also constrained to this value or a multiple of it.)** • **eXit options** - exits the Options section of the command

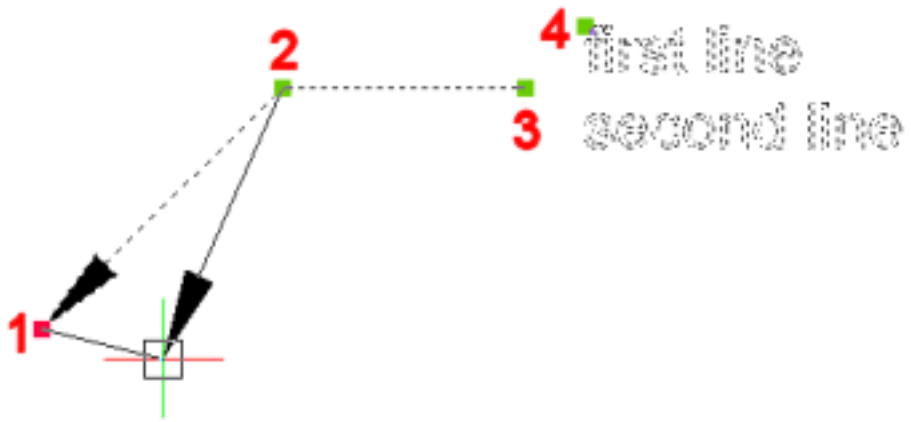
Grips Editing

MLeaders can be edited through grips.

1. Select the MLeader in the drawing. For a single segment leader entity, four grips display: Endpoint (1), Vertex (2), Landing (3) and Text (4).
2. Drag the grips to perform the following actions:

- Move the arrowhead (1)
- Move the landing and the mleader content (2)
- Edit the landing length(3)
- Move text (4)

Figure 6.1371



(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/MLeaderGripEditing.png})

Procedures

- To create a multileader (1.21.14)
- Adding vertices to a leader line of a multileader (1.21.14)
- Removing vertices from a leader line of a multileader (1.21.14)
- Adding leader lines to a multileader (1.21.14)
- Removing leader lines from a multileader (1.21.14)

Related Commands

MLeaderEdit (6.21.23) - adds leader lines to and removes leader lines from a multileader entity. MLeaderStyle (6.21.25) - Creates and manages multileader styles in the Drawing Explorer dialog box. BimTag (6.7.33) - creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model. BmBalloon (6.8.1) - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout. Dimleader (6.11.13) - places leaders with text, blocks, entities, or tolerances. Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. QLeader (6.25.1) - draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.

Figure 6.1372



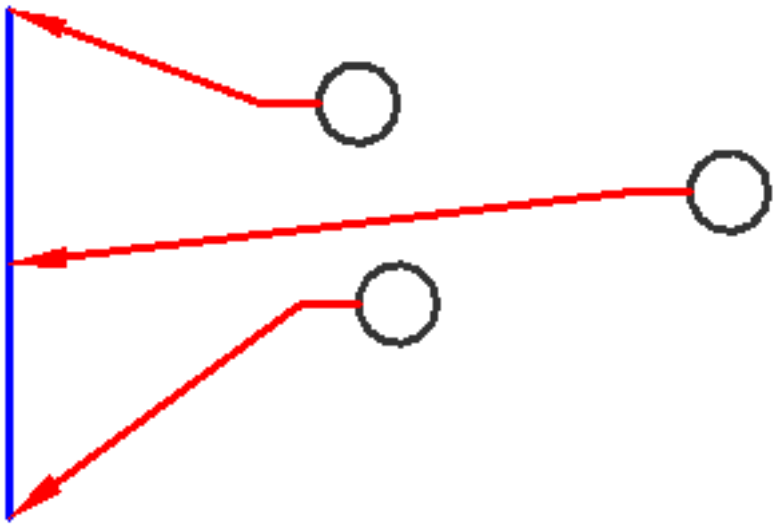
6.21.21 MLeaderAlign

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Aligns the balloons of two or more multileaders.

Accessing the Command


Keyboard: **mleaderalign** Menu: **Dimension | Align Multileader** Toolbar:**Dimension | Align Multileader :MLeaderAlign Prompts you:** Select multileaders: **(Select two or more multileaders, or else enter allto select all of them in the drawing)** Entities in set: 3 Select multileaders: **(Press Enter to finish the selection process and continue with the command)** Specify first point or [Options]: **(Pick an alignment point, or else enter o for options)** Specify second point: **(Pick a second alignment point)** Before: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comm
before.png})

Figure 6.1373



Command Options

Option	Description
Select multileaders	Select two or more multileaders. Enter **all** to select all of them in the drawing
Specify first point	Pick an alignment point
Specify second point	Pick a second alignment point. The blocks at the end of the multileaders will be aligned between the two pick pints.
Options	Prompts you: **Enter an option: [byMleader/onpolyLine/Parallel/Spacing/Circle]:** • **byMleader** - aligns other multileaders to a selected one • **onpolyLine** - aligns multileaders along an imaginary polyline • **Parallel** - aligns other multileaders to one that you select • **Spacing** - prompts you to specify the spacing between multileaders • **Circle** - arrays the multileaders around a circle
byMleader	Aligns other multileaders to a selected one: 1. **Select multileader to align** - **(choose one multileader to which other multileaders will be aligned.)** 2. **Specify direction** - **(move the cursor to indicate the alignment direction.)**

onpolyLine	Aligns multileaders along an imaginary polyline: 1. **Specify first point** - **(pick the point at which the imaginary polyline begins.)** 2. **Specify point** - **(pick a second point; at least two points must be picked.)** 3. **Specify point** - **(pick another point, or else press Enter to end the command.)** As you pick points, the program draws a temporary polyline made of yellow lines so that you can see the path. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/monpolyline.png})
Parallel	Aligns other multileaders to one that you select: • **Select multileader to align** - **(select one multileader, and then the program aligns the others in parallel)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mparallel.png})
Spacing	Prompts you to specify the spacing between multileader blocks: 1. **Specify spacing <0.00>** - **(enter a value or pick two points to indicate the distance.)** 2. **Select multileader to align** - **(select one or more multileaders.)** 3. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/spacing.png})
Circle	Prompts you for the center and radius of a circle around which the align the multileaders: 1. **Specify center** - **(pick the center point of the circle.)** 2. **Specify radius** - **(specify the radius of the circle)** As you indicate the radius, the program draws a temporary circle drawn in yellow so that you can see the path. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mcircle.png})

Related Commands

MLeader (6.21.20) - draws multiline leaders. MLeaderCollect (6.21.22)- collects mleaders with block content into a single mleader. MLeaderEdit (6.21.23) - edits multileaders. MLeaderEditExt (6.21.24) - adds and removes leader lines and vertices from multileaders. MLeaderStyle (6.21.25) - creates and edits named styles for multileaders.

Figure 6.1374

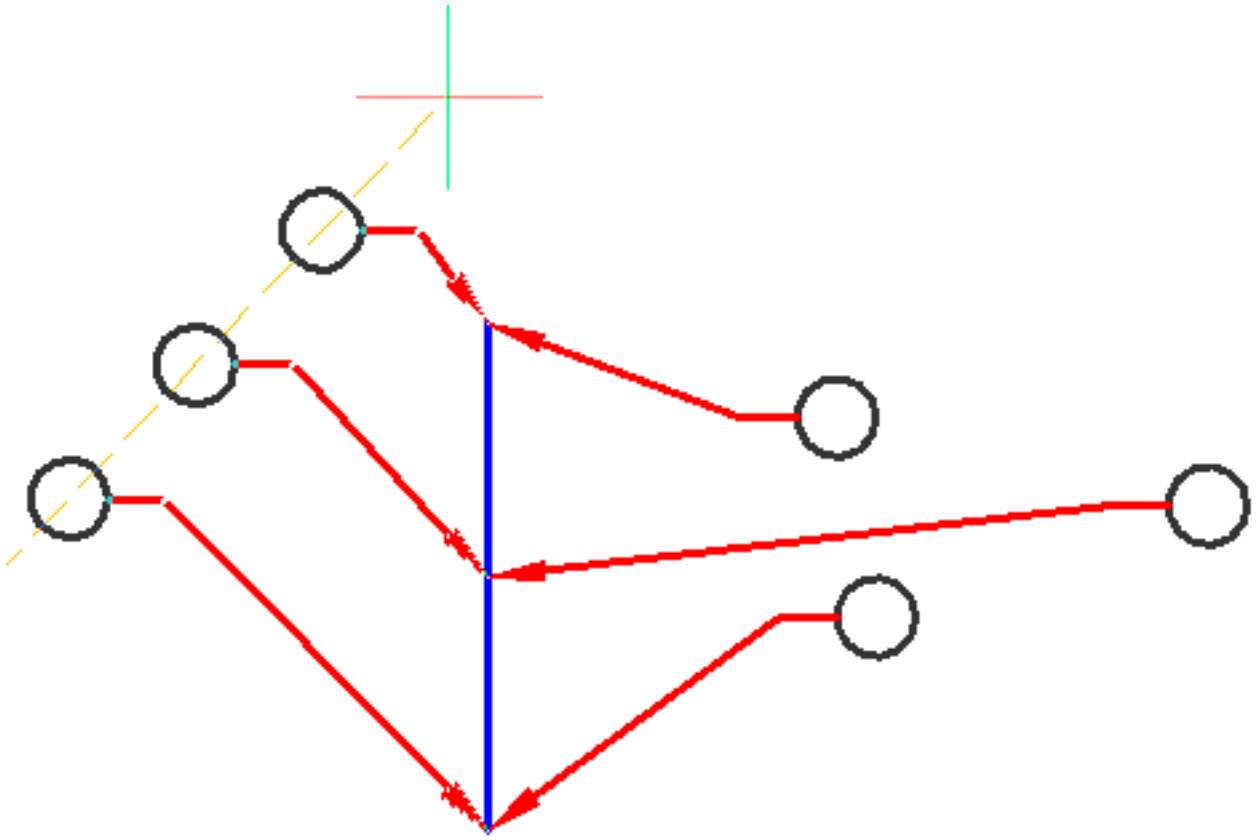


Figure 6.1375

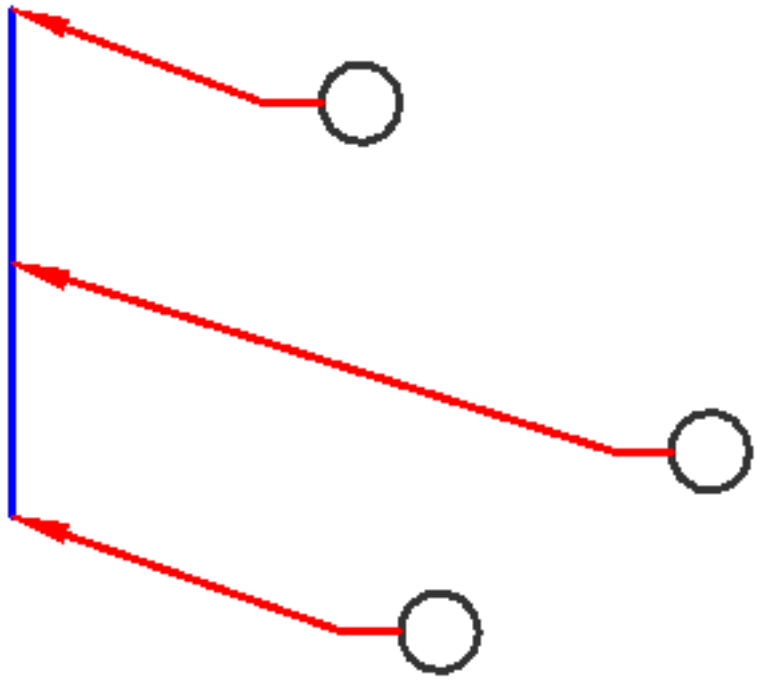


Figure 6.1376

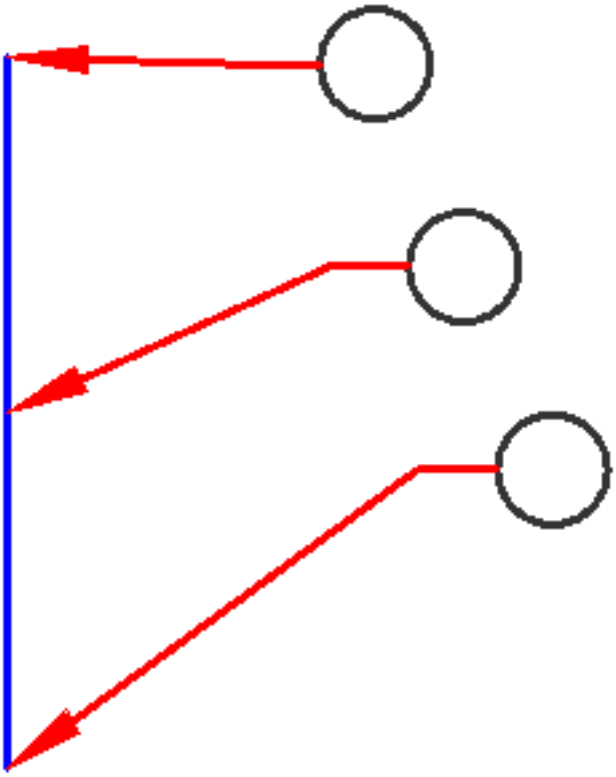


Figure 6.1377

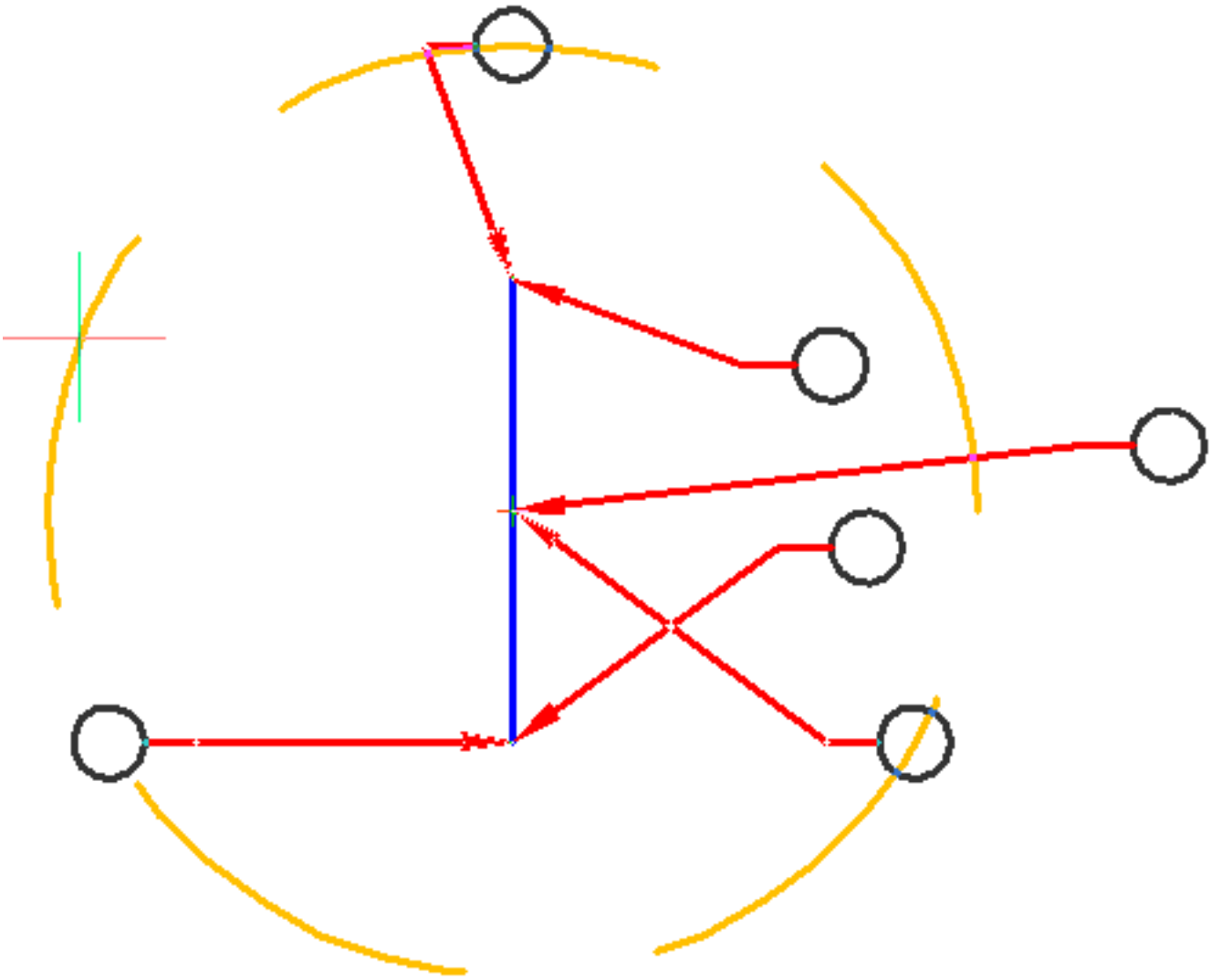


Figure 6.1378



6.21.22 MLeaderCollect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Collects two or more mutliline leaders into a single leader; works only with mleaders that have blocks as content.

Accessing the Command


Keyboard: **mleadercollect** Menu: **Dimension | Collect Multileader** Toolbar: **Dimension | Collect Multileader** **:MLeaderCollect Prompts you:** Select entities: **(Select at least two multileaders, or else enter all to select all of them in the drawing)** Entities in set: 3 Select entities: **(Select more multileaders, or else press Enter to end entity selection)** Specify collected multileader location or [Vertical/Horizontal/Wrap/Sorting/Collapsing]: **(Pick a point, or else enter an option)** BricssCAD collects the callout blocks to a single leader. Before: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mleadercollect-before.png}) After: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mleadercollect-after.png})

Figure 6.1379

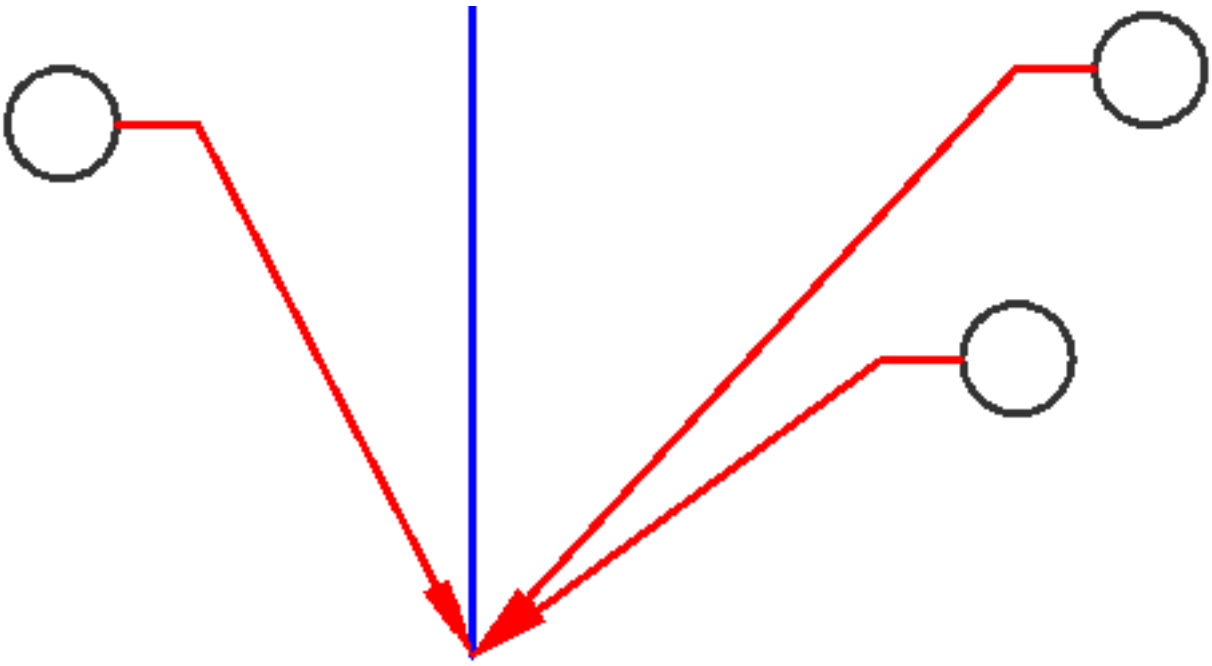
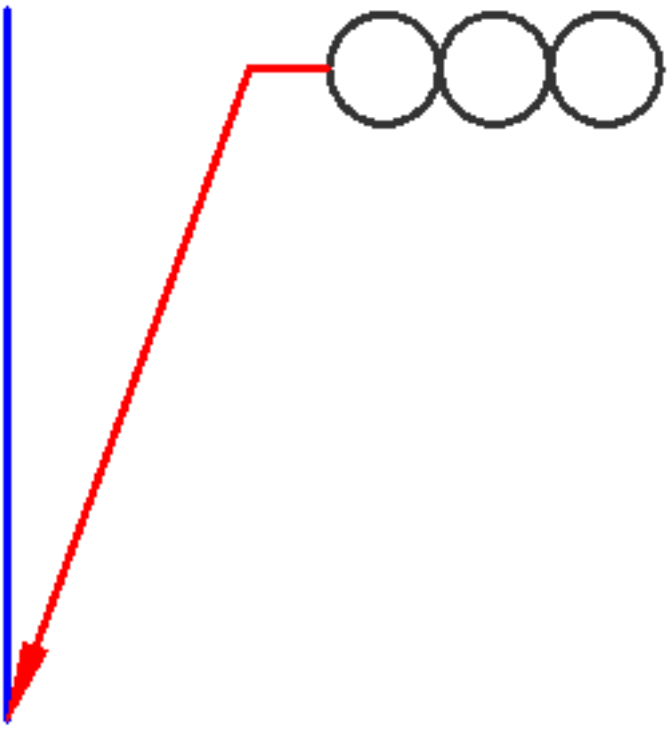


Figure 6.1380



Command Options

Option	Description
Select entities	Select two or more multileaders. Enter **all** to select all of them in the drawing. Press Enter to end entity selection
Specify collected multileader location	Pick at point at which to collect the new multileader
Vertical	Arranges the content blocks vertically
Horizontal	Arranges the content blocks horizontally
Wrap	Arranges the content blocks horizontally to a maximum width, and then places the remainder in the row below
Sorting	Sorts the content by name; prompts you: **Specify mode [None/Ascending/Descending]** - **(enter an option.)** • **None** - does not sort the blocks • **Ascending** - sorts from A to Z • **Descending** - sorts from Z to A
Collapsing	Arranges the content blocks horizontally

Related Commands

MLeader (6.21.20) - draws multiline leaders. MLeaderAlign (6.21.21) - aligns two or multileaders into geometric patterns. MLeaderEdit (6.21.23) - edits multileaders. MLeaderEditExt (6.21.24) - adds and removes leader lines and vertices from multileaders. MLeaderStyle (6.21.25) - creates and edits named styles for multileaders.

Figure 6.1381



6.21.23 MLeaderEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adds and removes leader to and from multileader entities.

Accessing the Command

 command bar: **mleaderedit** menu bar: **Dimension | Edit Multileader - Multileader Add Leaders Dimension | Edit Multileader - Multileader Remove Leaders** ribbon:**Draw | Leaders | Leaders | Edit Multileader** toolbar: **Dimensions** |  : **MLeaderEdit Prompts you in the command bar:** Select a multileader: **(Select an mleader entity)** Specify leader arrowhead location or [Remove leaders]: **(Pick a point or press R)** Specify leader arrowhead location or [Remove leaders]: **(Pick a point, or press Enter to end the command)**

Command Options

Option	Description
Select a multileader	Select a multileader entity
 Add leaders	Adds leaders to the selected mleader entity; prompts you: **Specify leader arrowhead location or [Remove leaders]** - **(choose an action)**: • Pick a another point to locate another new leader • Press **R** to remove leaders • Press **Enter** to end the comma
 Remove leaders	Removes leaders from the selected mleader entity. **Select leaders to remove or [Add leaders]** - **(choose an option)**: • Choose one or more leaders to remove • Press**A**to add leaders • Press **Enter** to end the command

Procedures

- Adding vertices to a leader line of a multileader (1.21.14)
- Removing vertices from a leader line of a multileader (1.21.14)
- Adding leader lines to a multileader (1.21.14)
- Removing leader lines from a multileader (1.21.14)

Related Commands

MLeader (6.21.20) - creates multileader entities using the current multileader style. MLeaderEditExt (6.21.24) - adds and removes leader lines, adds and removes vertices from a multileader entity. MLeaderStyle (6.21.25) - Creates and manages multileader styles in the Drawing Explorer dialog box. BimTag (6.7.33) -Creates associative tags in generated BIM section drawings for the corresponding building entities in a 3D BIM model.

Figure 6.1382



6.21.24 MLeaderEditExt

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adds and removes leader lines, adds and removes vertices, and changes the landing length of a multileader entity (short for ”muliline leader edit extended”).

Accessing the Command

command bar:**mleaderedit**ext quad:(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ : **MLeaderEditExt You are prompt:** Select option [Add vertex/add vertex at ENd/Remove vertex/Stretch vertex/add Leader/rEmove Leader/lengthen LAnding]: **(Enter an option)**

Option	Description
Add vertex	Adds a vertex to the leader line; prompts you: **Select vertex** - **(follow these steps)**: 1. Ignore the wording and instead select a leader line segment 2. Click to select it; a new vertex is added to the selected leader segment 3. Move the vertex, stretching the attached leader lines 4. Click to pick the location of the new vertex
add vertex at ENd	Adds a vertex and a new leader line at the arrowhead; prompts you: 1. **Select a leader of multileader** - **(choose a the leader of multileader.)** 2. **Select vertex** - **(move the cursor to locate the new vertex and leader line.)**
Remove vertex	Removes a vertex from the leader line; prompts you: **Select vertex** - **(move the cursor over a vertex, and then click to remove it. Removing the vertex between the leader and the landing removes the entire mleader.)**

Figure 6.1383



Stretch vertex	Stretch a leader line by moving a vertex; prompts you: **Select vertex** - **(follow these steps)**: 1. Move the cursor over a vertex 2. Click to select it 3. Move the vertex, stretching the attached leader lines 4. Click to pick the new vertex location
add Leader	Adds one or more leader lines to an existing mleader; prompts you: 1. **Select multileader** - **(select a multileader, and a new leader is drawn.)** 2. **Specify leader arrowhead location** - **(move the cursor to locate the arrowhead, and then click to place it.)** Specify leader arrowhead location - the command repeats so that you can add more leaders to this mleader; press **Esc** to end the command.
rEmove Leader	Removes a leader and associated vertex from the leader line; prompts you: **Select vertex** - **(ignore the wording and instead select a leader line segment, which is removed.)**
lengthen LAnding	Changes the length of the landing; prompts you: 1. **Select leader landing point** - **(select a landing; the point you pick is where the landing is lengthened.)** 2. **Specify leader landing location** - **(move the cursor to change the size of the landing, and then click to locate it.)**

Using the Quad is probably the best way to employ this command:

Quad Icon	Tool name	Description
	Add vertex	Adds a vertex to a multileader line.
	Remove vertex	Removes a vertex from a multileader
	Add Leader	Adds an leader line to a multileader.
	Remove Leader	Removes a leader line from a multi-leader.

Procedures

- Adding vertices to a leader line of a multileader (1.21.14)
- Removing vertices from a leader line of a multileader (1.21.14)
- Adding leader lines to a multileader (1.21.14)
- Removing leader lines from a multileader (1.21.14)

Related Commands

MLeaderEdit (6.21.23) - Adds leader lines to and removes leader lines from a multileader entity.

Figure 6.1384



6.21.25 MLeaderStyle

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and manages multileader styles through the Drawing Explorer dialog box.

Accessing the Command

command bar: **mleaderstyle** transparent: **'mleaderstyle** menu bar: **Tools | Drawing Explorer | Multileader Styles** ribbon:**Home | Settings | Drawing Explorer | Multileader Styles Draw | Leaders | Leaders | Multileader Styles** : **MLeaderStyle** Displays the Multileader Style section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mleaderstye-dlg.png}) (\vref {fig:C:/Dev/Task-2448/images/s

1. Arrowhead
2. Segment
3. Landing
4. Landing gap
5. Mtext
6. Block

Figure 6.1385

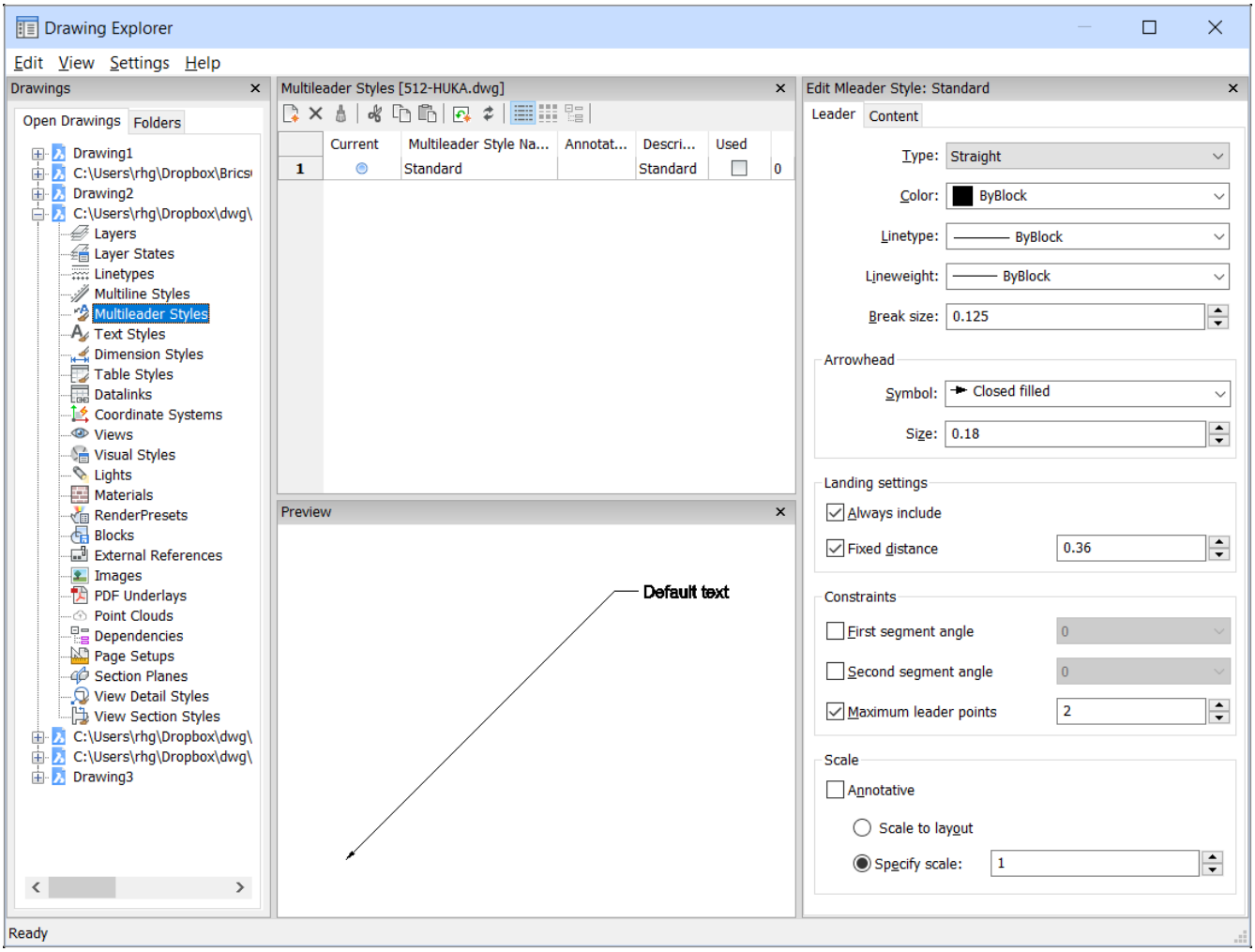
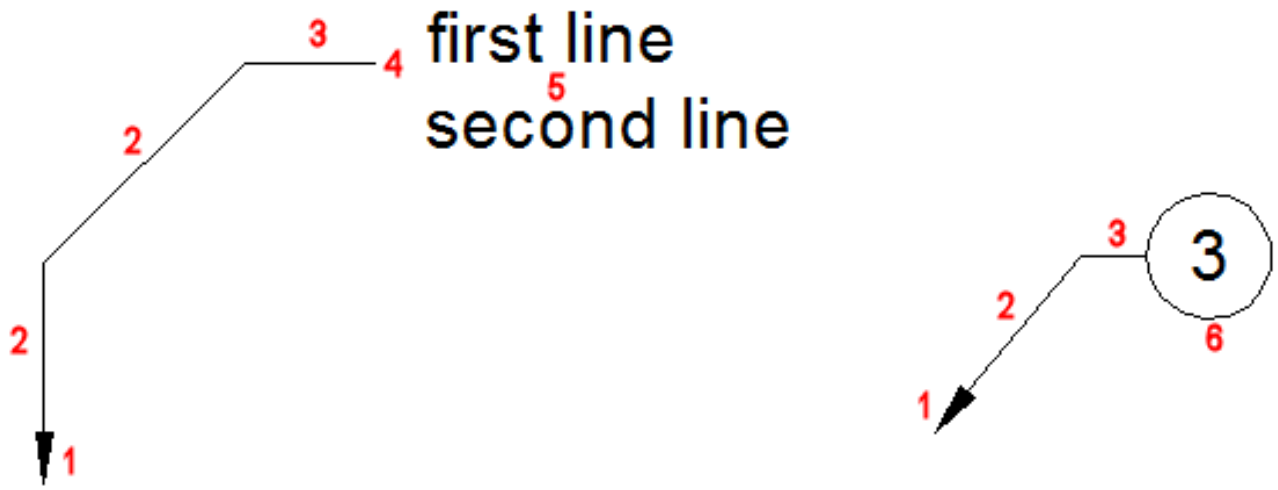


Figure 6.1386



Command Options

Option	Description
New	Creates a new multileader style; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mnewstyle-dlg.png}) 1. Type a name in the **Name** field. 2. (Optional) Select an existing style to create the new one as a copy of it. 3. Click**Create**.
Delete	Deletes the selected multileader style. You cannot delete the Standard style or a style that is being used by the drawing.

Leader Options	Description
Type	Specifies the default style of the leader line: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mleaderoptions.png}) • **Straight** - the leader is drawn from straight segments • **sPline** - a spline is drawn, using the pick points as the control points • **None**- no leader is drawn
Color	Sets the color for the the leader and the arrowhead: • Choose a color from the droplist • Choose **Select Color** to choose a color from the color dialog box
Linetype	Sets the linetype for the the leader: • Choose a linetype from the droplist • Choose **Load**to choose a linetype from the linetype dialog box
Lineweight	Sets the line weight for the the leader; choose a line weight from the droplist
Break Size	(This option is not yet implemented.) Specifies the width of the break when the DimBreak (6.11.7) command is applied to the multileader is selected.

Arrowhead	Determines the appearance of the leader arrowhead: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/m arrowhead.png}) **Symbol** - select an arrowhead style, or choose • **None** - no arrowhead is drawn • **Select arrow**- displays the Select Custom Arrow Block dialog box, from which you select a block that is in the current drawing: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands selectarrowheadblock-dlg.png}) **Size** - type a value to specify the length of the arrowhead (in drawing units) or click the up-down arrows to change the size
Landing Settings	Controls the creation and size of the leader landing: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/m landingsettings.png}) **Always include** - determines if a horizontal segment is drawn between the leader and the content: • **Yes** - the landing is drawn • **No** - the landing is not drawn, the content is placed next to the leader **Size** - type a value to specify the length of the landing (in drawing units) or click the up-down arrows to change the size
Constraints	Determines how the mleader entity is constrained as you draw it: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/m constraints.png}) • **First segment angle** - specifies the angles to which the first leader segment is constrained. For example, enter 15, and the leader is constrained to multiples of this angle, such as 0, 15, 30, 45, and so on. Enter 0 for unconstrained leader line • **Second segment angle** - specifies the angles to which the second leader segment is constrained • **Maximum leader points** - when on, specifies the maximum number of leader vertices; minimum is 2

Scale	Controls the scaling of the multileader entity: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mscale.png}) • **Annotative** - when on, sets the annotative property of the multileader entity • **Scale to layout** - applies a scaling factor relative to model space and paper space viewports, when annotative property is off • **Specify scale** - specifies the overall multileader scale, when annotative property is off
Content Options	**Description**
Content type	Sets the content type of the mleader entity. • **Block** - uses a block; displays options for the block • **Mtext** - uses mtext; displays options for the text • **None** - draws no content and displays no options
Block	Specifies the properties for the block being used as content for the mleader: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mblock.png}) **Source Block** - choose the name of a block from the droplist **Attachment** - determines where the block is attached to the leader: • **Center extents** - attaches the leader to the center of the extents of the block • **Insertion point** - attaches the leader to the block's insertion point (base point) **Color** - specifies the color of the block; **(choose an option from the droplist)**: • **ByBlock** - uses the colors of the block itself • **ByLayer** - applies the colors of the layer on which the block is located • **Color** - choose a swatch from the list • **Select Color** - choose a color from the Color dialog box **Scale** - **(type a value to specify the overall size of the block (in drawing units) or click the up-down arrows to change the size.)**

Mtext: Text Options

Determines the appearance of the mtextbeing used as content for the mleader: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mtextoptions.png}) **Default text** - specifies the default text that will always be included with the mleader;

click the  button to access the Mtext editor. **Text style** - choose a style name from the droplist; to add more styles to the drawing, use the Style (6.27.53) command or copy them from other drawings with the Drawing Explorer **Text color** -specifies the color of the text; chose an option from the droplist:

- **ByBlock** - uses the colors of the block itself
- **ByLayer** - applies the colors of the layer on which the block is located
- **Color** - choose a swatch from the list
- **Select Color** - choose a color from the Color dialog box

Text angle - specifies the angle of the mtext:

- **Always right reading** - text is always right-side up
- **As inserted** - text remains at the angle at which you inserted
- **Keep horizontal** - text is always kept horizontal (0 degrees)

Text height - sets the height of the text (in drawing units) **Left justify** - when on, forces the mtext to always be left-justified **Frame text** - when on, adds a rectangle around the mtext

Mtext: Leader Connection	Determines how the mtext connects to the leader: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mtext-connection.png}) **Attachment type** - specifies where the mtext connects to the leader; choosing Horizontal or Vertical presents different attachment droplists: **Horizontal** attachment type offers Left or Right attachments, so that you can specify where the mtext attaches to the leader separately for the right and left sides: • **Left attachment** - **(select an attachment style from the droplist.)** • **Right attachment** - **(select an attachment style from the droplist.)** **Vertical** attachment type offers Center or Overlines attachments, so that you can specify where the mtext attaches to the leader separately for the top and bottom sides: • **Top attachment** - **(choose Center (text is centered on the mline) or Overline and Center (landing line extends over the top of the mtext).)** • **Bottom attachment** - **(choose Center (text is centered under the mline) or Overline and Center (landing line extends underneath the mtext).)** **Landing gap** - defines the distance between the landing line and the mtext; if the Text Frame option is turned on, then the gap is between the Mtext and the text frame. **Extend leader to text** (this option is available only when attachment type is horizontal) - when checked, the landing line goes all the way to the text, rather than stopping at the mtext bounding box.

Procedures

- To create a multileader (1.21.14)
- Adding vertices to a leader line of a multileader (1.21.14)
- Removing vertices from a leader line of a multileader (1.21.14)
- Adding leader lines to a multileader (1.21.14)
- Removing leader lines from a multileader (1.21.14)

Related Commands

MLeader (6.21.20) - creates multileader entities using the current multileader style. MLeaderEdit (6.21.23) - adds leader lines to and removes leader lines from a multileader entity. Explorer (6.13.14) - opens the Drawing Explorer dialog box. BmBalloon (6.8.1) - creates associative balloon entities for assembly components in Model Space and for their generated views in a Paper Space layout. Dimleader (6.11.13) - places leaders with text, blocks, entities, or tolerances. Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. QLeader (6.25.1) - draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.

Figure 6.1387

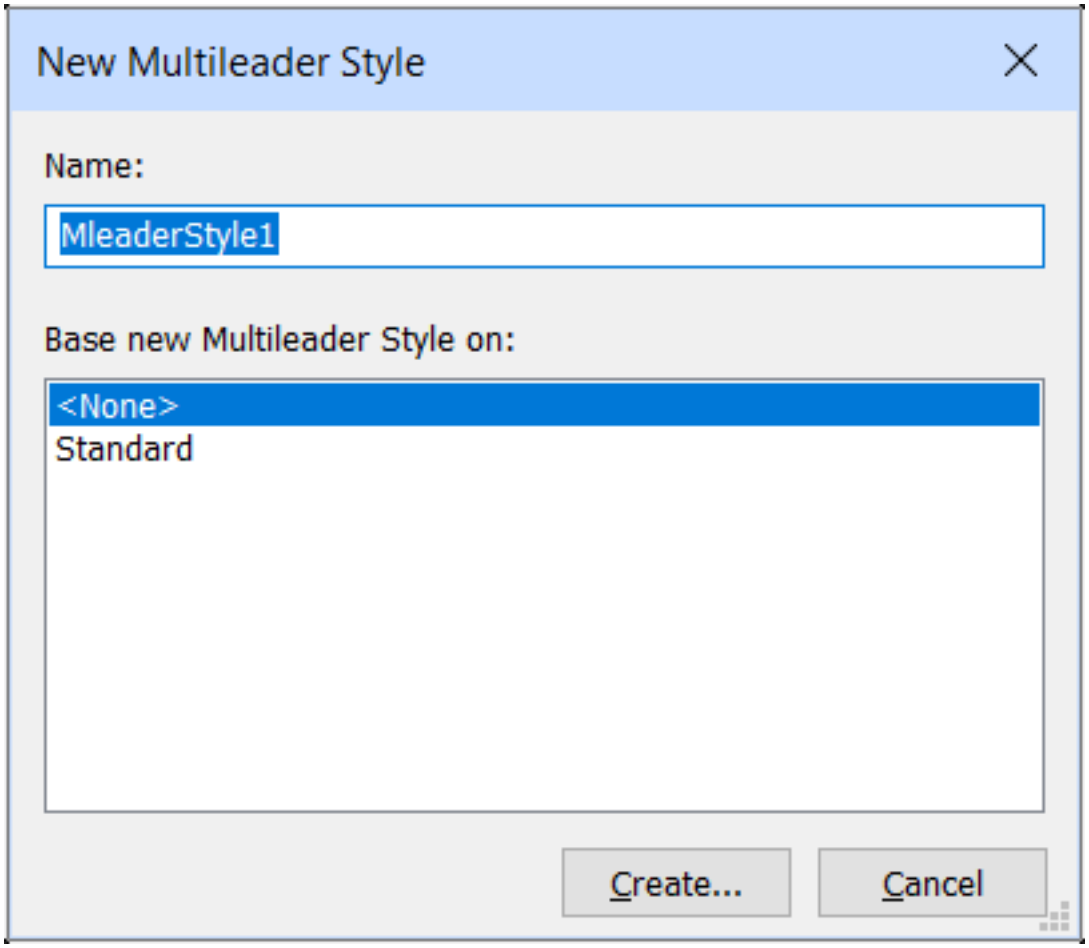


Figure 6.1388

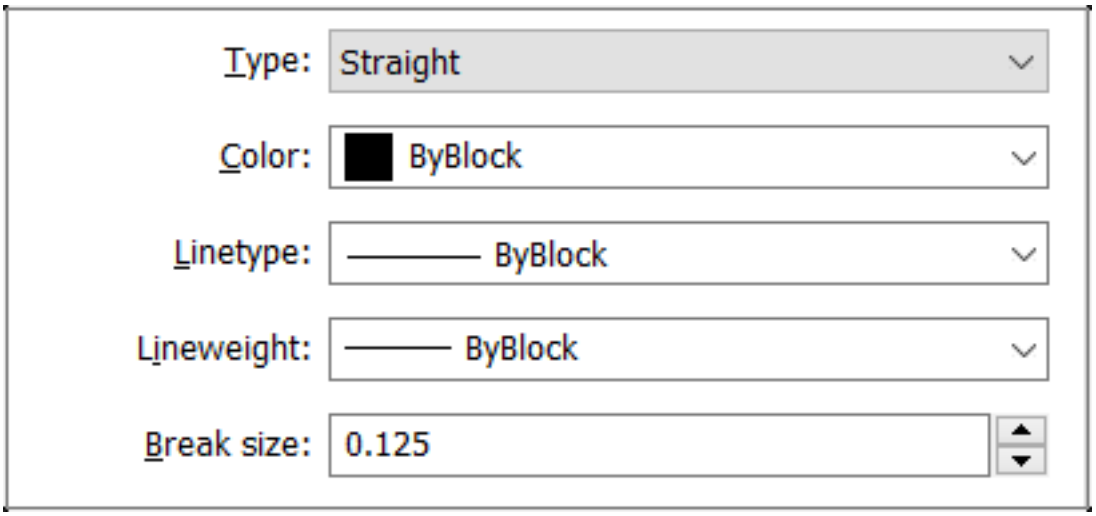


Figure 6.1389

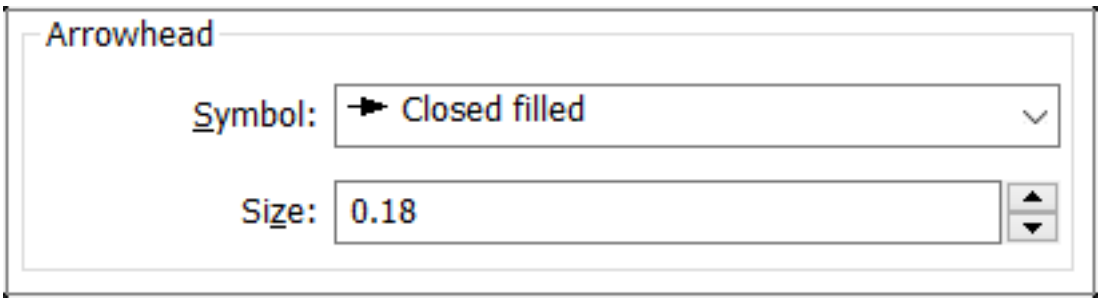


Figure 6.1390

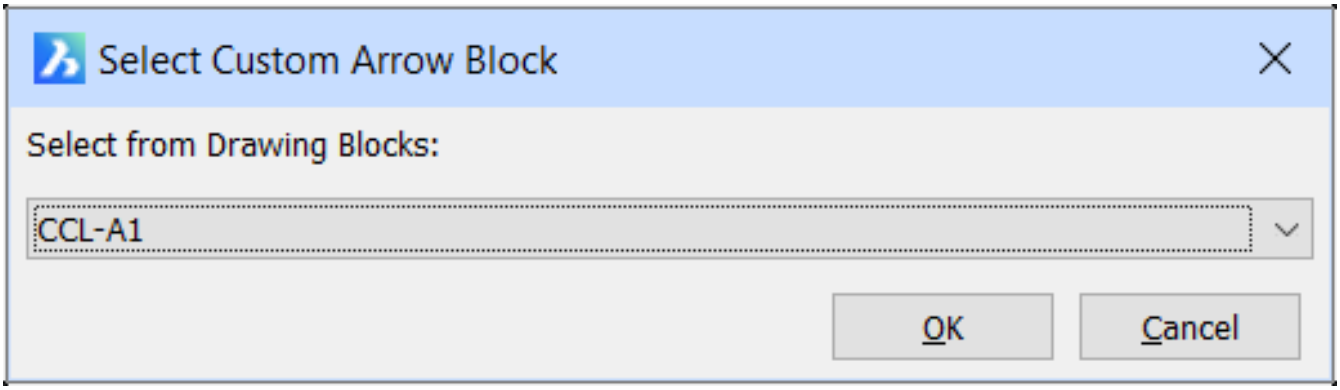


Figure 6.1391

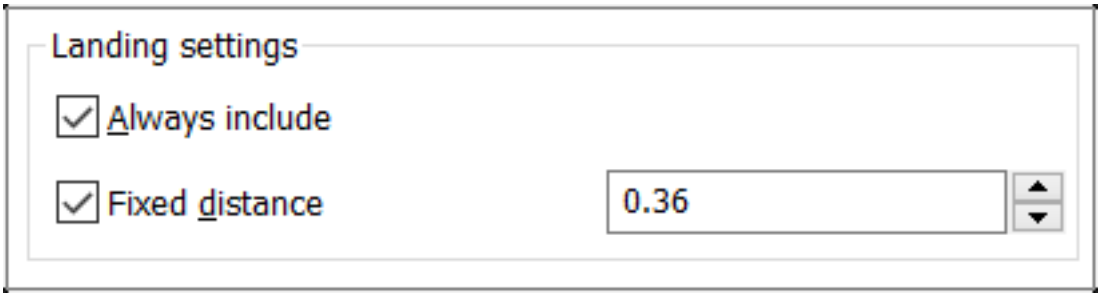


Figure 6.1392

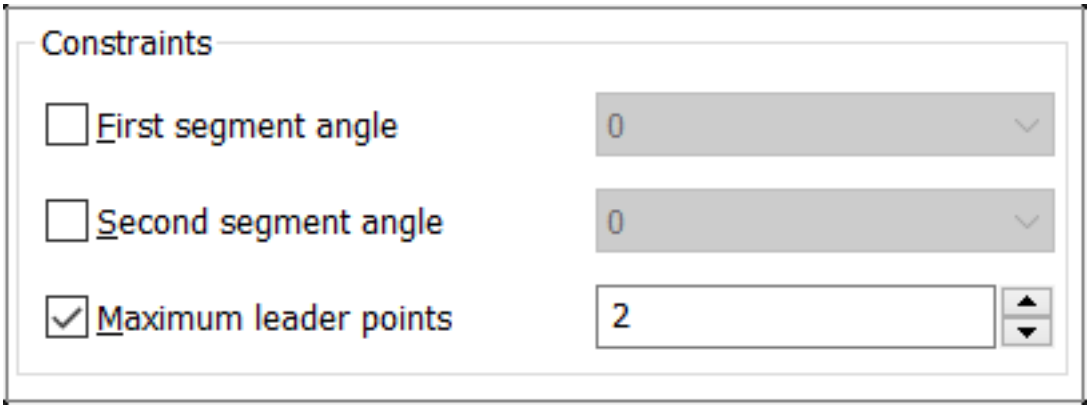


Figure 6.1393

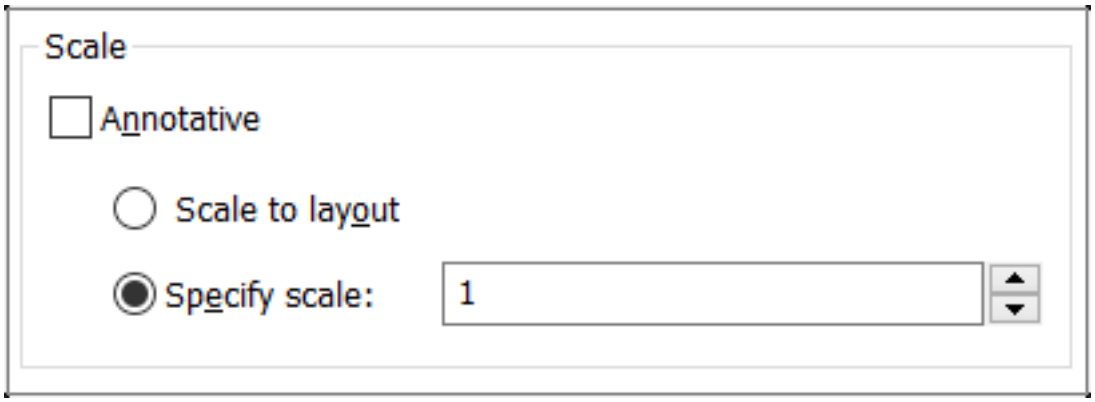


Figure 6.1394

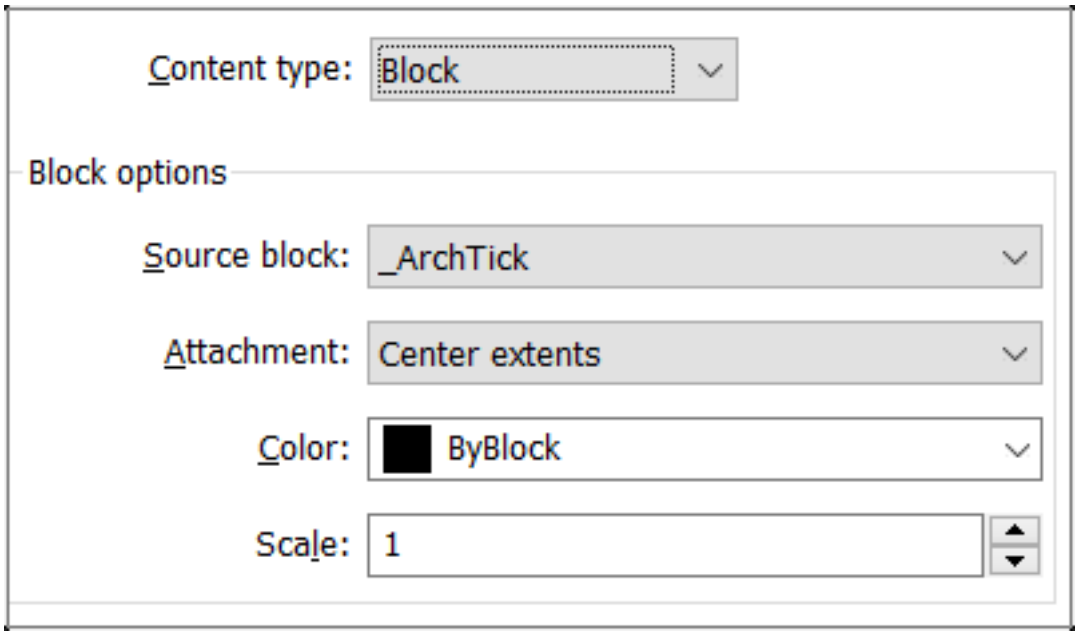


Figure 6.1395

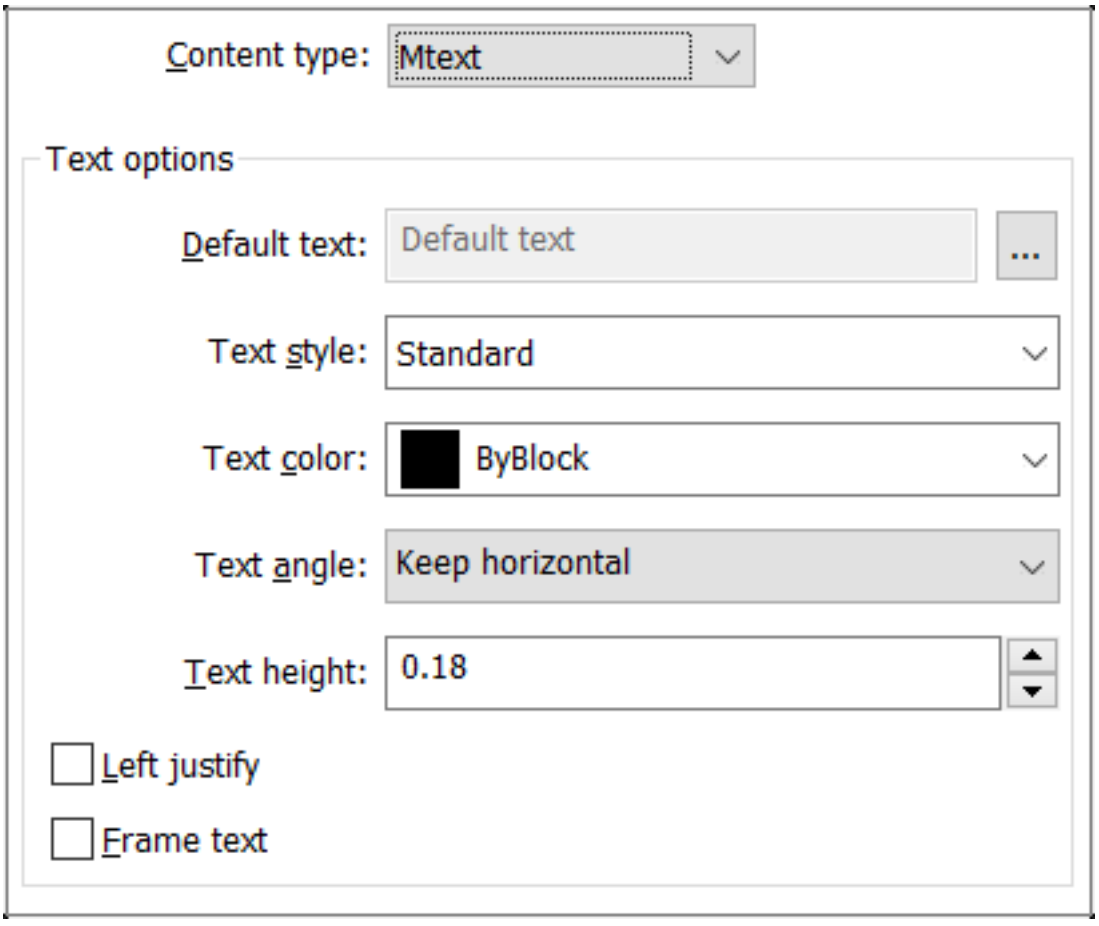


Figure 6.1396

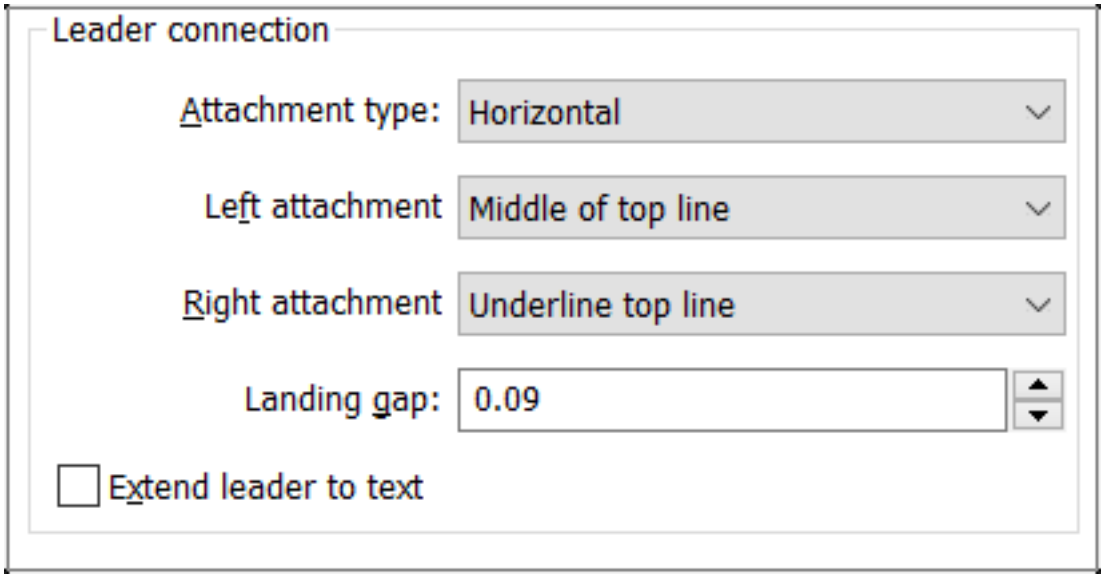
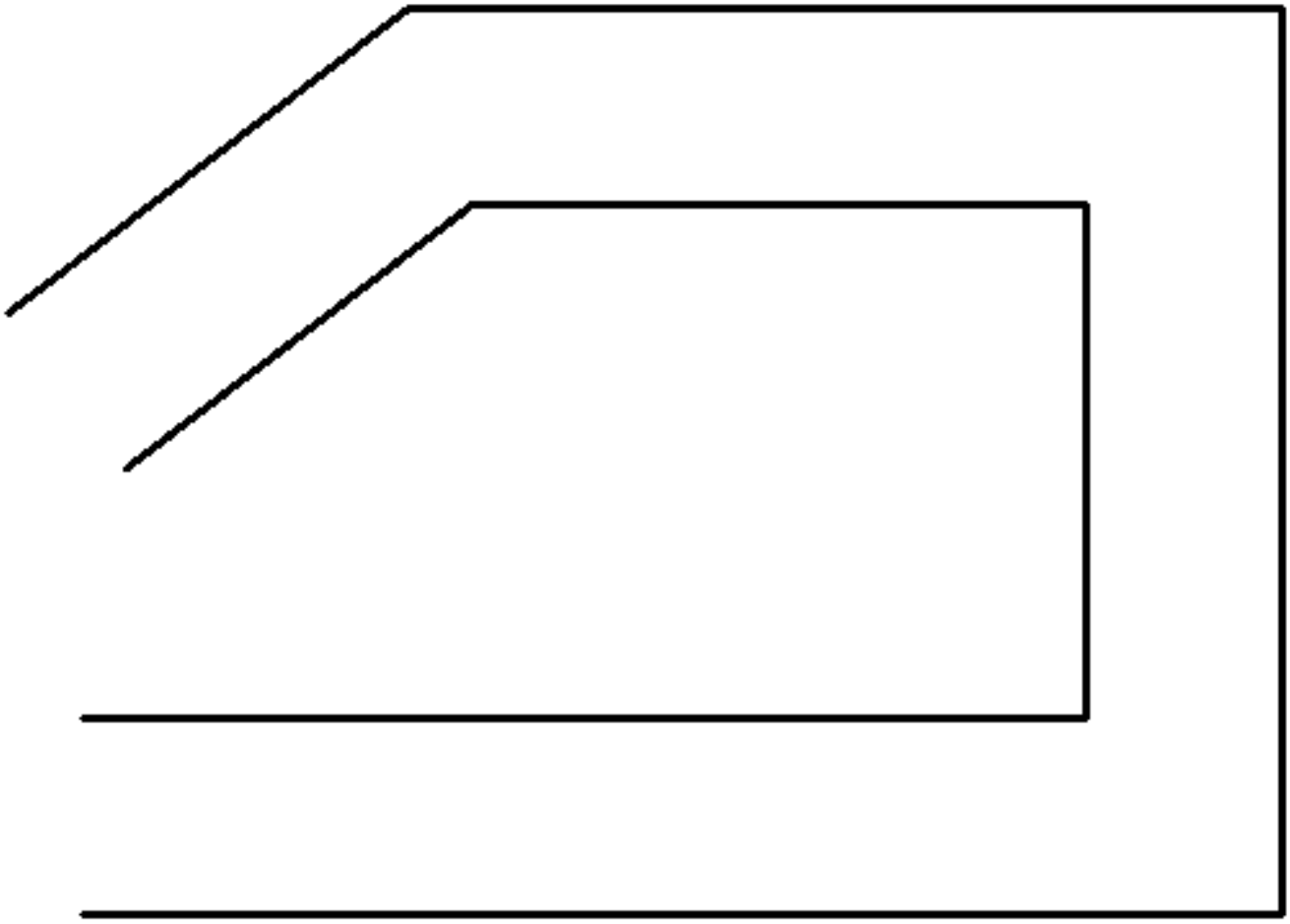


Figure 6.1397



Figure 6.1398



6.21.26 MLine

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws multilines made of two or more parallel lines, according to the style specified by the MlStyle (6.21.27) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mleg.png})

Accessing the Command



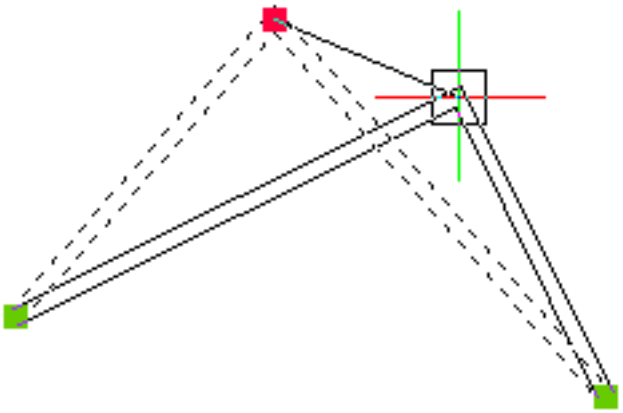
command bar: **mline** alias: **ml** menu bar: **Draw | Multiline** ribbon:**Draw | Draw | Line | Multiline** toolbar:**Draw | Lines| Multiline** : **MLine Prompts you in the command bar:** Start of line or [Justification/Scale/STyle/Follow] <Last point>**: (Pick a point in the drawing, or enter x,y,z coordinates, or else choose an option)** Set end point or [Angle/Length/Undo]: **(Pick another point, or choose an option)** Set end point or [Angle/Length/Follow/Undo]**: (Pick another point, or choose an option)** Set end point or [Angle/Length/Follow/Close/Undo]: **(Pick another point, choose an option, or else press Enter to stop)**

Command Options

Option	Description

Start of line	Specifies the starting point of the first multiline segment; the mline segment does not appear until you click the next point.
Last point	Press **Enter** to use the most recently specified point; this prompt is not seen when this command is first used in a drawing.
Justification	Specify the justification of the multiline relative to the point you pick; prompts you: **Enter MLine justification type [Top/Zero/Bottom] <Top>** - **(enter an option)**: • **Top** - the top of the multiline passes through the points you pick • **Zero**- the center of the multiline passes through the points you pick • **Bottom**- the bottom of the multiline passes through the points you pick The justification is saved inthe CMLJUST (1.13.11)variable.
Scale	Specifies the overall scale of the multiline' prompts you: **Enter MLine scale <1.00>** - **(type a scale factor.)** The scale is saved in the CMLSCALE (1.13.11)variable.
Style	Select the style to use for the multiline, as defined by the MlStyle (6.21.27) command; prompts you: **Enter MLine style name [? to list available styles]** - **(enter the name of a saved multiline style or else type ? to see a list of available styles.)** The current style is saved in the CMLSTYLE (1.13.11)variable.
Follow	Uses the endpoint of the previously drawn multiline, polyline, line or arc as the start point for the next mline segment.The segment is drawn at the same angle of the previous line, multiline or polyline segment or tangent in case of an arc or polyline arc segment.
End point	Specifies the endpoint of the mline segment; this prompt repeats until you press **Enter** to end the command.
Angle	Draws the mline segment at an angle and distance that you specify; prompts you: 1. **Angle of line** - **(specify the angle (measured counter-clockwise from the positive x-axis).)** 2. **Length of line** - **(specify the length of the segment.)**
Length	Draws the segment at a distance and angle that you specify; prompts you: 1. **Length of line** - **(specify the length of the segment.)** 2. **Angle of line** - **(specify the angle (measured counterclockwise from the positive x-axis).)**
Undo	Undoes (erases) the last segment.
Close	Closes the "polygon" by drawing an mline segment between the current endpoint and the first start point.

Figure 6.1399



Grips Editing

1. Select the multiline. Notice that it has two or more grips: at the start and endpoint and at each vertex.
2. Drag a grip to stretch the multiline segment(s).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/MLineGrips.png})

Related Commands

MlStyle (6.21.27) - creates and edits multiline styles. Line (6.20.31) - draws line segments. PLine (6.24.27) - draws connected lines and arcs.

Figure 6.1400



6.21.27 MlStyle

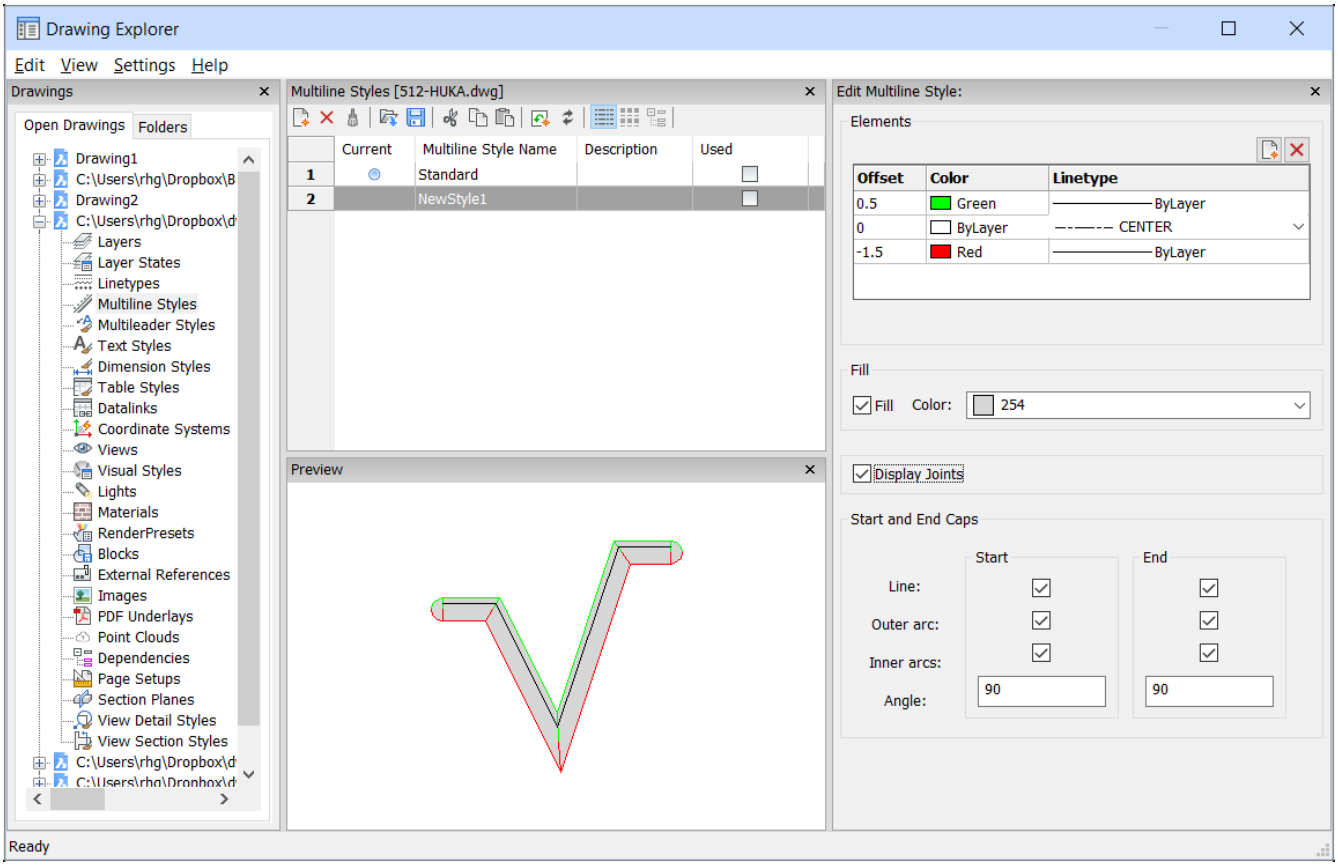
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates, edits, saves, and removes multiline styles used by the Mline (6.21.26) command. **Note:**Some properties defined by this command can be overridden by options in the MLine command.

Accessing the Command



command bar: **mlstyle** transparent: **'mlstyle** menu bar: **Tools | Drawing Explorer | Multiline Styles** ribbon:**Home | Settings | Drawing Explorer | Multiline Styles** : **MlStyle** Displays the Multiline Styles section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mlstyle-dlg.png})

Figure 6.1401



Command Options

Toolbar Option	Description
New	Creates a new multiline definition; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mnew-dlg.png}) 1. Type a name in the Name field. Spaces are not accepted. 2. (Optional) Select an existing multiline style to use as a template for the new style 3. Click**Create** 4. After the New Multiline Style dialog closes, the program adds the new style to the Details pane.(Optional) Type a description in the Description field.
Delete	Deletes the selected style(s).Press and hold the **Ctrl** key to select multiple styles. The 'Standard' style and styles in use cannot be deleted.
Purge	Purges the selected style(s).The 'Standard' style and styles in use cannot be purged.

Load from mln file	Loads multiline styles from a multiline (*.mln) file; displays a file dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mopenmln-dlg.png}) 1. Click **File**. 2. Select an MLN file. 3. Click **Open**. 4. Choose an mline style name in the list. 5. Click **OK**.
Save to mln file	Saves all multiline styles in the drawing to a multiline (*.mln) file for use by other CAD systems; displays a file dialog box: 1. Enter a name for the file. 2. Click **Save**.

Multiline Styles Options	**Description**
Current	The current multline style is marked by the dot.Click the tile in front of a style to make it current.
Multiline Style Names	Lists the names of multiline styles stored in the drawing. To rename it, click on it twice slowly, and then enter the new name.
Description	(Optional) Type a description for the style, such as 'Brick wall'.
Used	Toggles the use of the style in the drawing (this option seems to have no effect): • **Yes** - styles is to be used • **No** - style is not to be used

Edit Multiline Styles Option	**Description**

 Add	Adds a line to the multiline definition using the default properties: • Offset = 0 drawing units • Color = Bylayer • Linetype = Bylayer Once added, edit the table to change its look: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/m elements.png}) **Offset** - specifies how far each line is from the center line of the mline **Color** - specifies the color of each line: 1. Click the color swatch 2. From the Select Color dialog box, choose a color 3. Click **OK** to close the color dialog box **Linetype** - specifies the linetype for each line: 1. Click the droplist 2. Choose a linetype from the list, or else choose Load to access additional linetypes. As you add elements and change properties, the preview pane shows what the mline now looks like: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/m preview.png}) You must add at least one new style to use this option; the Add button does not work with the Standard style.
 Delete	Erases a line from the multiline definition without warning. You must add at least one new style to use this option; the Delete button does notwork with the Standard style.
Fill	When turned on, a solid fill floods the mline between the outer lines: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/m fill.png}) **Color** - choose a color from the droplist, or else choose Select Color for the full palette.
Display joints	When selected, the mline draws joints (diagonal lines) between segments: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
Start and End Caps	Closes the end segments of open multilines by a variety of methods: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M The numbers match the section of the dialog box shown below: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M

Related Commands

Mline (6.21.26) - creates multilines. Explorer (6.13.14) - opens the Drawing Explorer dialog box.

Figure 6.1402

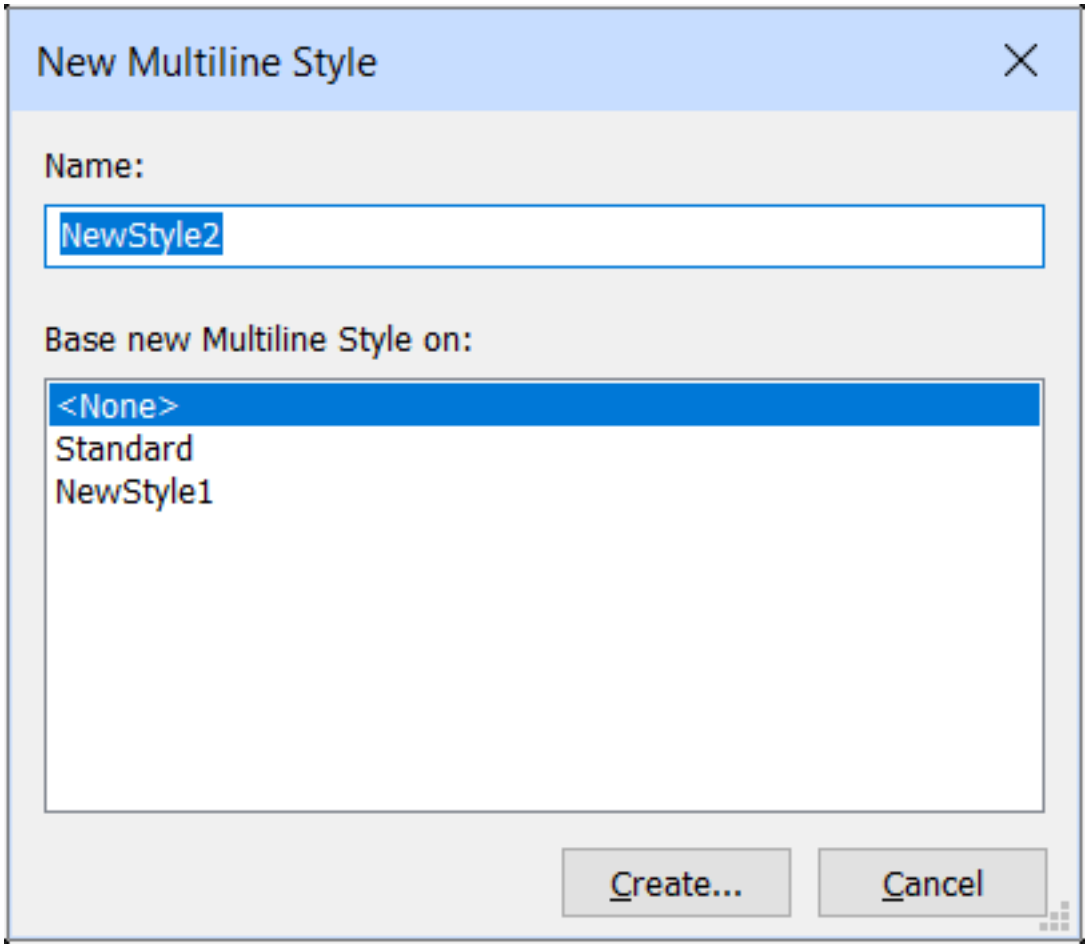


Figure 6.1403

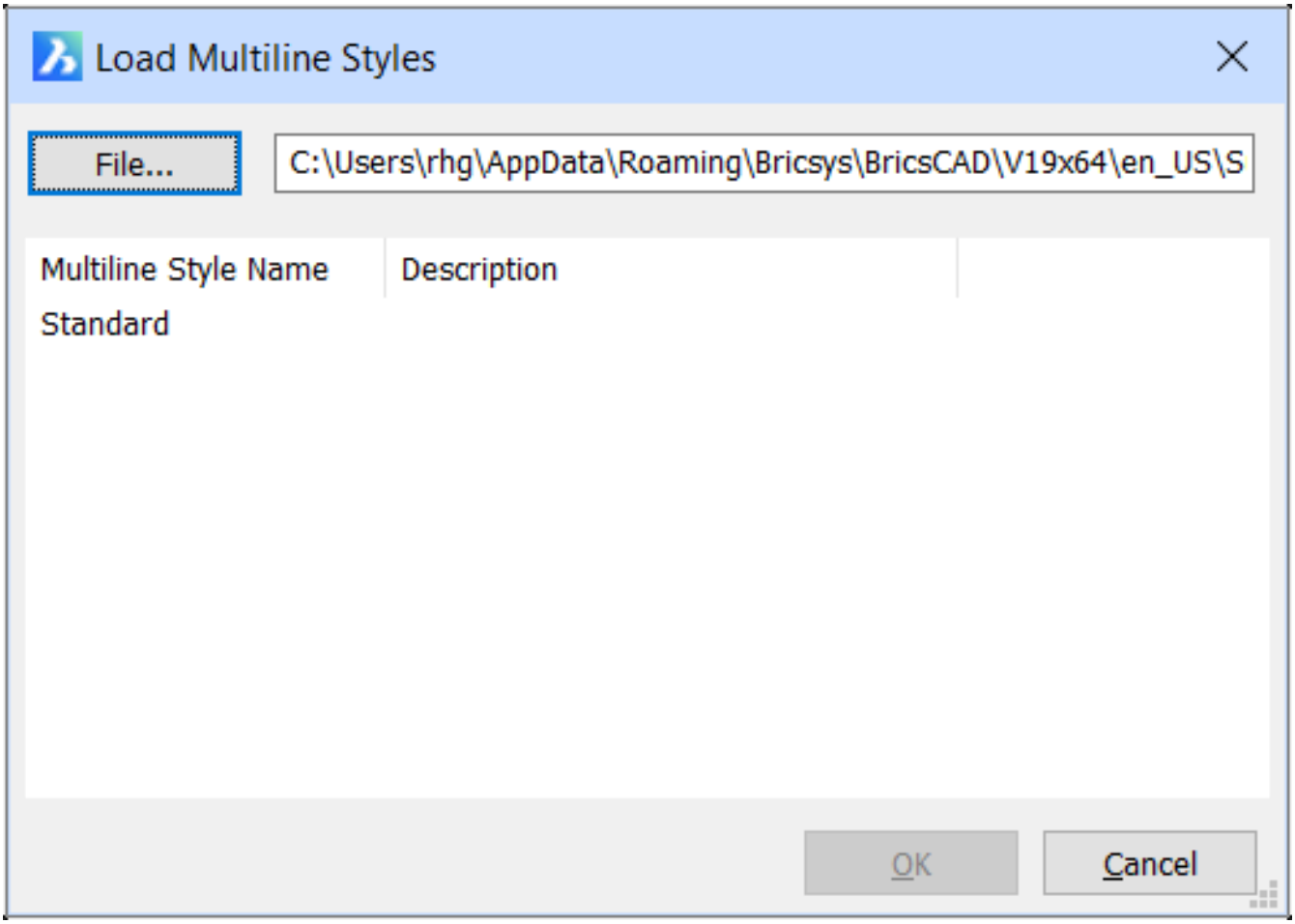


Figure 6.1404

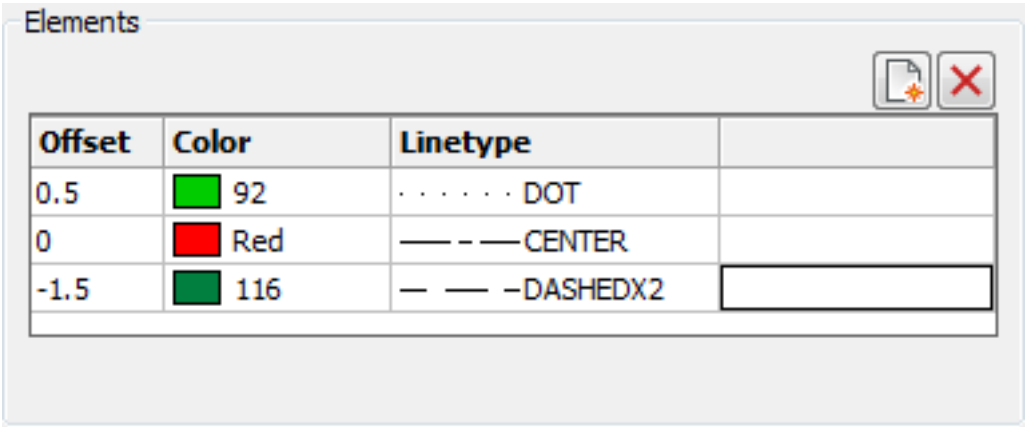


Figure 6.1405

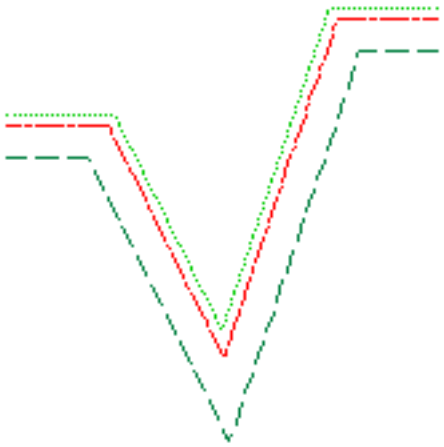


Figure 6.1406

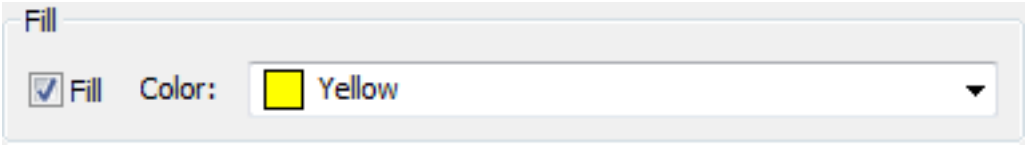


Figure 6.1407

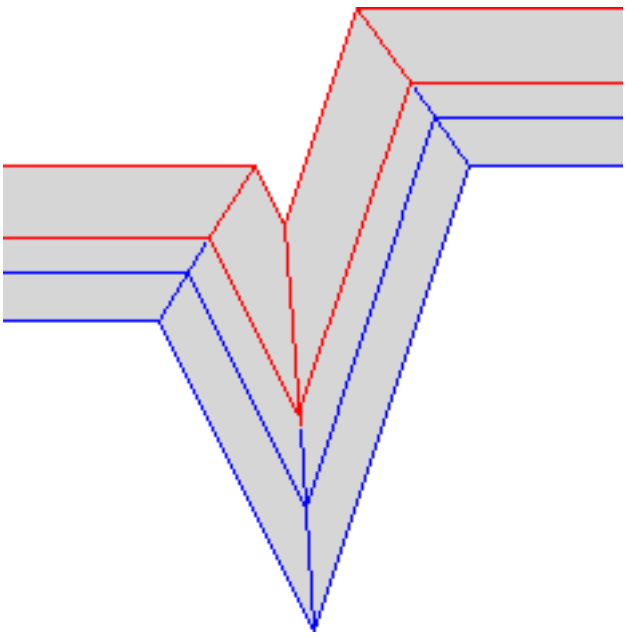


Figure 6.1408

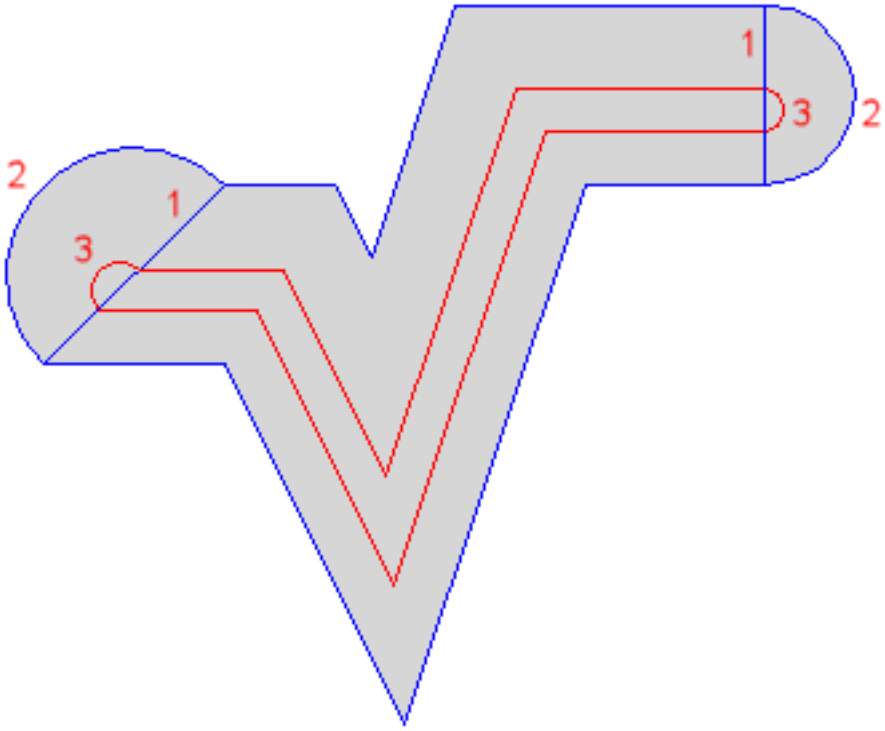


Figure 6.1409

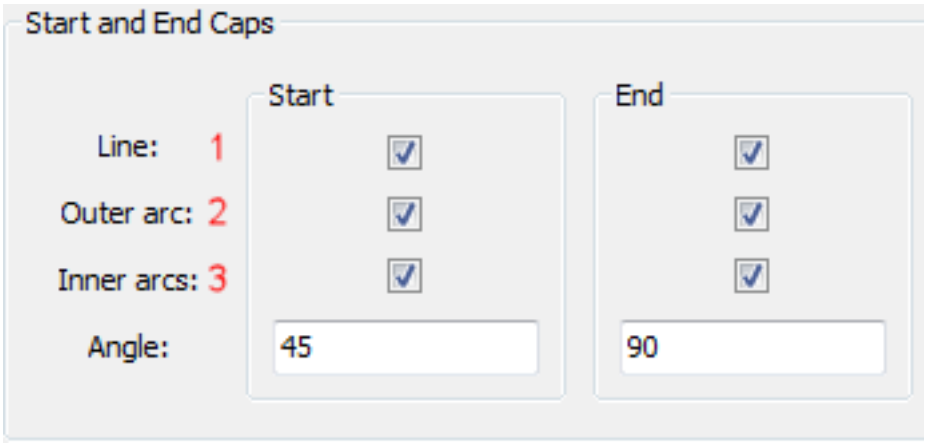


Figure 6.1410



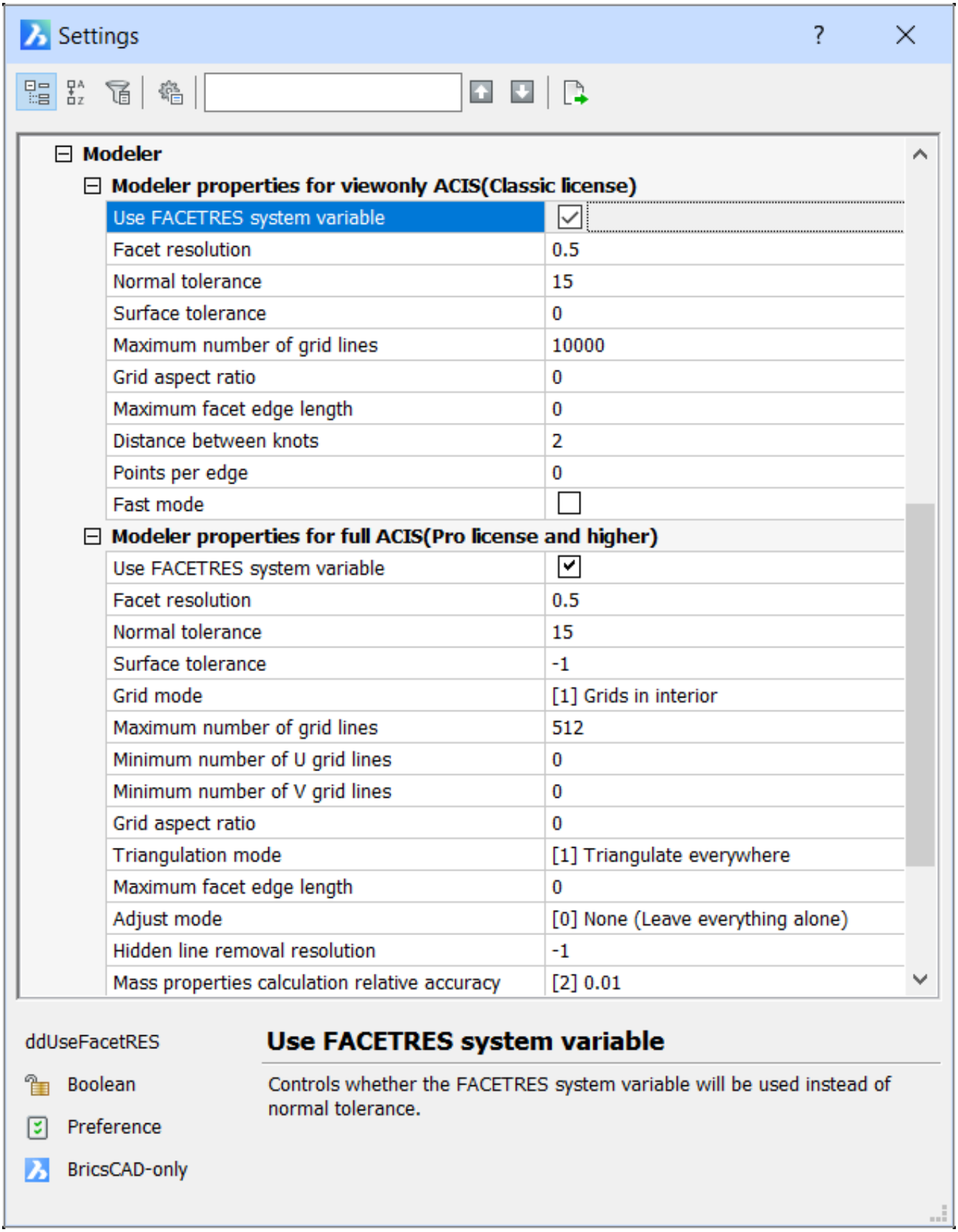
6.21.28 ModelerProperties

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Specifies settings for the ACIS modeler through the Settings dialog box. **Note:** There is a separate section for the Classic version of the program, as it does only views ACIS models and cannot create or edit them.

Accessing the Command

command bar: **modelerproperties** : **ModelerProperties** Displays the Modeler Properties for ACIS section of the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/modelerproeprties-dlg.png})

Figure 6.1411



Command Options

Option	Description
Use FACETRES system variable (editable in Classic)	Toggles the use of thethe FACETRESvariable: • **0** = do not use Facetres • **1** = use Facetres Sets the ddUseFacetRes variable.
Facet Resolution	Determines how smoothly shaded and rendered objects are displayed; range is0.01 to 10. Smaller values render more quickly but more coarsely. Sets the FaceTRes variable.
Normal tolerance	Specifies the maximum normal deviation allowed between normals on two adjacent nodes, in degrees.The proper value is usually independent of the model size Sets the spaNormalTol variable; spa is short for "Spatial," the maker of ACIS.
Surface tolerance	Specifies the maximum distance between a facet edge and the true surface; the value is dependent on the model size. Sets the spaSurfaceTol variable.
Grid mode	Controls how grids are used in the meshing process: • **0** - no grids used • **1** - grids used in the interior only • **2** - grids divide model edges • **3** - grids run in one direction only, either u or v Sets the spaGridMode variable.
Maximum number of grid lines	Specifies the maximum number of grid subdivisions to prevent the facet data of a face from getting too large. Sets the spaMaxNumGridLines variable..
Minimum number of U grid lines	Controls the minimum number of U grid lines: at least this number of grid lines are generated in U direction. Sets the spaMinUGridLines variable.
Minimum number of V grid lines	Controls the minimum number of V grid lines: a least this number of grid lines are generated in V direction. Sets the spaMinVGridLines variable..
Grid aspect ratio	Specifies the approximate aspect ratio of each cell in the grid.If the value is close to 1, then the cell is close to a square. This does not guarantee the aspect ration of the facet, which may consist of only part of a cell. Sets the spaGridAspectRatio variable.
Triangulation mode	Identifies which portion of the mesh is to be triangulated: • **0** - no triangulation is performed • **1** - everything istriangulated • **2** - triangulation goes up to boundaries • **3** - first level grids aretriangulated • **4** - up to three levels of fringe aretriangulated • **5** -up to three levels of fringe aretriangulated Sets the spaTriangMode variable.
Maximum facet edge length	Specifies the maximum length of a side of a cell in object space.As a facet can be not larger than the cell, this determines the maximum facet size. Sets the spaMaxFacetEdgeLength variable.

Adjust mode	Determines how triangles are smoothed: • **0** - no adjustments made • **1** - adjusts points surrounded by triangles by moving ndoes in the center of surrounding nodes • **2** - adjusts all grid nodes Sets the spaAdjustMode variable.
Hidden line removal resolution	Specifies the accuracy to be used for mass properties calculations, this number being is the smallest distance taken into consideration during Hidden Line Removal calculations. it ishighly recommended to use a negative number: • Acceptable range is from 0.01 to 0.000000000001 (1.e-2 to 1.e-12) • Accuracy is relative to the magnitude of the calculated value. • Negative value employs auto-calibration based on the size of the model • Use a value of 0.001 or smaller when dealing with objects of very small sizes Sets the AcisHlrResolution variable.
Mass Properties Calculation Relative Accuracy	Specifies the relative accuracy required for mass properties calculations. Since the specified accuracy is relative, the calculation result depends on the magnitude of the calculated value. A proper value lies between 1.e-2 and 1.e-12. Sets the MassPropAccuracy variable.

Related Commands

-ModelerProperties (6.21.29) - controls the various settings of the Acis modeler through the command bar.

Figure 6.1412



6.21.29 -ModelerProperties

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Specifies settings for the ACIS modeler through the command line. **Note:** The Classic version of the program displays the settings but can only edit the **USe facetres** setting, as it dos not include a full ACIS license.

Accessing the Command

command bar: -**modelerproperties** : -**ModelerProperties Prompts you in the command bar:** [view Properties/USe facetres/Surface tolerance/Normal tolerance/Max grid lines/Edge length/grid aspect Ratio/Adjust mode/Grid mode/Triangula mode/min U grid lines/min V grid lines/mass properties aCcuracy/eXit]<eXit>: **(Select an option) Note:** This command leaves out the FaceTRes and Hidden Line Accuracy settings found in the ModelProperties (6.21.28) command.

Command Options

Option	Description
view Properties	Reports the value of each variable, such as **FACETRES enabled : Yes Surface tolerance : -1" Normal tolerance : 1'-3" Maximum number of grid lines : 512 Maximum facet edge length : 0" Grid aspect ratio : 0.0000 Adjust mode : None Grid mode : Interior Triangulation mode : All Minimum number of U grid lines : 0 Minimum number of V grid lines : 0**
USe facetres (editable in Classic)	Toggles the use of thethe FACETRESvariable; prompts you: **Enable FACETRES: [Yes/No] <Yes>** - **(enter a value)**: • **No** = do not use Facetres • **Yes** = use Facetres Sets the ddUseFacetRes variable.
Surface tolerance	Specifies the maximum distance between a facet edge and the true surface; the value is dependent on the model size; prompts you: **Enter surface tolerance: <-1"> - (enter a value.)** Sets the spaSurfaceTol variable.
Normal tolerance	Specifies the maximum normal deviation allowed between normals on two adjacent nodes, in degrees.The proper value is usually independent of the model size; prompts you: **Enter normal tolerance: <1'-3"> - (enter a value.)** Sets the spaNormalTol variable; spa is short for "Spatial," the maker of ACIS.
Max grid lines	Specifies the maximum number of grid subdivisions to prevent the facet data of a face from getting too large; prompts you: **Maximum number of grid lines: <512> - (enter a value.)** Sets the spaMaxNumGridLines variable.
Edge length	Specifies the maximum length of a side of a cell in object space.As a facet can be not larger than the cell, this determines the maximum facet size; prompts you: **Maximum facet edge length: <0"> - (enter a value.)** Sets the spaMaxFacetEdgeLength variable.

grid aspect Ratio	Specifies the approximate aspect ratio of each cell in the grid.If the value is close to 1, then the cell is close to a square. This does not guarantee the aspect ration of the facet, which may consist of only part of a cell; prompts you: **Grid aspect ratio: <0.0000>** - **(enter a value.)** Sets the spaGridAspectRatio variable.
Adjust mode	Determines how triangles are smoothed; prompts you: **Enter adjust mode [None/non Grid/All]: <None>** - **(enter a value)**: • **None** - no adjustments made • **non Grid**- adjusts points surrounded by triangles by moving nodes in the center of surrounding nodes • **All**- adjusts all grid nodes Sets the spaAdjustMode variable.
Grid mode	Controls how grids are used in the meshing process; prompts you: **Enter grid mode [None/Interior/to Edges/One direction]: <Interior>**- **(enter a value)**: • **None** - no grids used • **Interiror**- grids used in the interior only • **to Edges**- grids divide model edges • **One direction**- grids run in one direction only, either u or v Sets the spaGridMode variable.
Triangulation mode	Identifies which portion of the mesh is to be triangulated; prompts you: **Enter triangulation mode [None/All/fringe 1/fringe 2/fringe 3/fringe 4]: <All>** - **(enter a value)**: • **None** - no triangulation is performed • **All** - everything istriangulated • **fringe 1**- triangulation goes up to boundaries • **fringe 2**- first level grids aretriangulated • **fringe 3**- up to three levels of fringe aretriangulated • **fringe 4**-up to three levels of fringe aretriangulated Sets the spaTriangMode variable.
min U grid lines	Controls the minimum number of U grid lines: at least this number of grid lines are generated in U direction; prompts you: **Minimum number of U grid lines: <0>** - **(enter a value.)** Sets the spaMinUGridLines variable.
min V grid lines	Controls the minimum number of V grid lines: a least this number of grid lines are generated in V direction; prompts you: **Minimum number of V grid lines: <0>** - **(enter a value.)** Sets the spaMinVGridLines variable..
mass properties aCcuracy	Specifies the relative accuracy required for mass properties calculations. Since the specified accuracy is relative, the calculation result depends on the magnitude of the calculated value. A proper value lies between 1.e-2 and 1.e-12; prompts you: **Mass properties calculation relative accuracy: <3/256”>** - **(enter a value.)** Sets the MassPropAccuracy variable.
eXit	Exits the command.

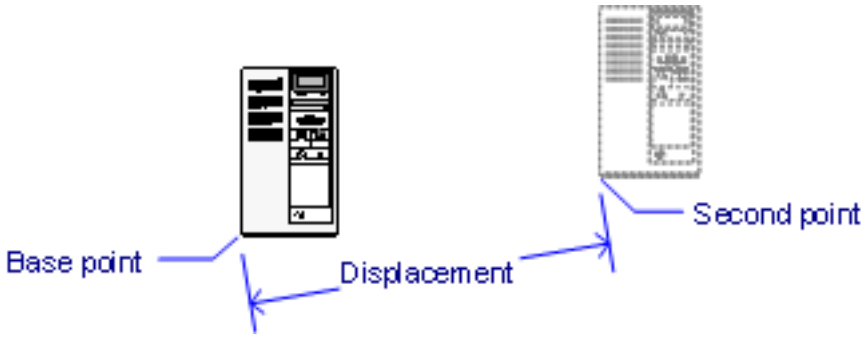
Related Commands

ModelerProperties (6.21.28) - controls the various settings of the Acis modeler through the **Settings** dialog box.

Figure 6.1413



Figure 6.1414



6.21.30 Move

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Moves entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image331.png})

Accessing the Command

 command bar: **move** alias: **m** menu bar: **Modify | Move** ribbon:**Change | Arrange | Move** toolbar: **Modify** |  : **Move Prompts you in the command bar:** Select entities to move: **(Choose one or more entities)** Entities in set: 1 Select entities to move: **(Choose more entities, or press Enter to continue)** Enter base point [Displacement] <Displacement>: **(Pick a point, enter x,y coordinates, or type D)** Enter second point <Use base point as displacement>: **(Pick another point, or press Enter)**

Command Options

Option	Description
Select entities to move	Specifies the entities to be moved.
Enter base point	Specifies the starting point: • Pick a point in the drawing • Enter x,y,z coordinates The z-coordinate is optional. If not specified, then z = 0. Use entity snap modes to ensure accuracy.To move entities horizontally or vertically, hold down the **Shift** key.
Enter second point	Specifies the destination for the entities: • Pick a point in the drawing • Enter x,y,z coordinates
Displacement	Specifies the displacement vector to move the entities; prompts you: **Enter displacement vector <100,200,0>** - **(enter x,y,z coordinates to specify the displacement vector from the origin of the current UCS (user-defined coordinate system).)** "Vector" is fancy term for distance and angle being specified at the same time.

Use base point as displacement	Uses the x,y,z coordinates of the base point (from the drawing origin of 0,0,0) as the displacement vector.

Procedures

- Using the Move command (1.23.11)
- Moving entities using grips (1.23.11)
- Using Cut and Paste to move entities between drawings (1.23.11)
- Move a selection as a block (1.23.11)
- To move entities using grips editing (1.23.4)

Related Commands

PEdit (6.24.20) - moves the vertices of polylines. CutClip (6.9.59) - moves entities from the drawing to the Clipboard.

Figure 6.1415



6.21.31 MSlide

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates slides (just like a screen grab) of the current drawing (short for "make slide") and then saves them to disc in these formats:

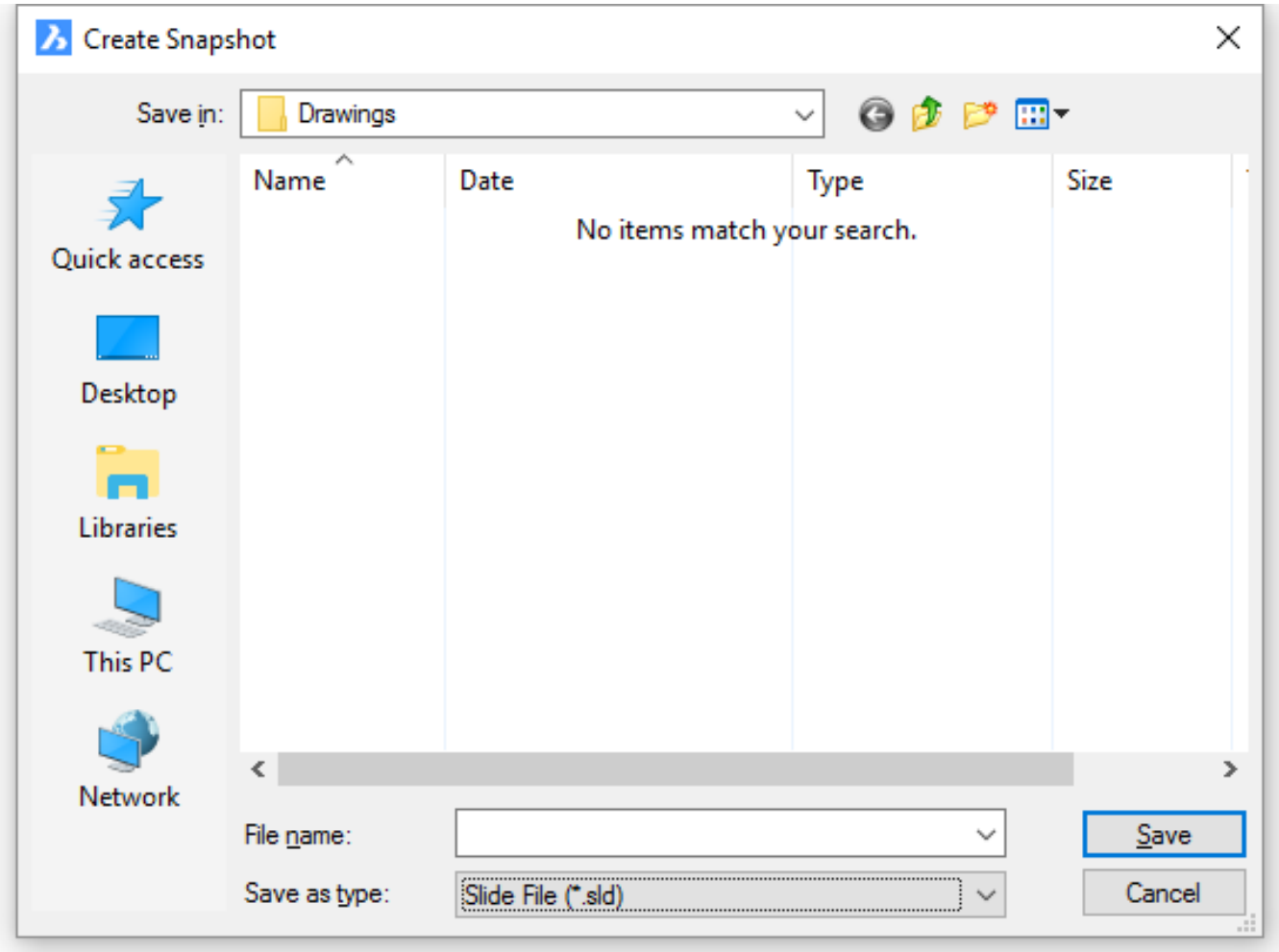
- **SLD** - compatible with other CAD programs
- **EMF** - enhanced WMF format
- **WMF** - Windows meta format

Notes: This command on Linux exports only to the Slide File (*.sld)format.To view slide files made with this command, use the VSlide (6.31.22) command.

Accessing the Command

 command bar: **mslide** alias: **msnapshot** menu bar: **Tools | Make Snapshot** ribbon:**Tools | Action Recorder | Make Snapshot** toolbar: **Tools |**  : **MSlide** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel
Enter a file name, choose a format, and then click **Save**.

Figure 6.1416



Command Options

Option	Description
File name	Specifies the name of the slide (snapshot).
Save as type	Specifies the file format: • **SLD** - compatible with other CAD programs • **EMF** - enhanced WMF format • **WMF** - Windows meta format
Save	Saves the slide file.

Related Commands

VSlide (6.31.22) - views WMF, EMF, and SLD files. BmpOut (6.6.14) - exports drawings in BMP format. Export (6.13.15) - exports drawings in BMP, DWF, DXF, DWT, PDF, WMF, EMF, SLD, and older DWG formats. WmfOut (6.33.12) - exports drawings in WMF, EMF, and SLD formats.

Figure 6.1417



6.21.32 MoveEData

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Moves extended entity data from one entity to one or more others.

Accessing the Command

 command bar: **moveedata** transparent: **'moveedata** menu bar: **Tools | Entity Data | Move Entity Data** toolbar: **Entity Data |**  : **MoveEData Prompts you in the command bar:** Move Entity Data: Name of application or [? to list application names]: **(Enter name of application, or type ?)** Select entity from which to move data: **(Choose one entity)** Select entity to which to move data: **(Choose another entity)** Select entity from which to move data: **(Choose another entity, or press Enter to exit the command)**

Command Options

Option	Description
Name of application	Specifies the name of the application that generated the extended entity data.
? to list application names	Lists the names of registered applications.
Select entity From which to move data	Selects the source entity from which to move edata.The entity must contain edata, which can be created with the EditEData (6.13.3) command.
Select entity to which to move data	Specifies the target entity to receive the edata.

Related Commands

CopyEData (6.9.53) - copies extended entity data to other entities. DelEData (6.10.25) - erases extended entity data from entities. EditEData (6.13.3) - creates and edits extended entity data. ReAssocApp (6.26.2) - associates applications with entities.

Figure 6.1418



6.21.33 -MText

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places formatted, paragraph text in a bounding box that limits the extent of the text; uses the command bar to enter and format the text. **Note:** The text does not appear in the drawing until you end the command (by pressing**Enter** at the 'Mtext' prompt.

Accessing the Command

command bar: **-mtext** : **-MText Prompts you in the command bar:** Multiline Text: First corner for block of text: **(Pick the first point for the bounding box, or else enter x,y coordinates)** Select Opposite corner for block of text or [Justification/Rotation angle/text Style/text Height/Direction/Width]: **(Pick another corner for the bounding box, or enter an option)** Mtext: **(Enter text)** Mtext: **(Enter more text, or press Enter to exit the command)**

Command Options

Option	Description
First corner for block of text	Specifies one corner of the rectangle that bounds the text block.
Opposite corner for block of text	Specifies the other corner.
Justification	Specifies the justification (1.21.20) (horizontal alignment) of the text inside the bounding box; prompts you: **Specify text block justification [Top-Left/Top-Center/Top-Right/MiddLe-left/Middle-Center/Middle-Right/Bottom-Left/Bottom-Center/Bottom-Right] <Top Left>** - **(enter a justification mode.)**
Rotation	Specifies the rotation angle of the text block; prompts you: **Rotation angle for block of text <0>** - **(enter an angle.)** Positive angles rotate the text block counter-clockwise.
Style	Specifies the text style to use for the mtext; prompts you: **Text style to use or [? to list] <Standard>** - **(enter the name of a text style.)** Enter **?** for a list of styles defined in the current drawing. See Style (6.27.53) command.
Height	Specifies the height of the text; prompts you: **Height of text <0.2000>** - **(enter a number.)** This option is unavailable when the style specifies the text height.
Direction	Specifies the direction in which the bounding box is expanded; prompts you: **Specify drawing direction [Left-to-right/Top-to-bottom/by-Style] <Left-to-Right>** - **(enter an option)**: • **Left to right** - text is placed at the left, and expands the bounding box to right. • **Top to bottom** - text is placed at the top, and expands the bottom of the bounding box. • **By Style** - returns to the previous prompt.
Width	Specifies the width of the bounding box; prompts you: **Width of text box** - **(enter a value.)** When set to 0, the box has "infinite" width.

Mtext	Specifies the text; prompts you: **Mtext** - specifies the text to be placed in the drawing. This prompt repeats until you press **Enter**to end the command.

Grips Editing

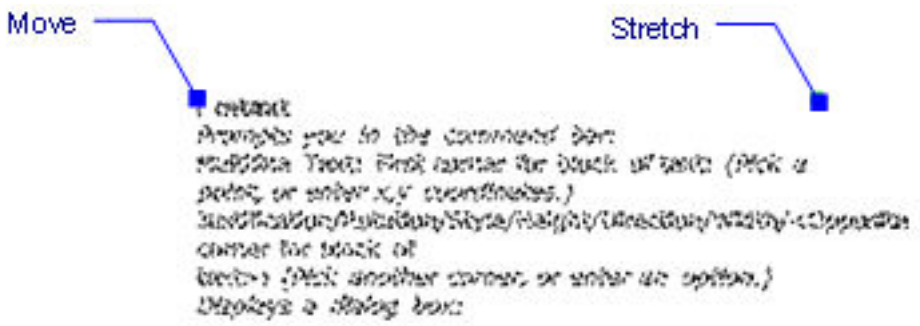
Mtext bounding boxes can be edited directly through grips:

1. Select the mtext. Notice that it has two grips.
2. Drag the grips to perform the following actions:

- Drag one grip to move the mtext.
- Drag the other grip to stretch the width of the bounding box (make it wider or narrower; the text reflows automatically to accommodate the change in width).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image472.jpg})

Figure 6.1419



Procedures

- To create single line text (1.21.20)
- To create multiline text (1.21.20)
- Including special text characters and symbols (1.21.20)
- To add a background mask (1.21.20)
- General procedure to create fields (1.21.23)
- Defining text system variables (1.21.17)

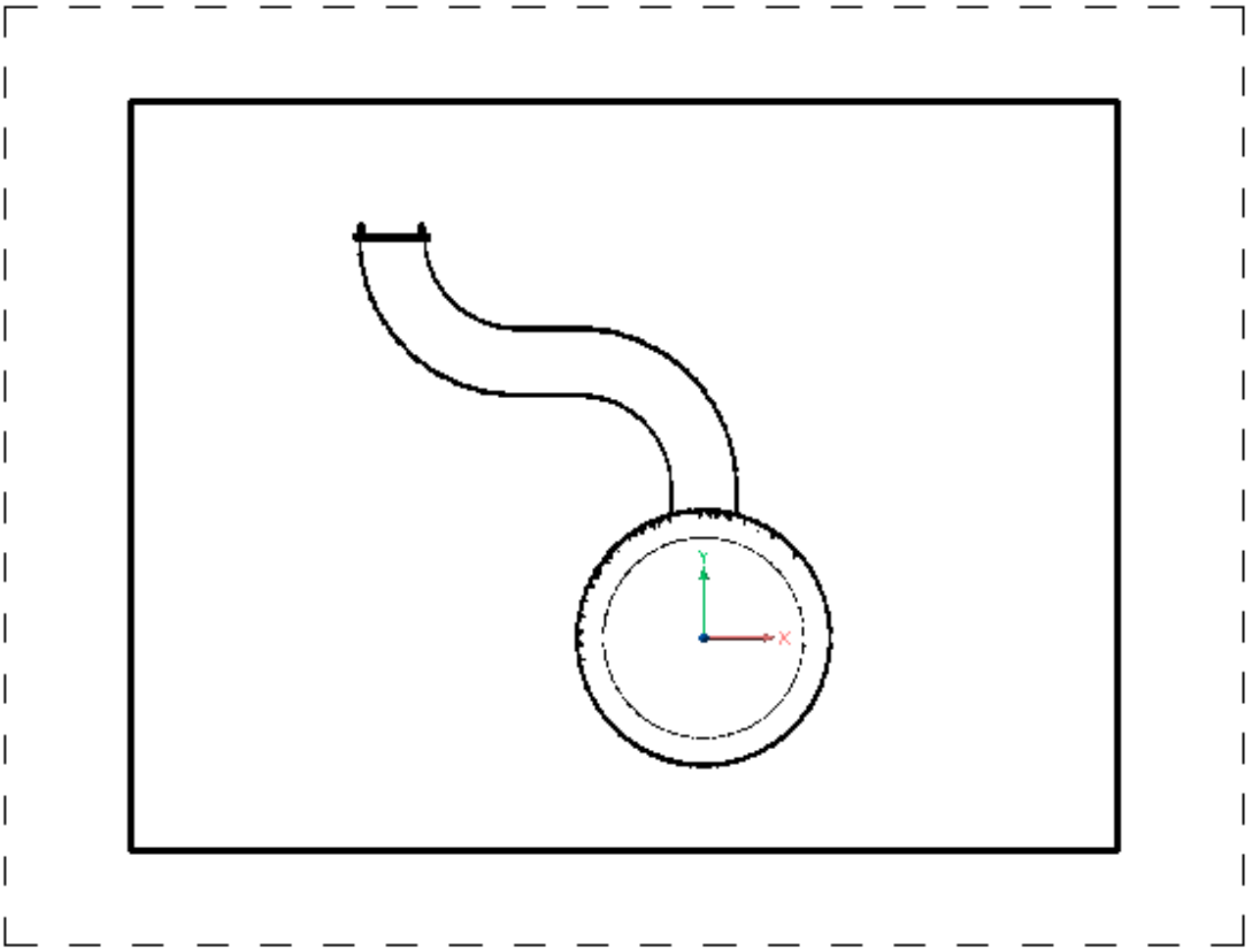
Related Commands

Text (6.29.12) - places single-lines of text in drawings. -Text (6.29.12) - places text and evaluates LISP expressions. MText (6.21.35) - places paragraph text. AttDef (6.4.27) - defines attribute text. DdEdit (6.10.14) - edits text. Style (6.27.53) - creates and modifies text styles through the Drawing Explorer. -Style (6.27.54) - creates and modifies text styles through the command bar. Change (6.9.10) - changes the position of text. QText (6.25.6) - toggles between text and rectangular representations. TxtExp (6.29.27) - explodes text into polylines. Spell (6.27.63) - checks the spelling of text in drawings. Find (6.14.8) - finds and replaces text. Properties (6.24.49) - modifies the properties of text.

Figure 6.1420



Figure 6.1421



6.21.34 MSpace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Switch from paper space to model space viewport in a layout tab. **Note:**This command operates only when a layout tab is in paper space mode. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en eg.png})

Accessing the Command

command bar: **mspace** shortcut: **double-click inside the viewport** alias: **ms** status bar: **P:Layout** : **MSpace** Displays no prompt in the command bar. To reverse the action of this command, use the PSpace (6.24.54) command, or else double-click outside the viewport border.

Command Options

No options.

Status Bar Menu

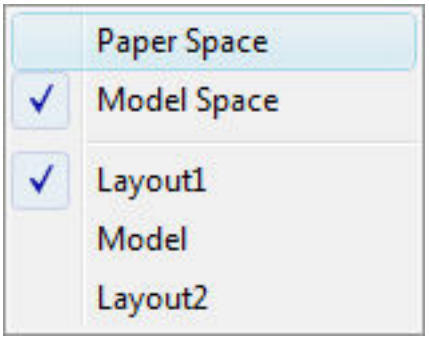
Right-click M:Layout on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image521.jpg})

Shortcut Menu	Description
Paper Space	Switches to paper space in the current layout.
Model Space	Switches to model space in the current layout.
Layout1	Switches the drawing editor to Layout1.
Model	Switches the drawing editor to Layout1.
Layout2	Switches the drawing editor to Layout2. When the drawing contains other layouts, they are also listed in the shortcut menu.

Figure 6.1422



Figure 6.1423



Related Commands

ChSpace (6.9.19) - Moves entities from paper space to model space and vice versa. MView (6.21.37) - creates viewports in paper space. VpClip (6.31.17) - clips viewports. PSpace (6.24.54) - switches to paper space. VPorts (6.31.20) - creates viewports in model space. RedrawAll (6.26.10) - redraws all viewports. RegenAll (6.26.16) - regenerates the drawing in all viewports. Zoom (6.35.7) - utilizes the XP option to specify the zoom factor in paper space relative to model space.

Figure 6.1424



Figure 6.1425



6.21.35 MText

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places formatted, paragraph text in a bounding box that limits the extent of the text. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image539.jpg})

Accessing the Command

command bar: **mtext** aliases: **t, mt** menu bar: **Draw | Multiline Text** ribbon:**Draw | Text | Multiline Text** toolbar: **Draw |** : **MText Prompts you in the command bar:** Multiline Text: First corner for block of text: **(Pick the first point for the bounding box, or else enter x,y coordinates)** Select Opposite corner for block of text or [Justification/Rotation angle/text Style/text Height/Direction/Width]: **(Pick another corner for the bounding box, or enter an option)** Displays theText Formatting toolbar and bounding box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
Enter text in the bounding box, format it, and then click **OK**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V

Figure 6.1426

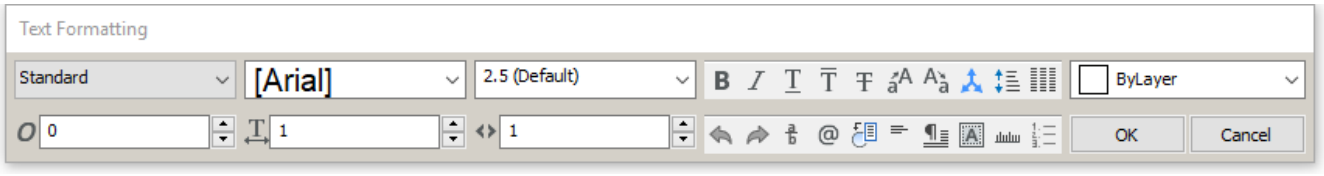


Figure 6.1427



Command Options

Option	Description
Style	Specifies the style: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/F Choose a style name from the droplist. See Style (6.27.53) command.
Font	Specifies the font. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/F Choose a font name from the droplist; this overrides the font specified by the style. BricsCAD can use any font installed on your computer. To apply a font to just one or more words: 1. Highlight the word 2. Choose the font from the droplist To apply a font to just one or two words, **highlight them**, and then choose the font from the droplist.
Height	Specifies the text height. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/F • Choose a preset text height from the droplist • Enter a value To change the height of a few words, highlight them, and then specify the height.
Bold	Toggles boldfacing of text. SHX fonts cannot be boldfaced.To apply boldfacing : 1. Select some or all of the text 2. Click the **B** button. To remove boldfacing, repeat the procedure.
Italic	Toggles italicization of text.To apply italics: 1. Select some or all of the text 2. Click the **I** button Repeat the procedure to remove italics.SHX fonts cannot be italicized.
Underline	Toggles the underlining of text; to apply underlining : 1. Select some or all of the text 2. Click the **U** button Repeat the procedure to remove underlines.

Overline	Toggles overlining of text; to apply overlining: 1. Select some or all of the text 2. Click the **O** button. Repeat the procedure to remove overlines.
Strikethrough	Toggles the strikethrough of the text; to apply strikethrough: 1. Select some or all of the text 2. Click the **T** button. Repeat the procedure to remove strikethroughs.
Upper case	Sets selected text in upper case letters, like ABC.
Lower case	Sets selected text in lower case letters, like abc.
Annotative	Toggles the annotative property of the MText. Click the annotative button to apply or remove the annotative scale.
Line spacing	Sets the line spacing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L
Color	Specifies the color of the text. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M 1. Select some or all of the text. 2. Click the **Color** button. 3. Choose a color from the droplist.
Oblique Angle	Specifies the angle by which to slant the text: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Enter an angle, or else click the spinners to change the value. • Less than 0 degrees - characters slant backwards • 0 - characters are upright • More than 0 degrees - characters slant forwards
Width Factor	Specifies the width of character: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M • Smaller than 1 - characters are narrow • 1 - characters have normal width • Greater than 1 - characters are wide
Character Spacing	Specifies the spacing between characters: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M • Smaller than 1 - characters spacing is tight • 1 - characters have normal spacing • Greater than 1 - characters spacing is loose

Undo	Undoes the last format applied.
Redo	Redoes the last undo.
Stack Fractions	Stacks text according to the special characters -- / # or ^ -- embedded in the text: • **/** - stacks vertically (separated by a horizontal line) • **#** - stacks diagonally (separated by a diagonal line) • **^** - stacks vertically not separated by a line (tolerance stack) To stack fractions: 1. Enter fraction text, such as 3/4 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou 2. Select the fraction text; e.g. double click: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou 3. Click the **Stack Fraction** button ( ): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou 4. Repeat steps 2 and 3 to unstack the fractions. To edit the stack properties: 1. Double click the stacked text.The Stack Properties dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou 2. **Text** - type values in both fields of the text to be stacked 3. **Alignment** - select an alignment option: Bottom, Center, or Top 4. **Style** - select an stack style: Diagonal Fraction, Horizontal Fraction, or Tolerance 5. **Text size** - specify how much larger or smaller the stacked text should be in the range of 25% to 125%, or else enter a value. 6. Click the **OK** button to apply the changes.
Insert Symbol	Inserts special symbols.Click the button, and then choose a symbol. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M You can enter the symbols directly by typing their codes, such as **%%d** for the degree symbol.
Field	Inserts a field. See the Field (6.14.3)command.

Text justification	Specifies the text justification (alignment) within the bounding box.Click the button, and then choose an alignment from the droplist. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M The icon on the button indicates the current justification mode.
Paragraph	Specifies the text alignment of a paragraph.Click the button, and then choose an alignment from the droplist. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
Background Mask	Sets the background mask; displays a dialog box: (\vref {fig:C:/help/enUS/V18/CmdRef/source/BackgroundMaskDialog.pn
Ruler	Toggles the display of the ruler. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M 1. First line indent marker 2. Second line indent marker 3. Right indent marker 4. Tab type
Lists	Creates numbered and bulleted lists. Click the button, and then choose an option from the droplist. (\vref {fig:C:/help/enUS/V18/CmdRef/source/MTextLists.png}) • **Enable lists and bullets** - turns on list making • **Allow automatic lists indexing** - pressing **Enter** adds a new list item

Columns

Divides the text in columns.Right click and choose **Columns**in the context menu. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
The Column Properties dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M

- **Column type** - defines how the number of columns is controlled.
- **Column number** - number of columns
- **Height** - maximum height of the columns.
- **Width** - width of each column.
- **Gutter width** - space between the columns.

Click the **Column type** list button and select a column type in the list: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M

- **Dynamic (auto height)** option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou Type a value in the **Height**, **Width**, and **Gutter width** fields. The number of columns is calculated automatically.Optionally, adjust the column height by dragging the control button.
- **Dynamic (manual height)** option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou Type a value in the **Width**, and **Gutter width** fields. The number of columns is calculated automatically.The **Height** is controlled by dragging the control button for each column separately.
- **Static** option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou Type a value in the **Column number**, **Height**, **Width**, and **Gutter width** fields.Optionally, adjust the column height by dragging the control button.

Grips Editing

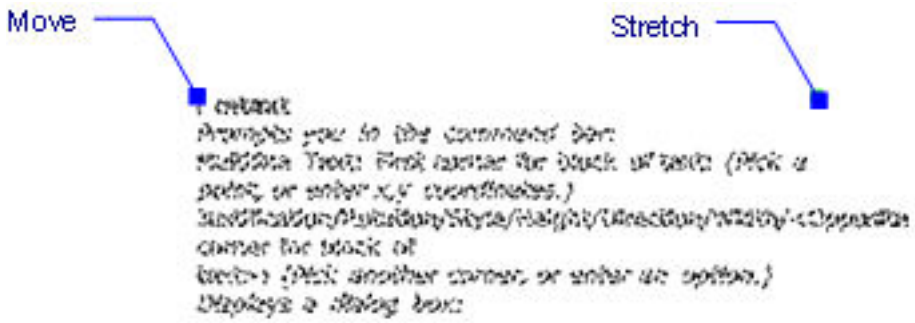
Mtext bounding boxes can be edited directly through grips:

1. Select the mtext. Notice that it has two grips.
2. Drag the grips to perform the following actions:

- Drag the left grip to move the mtext.
- Drag the right grip to stretch the width of the bounding box (make it wider or narrower; the text reflows automatically to accommodate the change in width).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image472.jpg})

Figure 6.1428



Procedures

- To create single line text (1.21.20)
- To create multiline text (1.21.20)
- Including special text characters and symbols (1.21.20)
- To add a background mask (1.21.20)
- To add a text frame (1.21.20)
- General procedure to create fields (1.21.23)
- Defining text system variables (1.21.17)

Related Commands

-MText (6.21.33) - places paragraph text. Text (6.29.12) - places single-lines of text in drawings. -Text (6.29.12) - places text and evaluates LISP expressions. AttDef (6.4.27) - defines attribute text. DdEdit (6.10.14) - edits text. Field (6.14.3) - inserts a field in a multiline text, table cell or attribute. Updatefield (6.30.10) - Updates fields in selected Mtexts. Style (6.27.53) - creates and modifies text styles through the Drawing Explorer. -Style (6.27.54) - creates and modifies text styles through the command bar. QText (6.25.6) - toggles between text and rectangular representations. Change (6.9.10) - changes the position of text. TxtExp (6.29.27) - explodes text into polylines. Spell (6.27.63) - checks the spelling of text in drawings. Find (6.14.8) - finds and replaces text. Properties (6.24.49) - modifies the properties of text.

Figure 6.1429

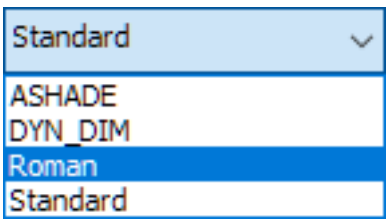


Figure 6.1430

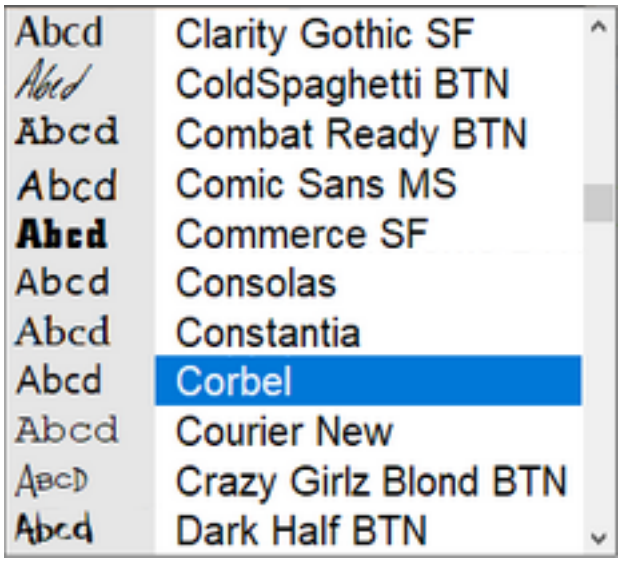


Figure 6.1431

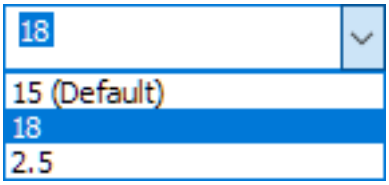


Figure 6.1432

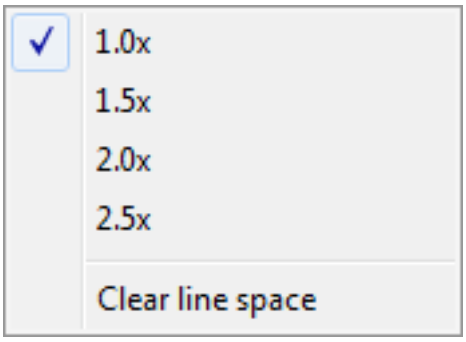


Figure 6.1433

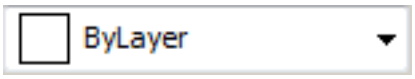


Figure 6.1434



Figure 6.1435



Figure 6.1436



Figure 6.1437

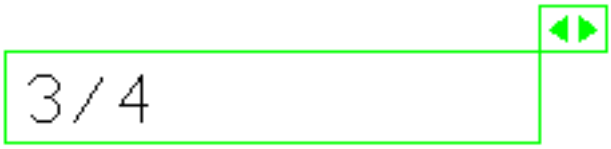


Figure 6.1438

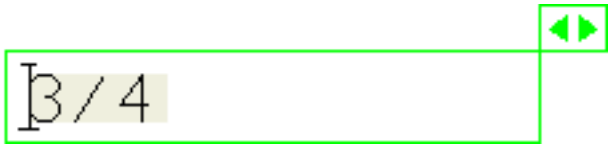


Figure 6.1439

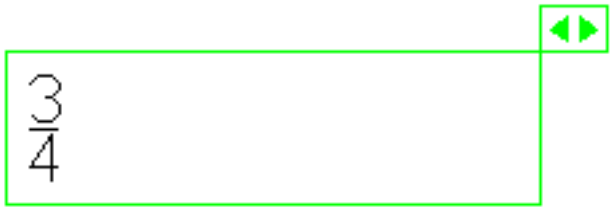


Figure 6.1440

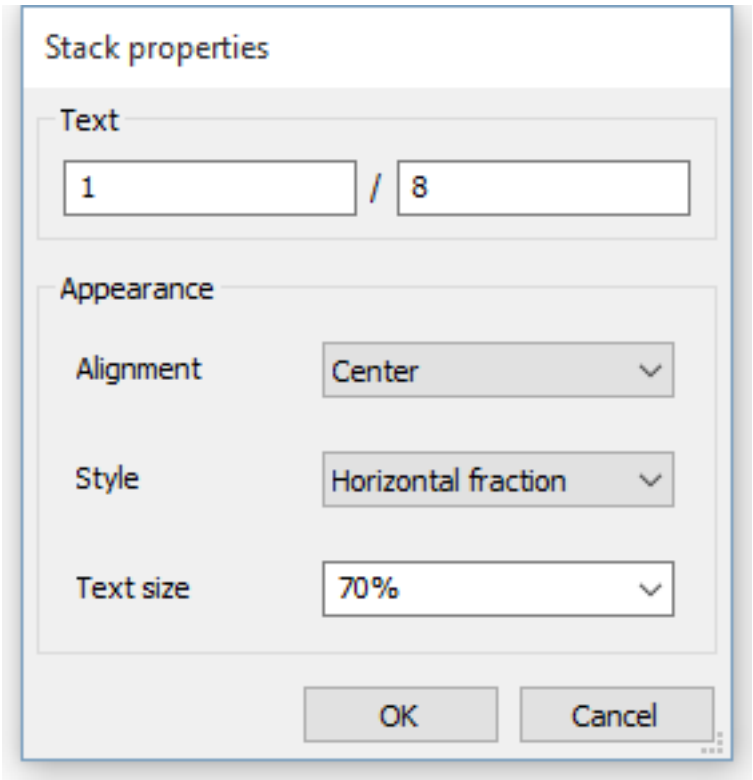


Figure 6.1441

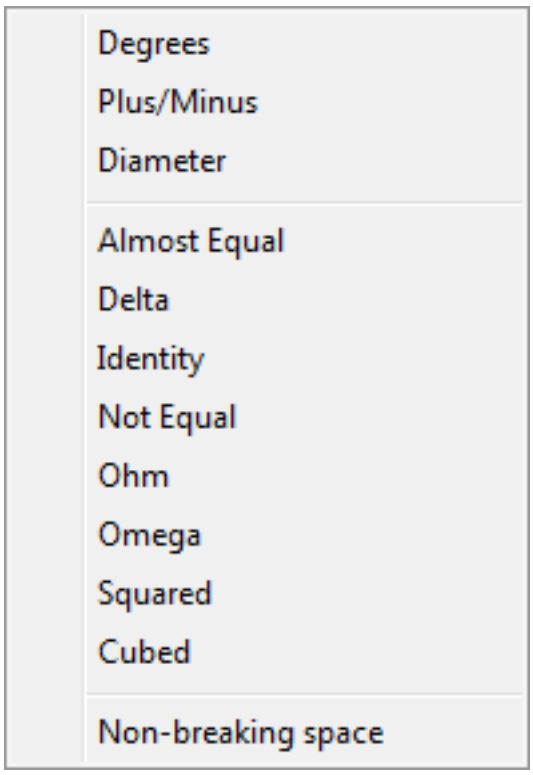


Figure 6.1442

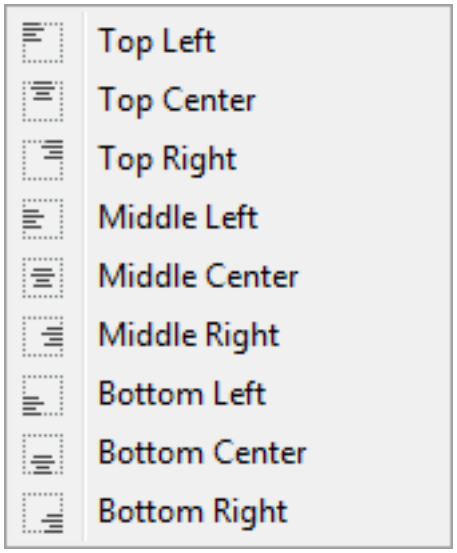


Figure 6.1443

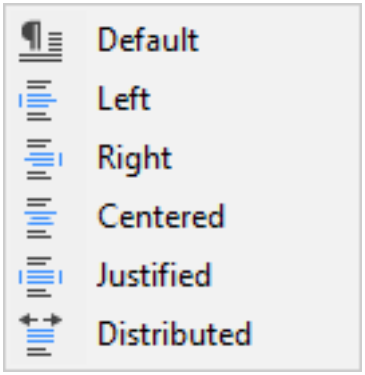


Figure 6.1444

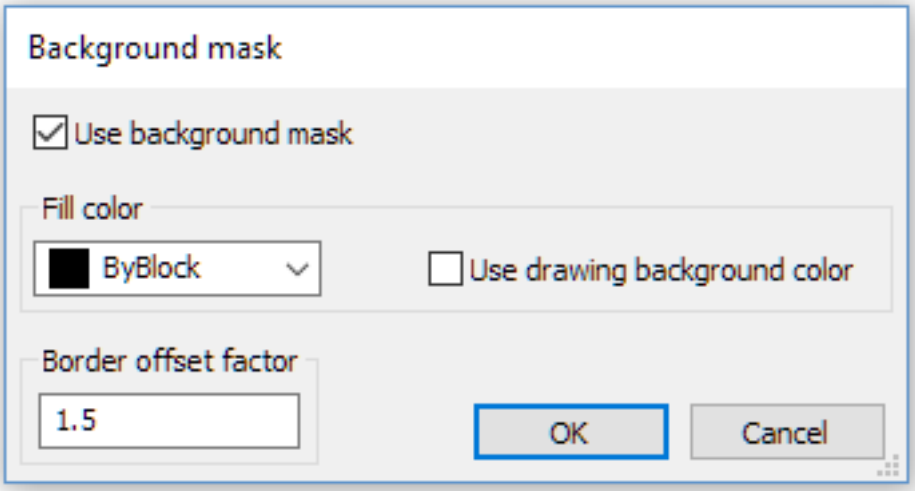


Figure 6.1445



Figure 6.1446

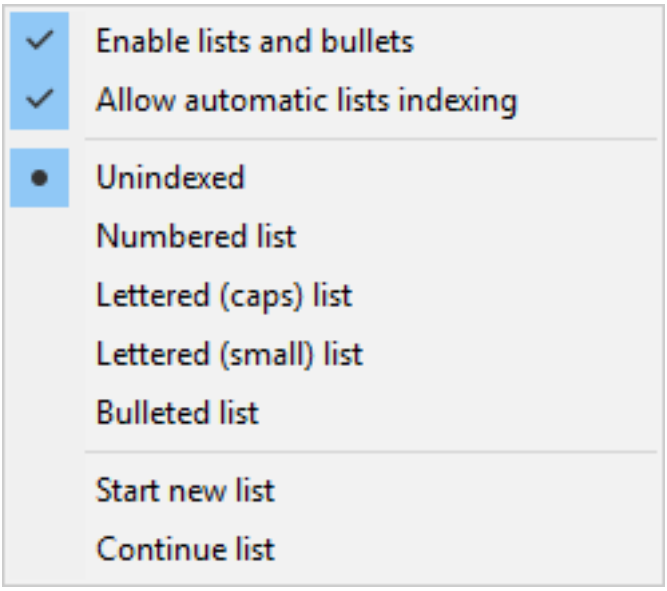


Figure 6.1447

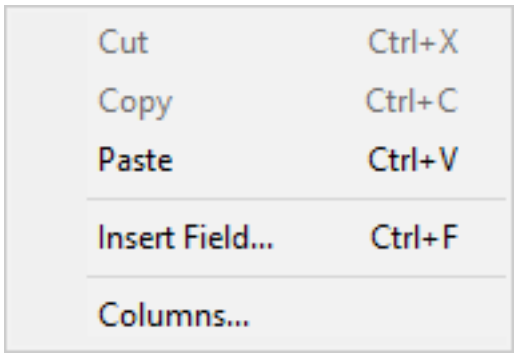


Figure 6.1448

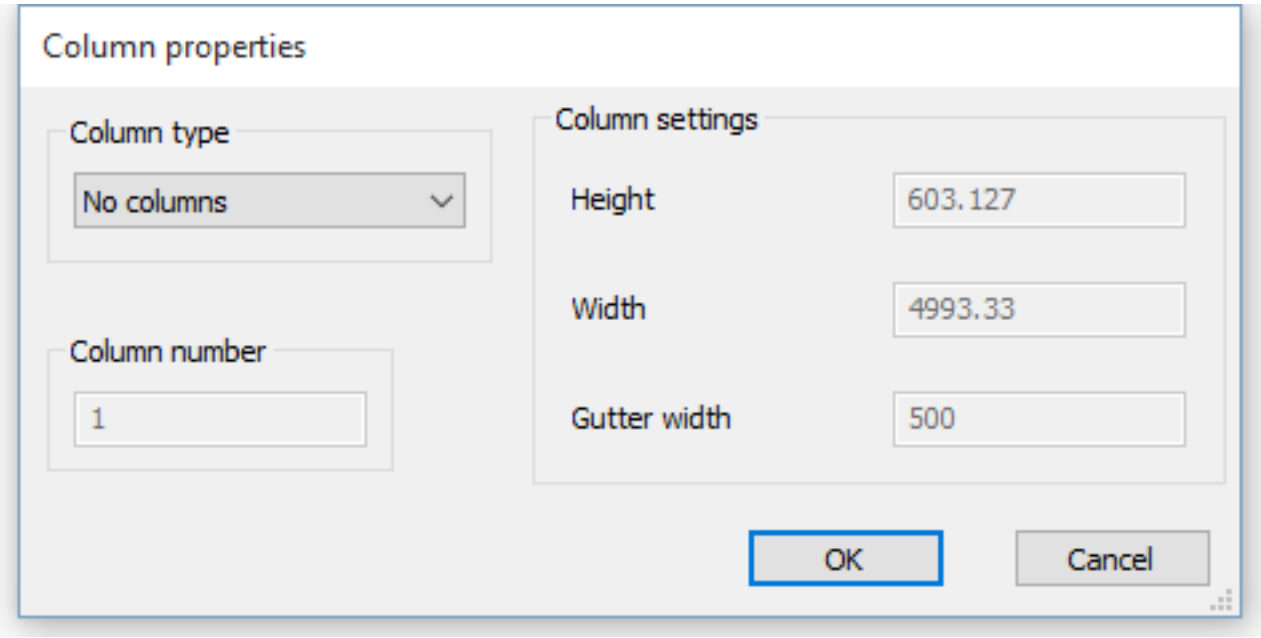


Figure 6.1449

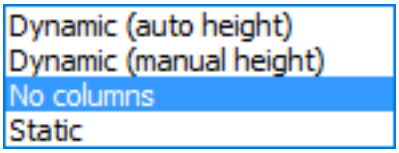


Figure 6.1450

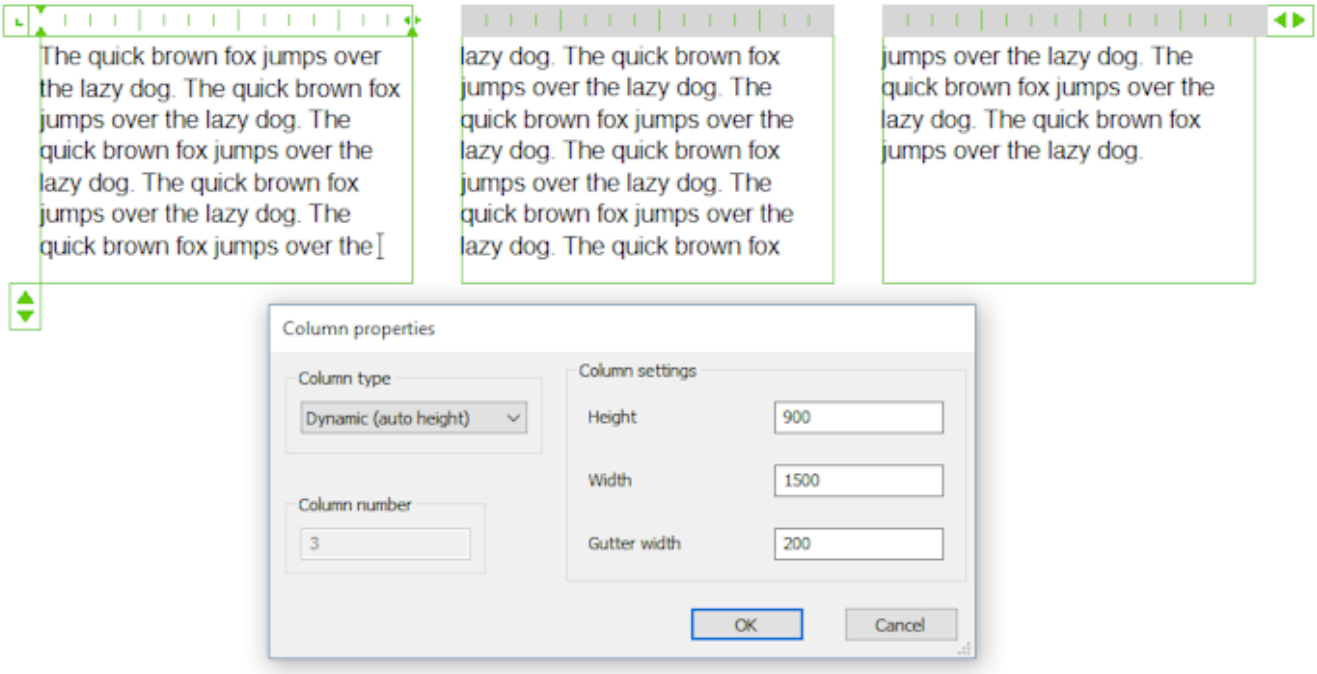


Figure 6.1451

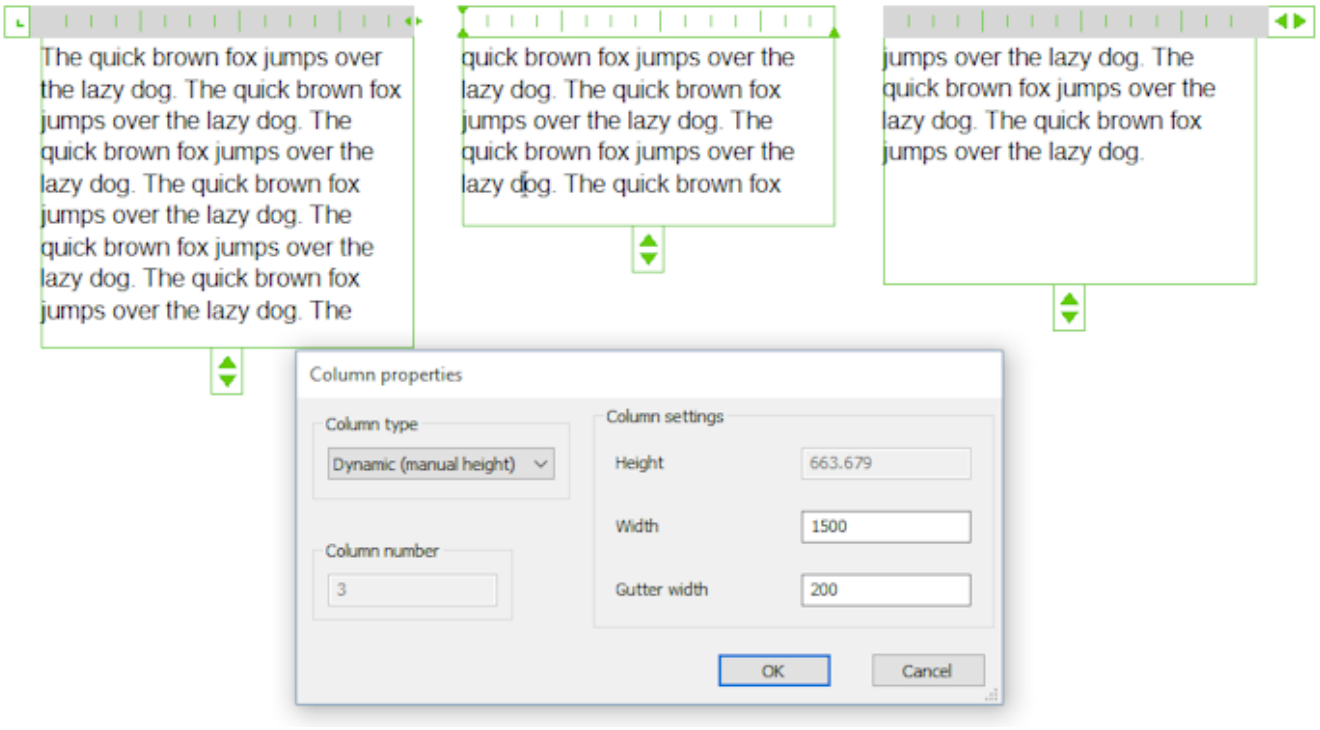
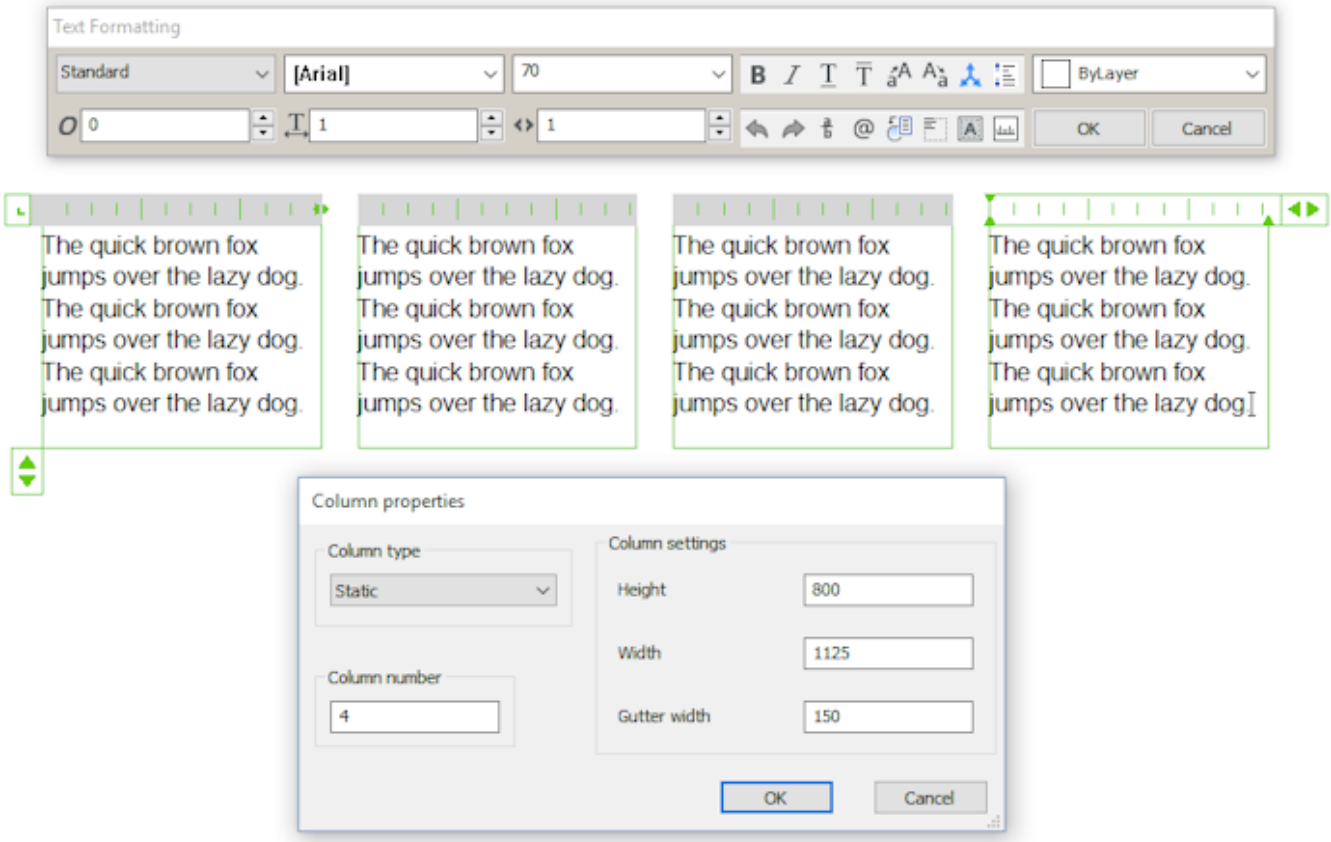


Figure 6.1452



6.21.36 Mtp

   Allows to specify a point as the midpoint between two points whenever a command prompts to enter a point (transparent command).

Accessing the Command

transparent: **mtp** alias: **m2p** toolbar : **Entity Snaps**  **: Mtp Prompts you in the command bar:** First point of mid: (Identify the first reference point) Second point of mid: (Identify the second reference point)

Command Options

None

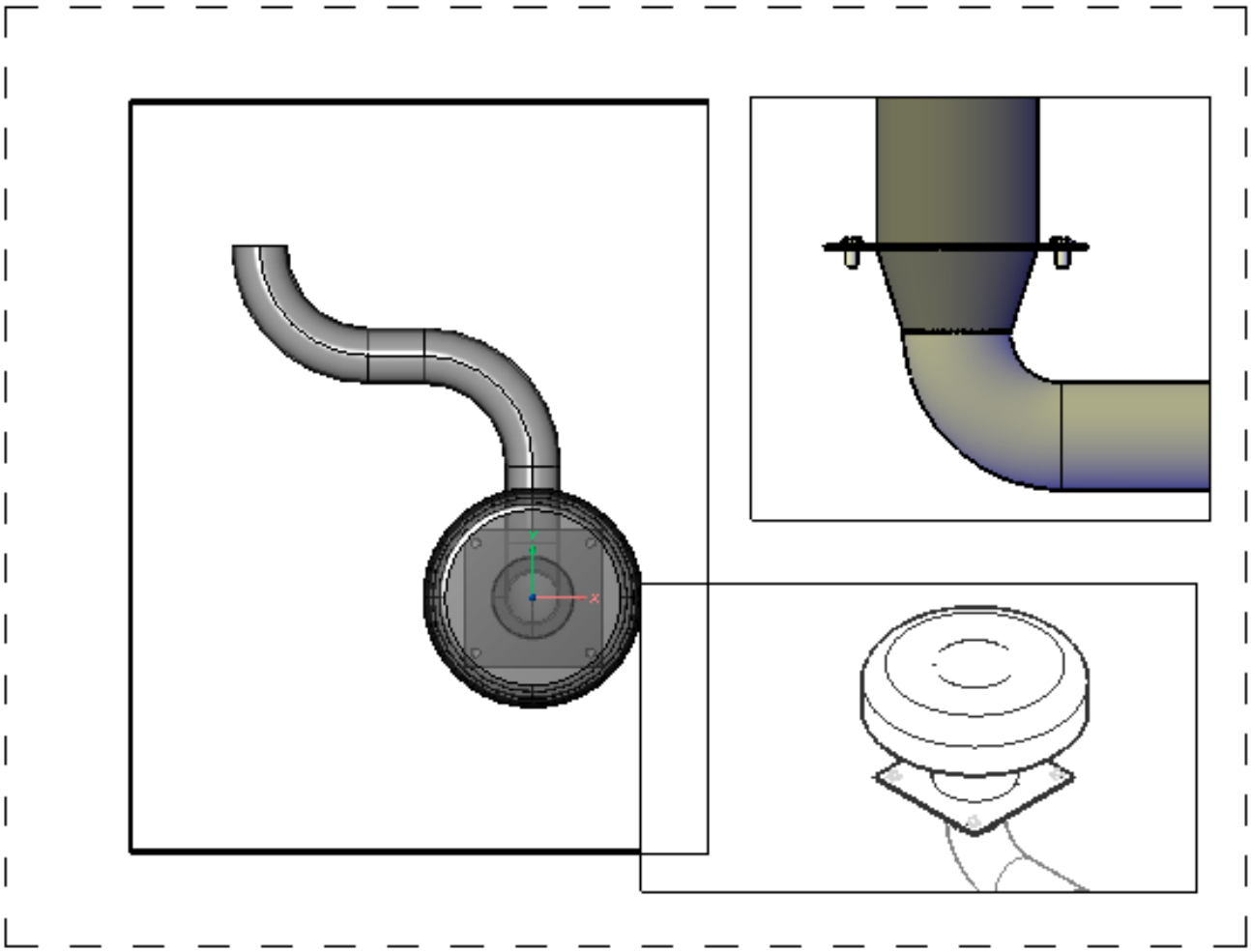
Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.1453



Figure 6.1454



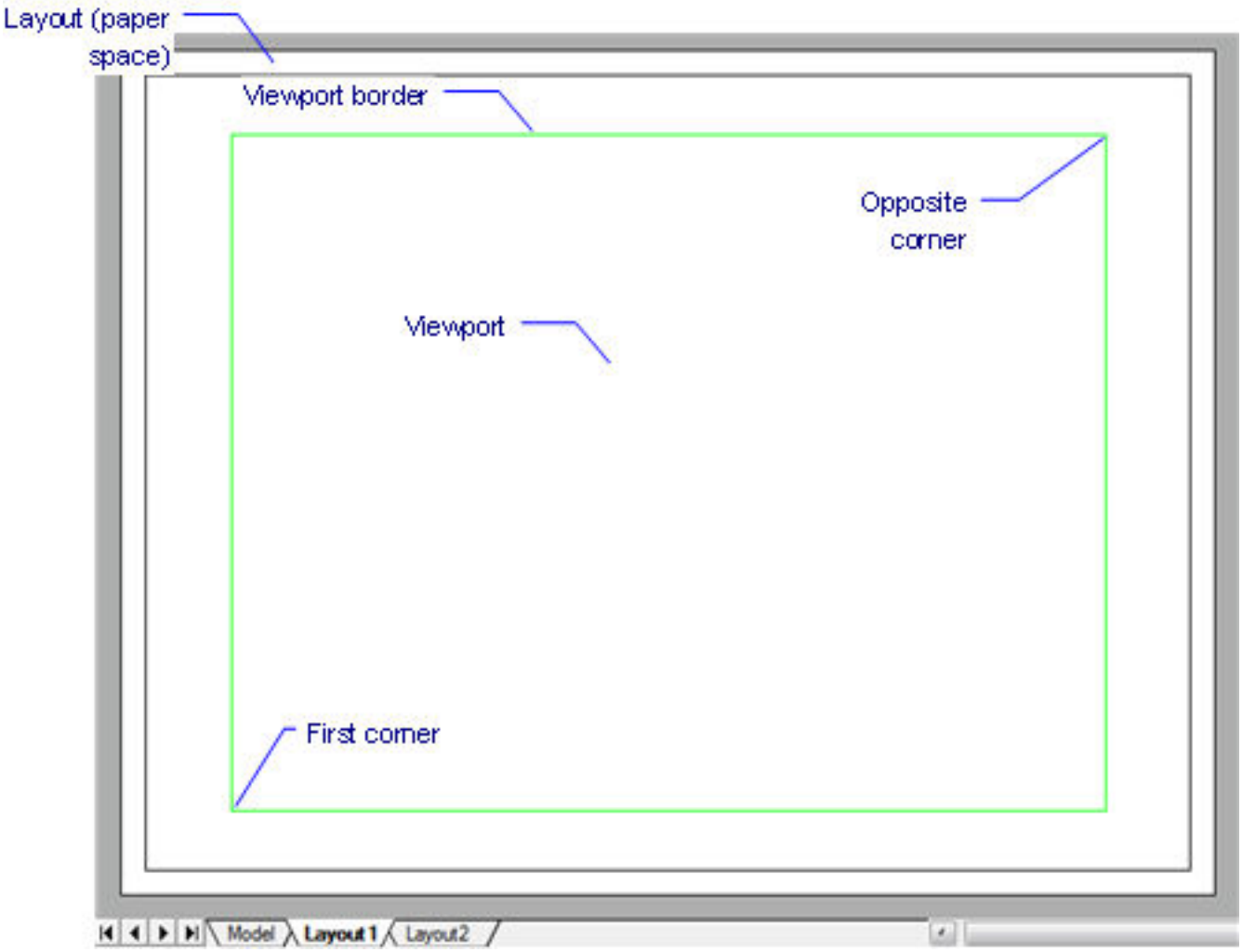
6.21.37 MView

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates one or more viewports in layout tabs (short for "make viewports") to display entities drawn in Model tab. Each viewport can have its own settings, shown below. **Note:**This command is for making viewports in layout tabs. To create viewports in Model tab, use the VPorts (6.31.20) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/M/mview-eg.png})

Accessing the Command

 command bar: **mview** alias: **mv** menu bar: **View | Paper Space Views** ribbon:**View | View | Paper Space Views** toolbar: **View |**  : **MView Prompts you in the command bar:** Viewports: Set first corner or [ON/OFf/Lock/Fit/create 2 viewports/create 3 viewports/create 4 viewports/Object/Polygonal]: **(Pick a point, or enter an option) Opposite corner:** (Pick another point.) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image2

Figure 6.1455



Command Options

Option	Description
First corner	Specifies one corner for a new, rectangular viewport.Pick a point, or enter x,y coordinates.
Opposite corner	Specifies the other corner for a new viewport.
ON	Turns on viewports that are turned off; prompts you: **Select viewports to turn on** - **(choose one or more viewports.)**
OFF	Turns off viewports; prompts you: **Select viewports to turn off** - **(choose one or more viewports.)** This option hides the content of the viewport; the viewport border remains visible.To hide the viewport border, place it on a separate layer, and then freeze the layer.
Lock	Locks the viewport's scale factor, relative to model space; prompts you: **Viewport locking [ON/OFf] <On>** - **(type ON or OFF.)** • **ON** - locks the viewport's scale factor • **OFf**- unlocks the viewport's scale factor Set the scale factor with the Zoom (6.35.7) command's XP option.
Fit	Draws a rectangular viewport that fits the current layout. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
2	Draws two rectangular viewports; prompts you: 1. **Two viewports [Horizontal/Vertical] <Vertical>** - **(enter an option.)** 2. **First corner of bounding rectangle or [Fit (6.21.37)]** - **(pick a point or choose the Fit option)** 3. **Opposite corner** - **(Pick a point.)**
Horizontal	Draws two horizontal viewports of equal size: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Vertical	Draws two vertical viewports of equal size: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
3	Draws three rectangular viewports; prompts you: 1. **Three viewports [Horizontal/Vertical/Above/Below/Left/Right] <Right>** - **(enter an option.)** 2. **First corner of bounding rectangle or [Fit (6.21.37)]** - **(Pick a point or choose the Fit option.)** 3. **Opposite corner** - **(Pick a point.)**
Horizontal	Draws three horizontal viewports of equal size: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Vertical	Draws three vertical viewports of equal size: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Above	Draws one viewport above two side-by-side viewports. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Below	Draws two side-by-side viewports above one viewport. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Left	Draws one viewport to the left of two stacked viewports. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Right	Draws one viewport to the right of two stacked viewports. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
4	Draws four rectangular viewports of equal size; prompts you: 1. **First corner of bounding rectangle or [Fit (6.21.37)]** - **(Pick a point or choose the Fit option.)** 2. **Opposite corner** - **(Pick a point.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Object	Converts an object into a viewport border; prompts you: **Select an object to clip viewport** - **(specifies a closed object.)** The viewport is traced over the object; the original object remains in the drawing.This option is useful for ensuring a viewport border precisely matches an outline. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Polygonal	Draws non-rectangular viewports made of lines and arcs; prompts you: 1. **Specify start point** - **(specifiy the starting point for the polygon.)** 2. **Set next point or [draw Arcs/Distance/Follow]** - **(specify a point, or enter an option.)** 3. **Set next point or [draw Arcs/Distance/Follow/Undo]** - **(specify a point, or enter an option.)** 4. **Set next point or [draw Arcs/Close/Distance/Follow/Undo]** - **(specify a point, enter an option, or press Enter to close the polyline.)** • **draw Arcs**- enters arc drawing mode; prompts you: **Set end of arc or [draw Lines/ Angle/ CEnter /Direction/ Radius/ Second point]** - **(enter an option; see theArc (6.4.12) command Use the Draw Lines option to exit arc drawing mode.)** • **Close** - closes the polygon and exits the command. • **Distance** - draws the next line segment at a specified distance and angle; prompts you: **Distance for segment** - **(type a value.) Angle of segment** - **(type a value.)** • **Follow** - draws the next line segment at the same angle; prompts you: **Set next point or [draw Arcs/Distance/Follow/Undo]** - **(pick a point or enter an option.)** • **Undo**- undoes (undraws) the last segment.

Related Commands

MvSetup (6.21.39) - creates multiple paper space viewports; in model space the command creates a rectangle, representing a paper sheet with respect to a specified scale. MSpace (6.21.34) - switches to model space. PSpace (6.24.54) - switches to paper space. VpClip (6.31.17) - clips viewports. VPorts (6.31.20) - creates viewports in model space. RedrawAll (6.26.10) - redraws all viewports. RegenAll (6.26.16) - regenerates the drawing in all viewports. Zoom (6.35.7) - utilizes the XP option to specify the zoom factor in paper space relative to model space.

Figure 6.1456



Figure 6.1457



Figure 6.1458

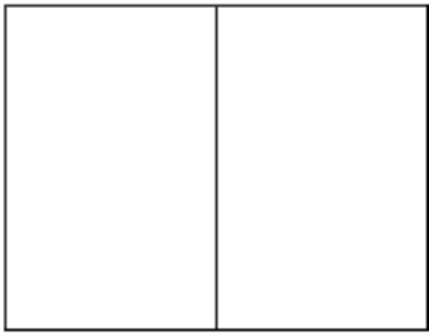


Figure 6.1459

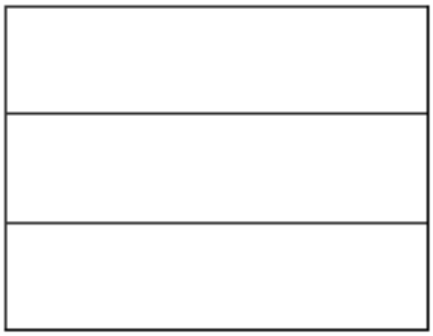


Figure 6.1460

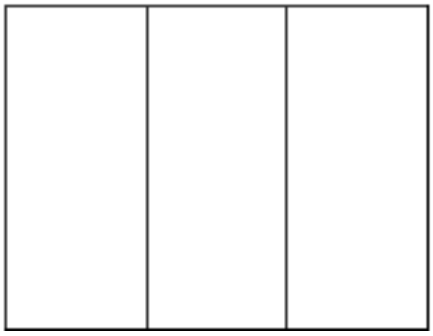


Figure 6.1461



Figure 6.1462



Figure 6.1463

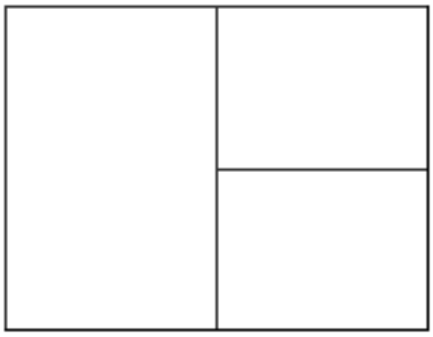


Figure 6.1464

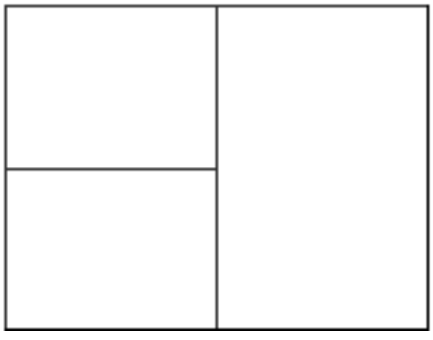


Figure 6.1465



Figure 6.1466

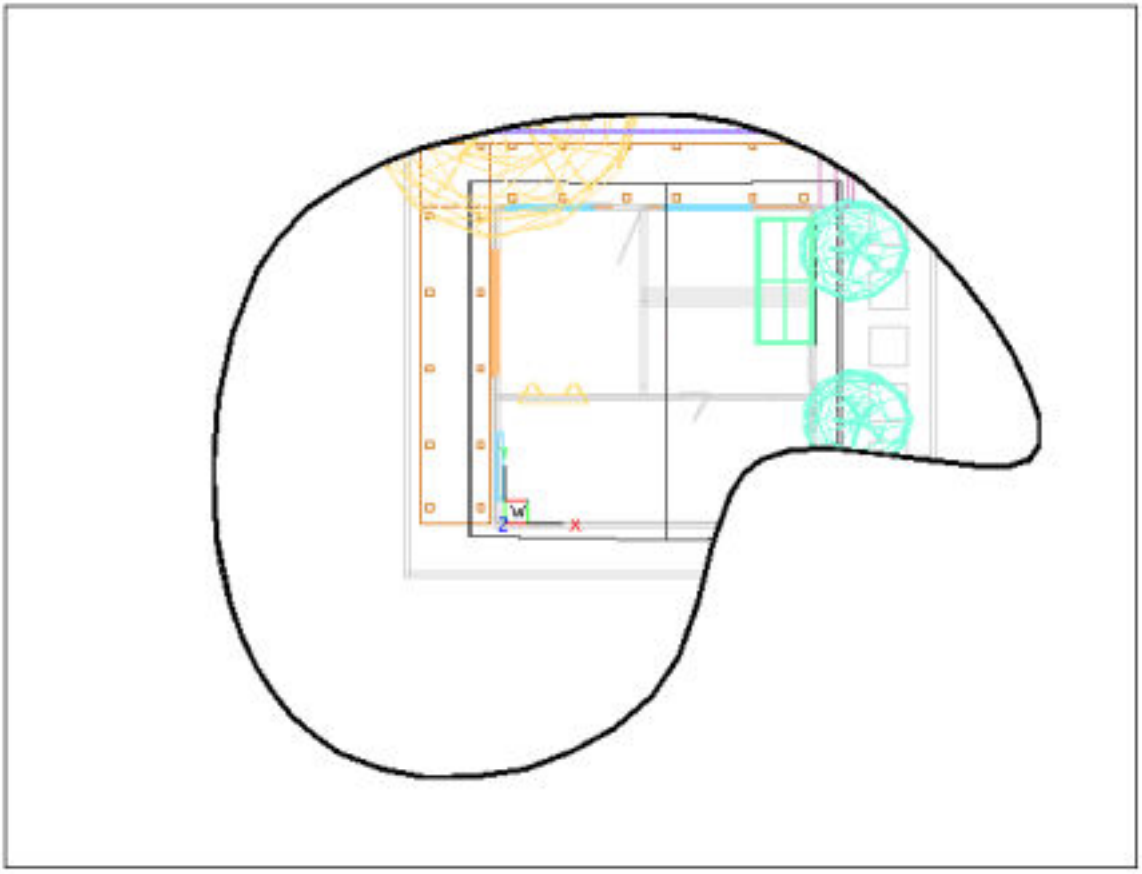


Figure 6.1467



6.21.38 Multiple

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Repeats commands until you press **Esc**. **Note:** This command is useful for commands that do not repeat automatically.

Accessing the Command

command bar: **multiple** transparent: **'multiple** : **Multiple Prompts you in the command bar:** Enter command name to repeat: **(Enter the name of a command)** The command is executed and then repeated until you press Esc.

Command Options

Option	Description
Enter command name to repeat	Enter the name of a command.The command is executed, and then repeated until you press **Esc**.

Related Commands

All.

Figure 6.1468



6.21.39 MvSetup

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates multiple paper space viewports, and then aligns, rotates and scales them (short for "multiple viewports setup"). **Note:** This command operates differently, depending on whether it starts in Model tab or a layout tab.

Accessing the Command

command bar: **mvsetup** transparent: **'mvsetup** : **MvSetup When in model space,prompts you in the command bar:** Enable paper space? [No/Yes] <Yes>: **(Enter N to stay in Model tab; pressing Y switches to to the most recently used layout and the carries on with the paper space prompts; see below)** Enter units type [Scientific/Decimal/Engineering/Arch **(Choose a unit type)** Example of Metric scales: 10 is 1:10 Enter scale: (**Enter a number that represents the scale factor)** Enter paper width: **(Type the paper width in drawing units)** Enter paper height: **(Type the paper height in drawing units)** Draws a rectangle representing the paper sheet with at the specified scale, with the lower left corner at the origin of the current coordinate system. **When in paper space,prompts you in the command bar:** Enter an option [Align/Create/Scale viewports/Undo]: (**Enter an option)**

Command Options

Paper Space Option	Description
Align	Pans the display in a viewport to align it with a base point in a reference viewport; prompts you: **Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]** - **(enter an option.)** Depending on the relative position of the two viewports, the display of the panned viewport might be shifted outside of the viewport boundary.
Angled	Pans the view in the second viewport is panned over the specified distance and angle. 1. **Specify basepoint** - **(pick a point in the reference viewport.)** 2. **Specify point in viewport to be panned** - **(pick a point, which will be used as the anchor point.)** 3. **Specify offset distance from base** - **(specify the desired distance between the basepoint and the anchor point. Either type the distance or pick two points.)** 4. **Specify angle from base** - **(specify the desired angle of the line between the basepoint and the anchor point. Either type the angle or pick two points.)**

Horizontal	Pans the view in the second viewport vertically to align the base point and the anchor point horizontally; prompts you: 1. **Specify basepoint** - **(pick a point in the reference viewport.)** 2. **Specify point in viewport to be panned** - **(pick a point, which will be used as the anchor point.)**
Vertical	Pans the view in the second viewport horizontally to align the base point and the anchor point vertically; prompts you: 1. **Specify basepoint** - **(pick a point in the reference viewport.)** 2. **Specify point in viewport to be panned** - **(pick a point, which will be used as the anchor point.)**
Rotate view	Rotates the view in the viewport by the specified angle; prompts you: 1. **Specify basepoint in the viewport to be rotated** - **(pick a point to define the basepoint of the rotation.)** 2. **Specify rotation angle** - **(enter a rotation angle or pick a second point.)** When the VPROTATEASSOC variable is on, the display in a viewport is rotated if you rotate the viewport.
Undo	Undoes the previous alignment action and returns to the alignment options prompt.
Create	Creates viewports in the current layout; prompts you: **Enter an option [Delete objects/Create viewports/Undo] <Create viewports>** - **(enter an option.)**
Delete entities	Deletes viewport entities; prompts you: **Select entities** - **(select the viewports you want to delete.)**

Create Viewports	Creates new viewports; prompts you: **Enter choice for layout options <0-3> [None/Single/Std. Engineering/Array of viewports] <0>** - **(enter an option)**: • **0 (None)**- makes no viewports • **1(Single)** - makes a single viewport; prompts you: 1. **Specify first corner of bounding area for viewport(s)** - **(pick a point.)** 2. **Other corner of rectangle**- **(pick a point.)** • **2 (Std Engineering)** - makes four viewports preset to top, front, right and south-east isometric viewpoints; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M 1. **Specify first corner of bounding area for viewport(s)** - **(pick a point.)** 2. **Other corner of rectangle** - **(pick a point.)** 3. **Specify distance between viewports in X direction <0.000>** - **(enter a horizontal spacing distance between viewports.)** 4. **Specify distance between viewports in Y direction <0.000>** - **(enter a vertical spacing distance between viewports.)** • **3 (Array of viewports)** - creates the number of viewports that you want; prompts you: 1. **Specify first corner of bounding area for viewport(s)** - **(pick a point.)** 2. **Other corner of rectangle** - **(pick another point.)** 3. **Enter number of viewports in X direction <1>** - **(type the number of viewports you want placed across the drawing horizontally.)** 4. **Enter number of viewports in Y direction <1>** - **(type the number of viewports you want vertically.)** 5. **Specify distance between viewports in X direction <0.000>** - **(enter a horizontal spacing distance between viewports.)** 6. **Specify distance between viewports in Y direction <0.000>** - **(enter a vertical spacing distance between viewports.)**
Undo	Undoes the creation of viewports.

Scale Viewports	Specifies the scaling inside viewports; prompts you: 1. **Select entities** - **(select the viewport entities you want to scale.) Entities in set: 20** 2. **Select entities** - **(select more viewports, or press Enter to stop selecting viewports.)** 3. **Set the scaling mode of viewports: Interactively/<Uniform>** - **(press Enter to scale all viewports equally, or else choose Interactively to set the scale for each viewport separately.)** 4. **Enter the number of paper space units <1.0>** - **(enter a number.)** 5. **Enter the number of model space units <1.0>** - **(enter a number.)**
Undo	Undoes all operations executed by the current MvSetup command session.

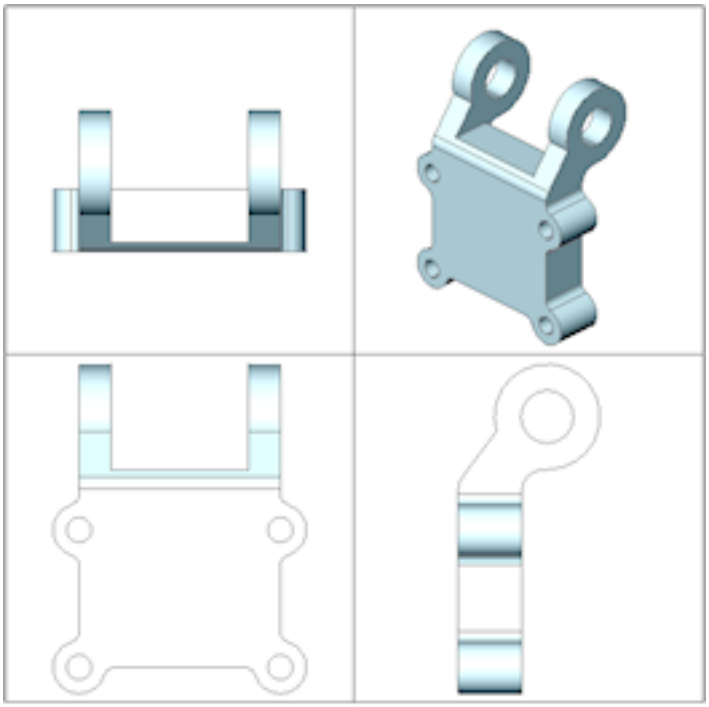
Procedures

- Creating an array of layout viewports (1.16.14)
- Setting the scale of a selection of viewports (1.16.14)
- To rotate the display in a viewport (1.16.14)
- To align viewports (1.16.14)

Related Commands

AlignSpace (6.4.7) - adjusts viewport angle, zoom factor and pan position based on alignment points specified in model space and paper space. MView (6.21.37) - creates one or more viewports in layout tabs. MSpace (6.21.34) - switches to model space. PSpace (6.24.54) - switches to paper space. VpClip (6.31.17) - clips viewports. VPorts (6.31.20) - creates viewports in model space. RedrawAll (6.26.10) - redraws all viewports. RegenAll (6.26.16) - regenerates the drawing in all viewports. Zoom (6.35.7) - utilizes the XP option to specify the zoom factor in paper space relative to model space.

Figure 6.1469



6.22 N

6.22.1 Navigate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Walks and flies through 3D models with the keyboard, as used in computer games.

Figure 6.1470



Accessing the Command

 command bar: **navigate** menu bar: **View | Navigate** toolbar: **View** |  Quad: **General** |  : **navigate**
Prompts you in the command bar: Flight mode is Off, STEPSIZE=30, STEPSPERSEC=10 >>Press Esc to quit or right click to modify navigation settings. Hover the mouse cursor over the target location and then hold down a key to navigate; see list below. Move the cursor to change the target location.

Command Options

Keyboard Option Description A Left Arrow Moves to the left W Up Arrow Moves forward S Down Arrow Moves backwards D Right Arrow Moves to the right F Toggles flight mode:

- **On** - flight mode
- **Off** - walk mode restricts movement to the xy-plane.

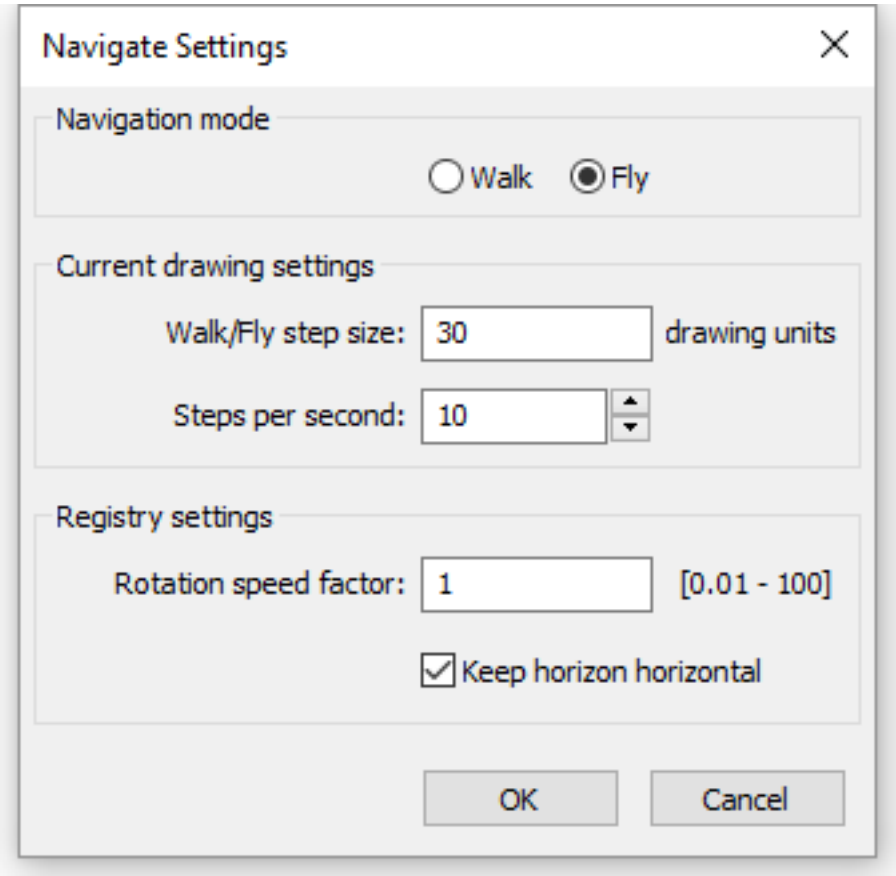
Esc Exits the command.

Mouse Option	Description
Hold down left button and drag the mouse	Look around mode
Right click	Brings up the Navigation Settings dialog box; see below.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/NavigateSettingsDialog.png})

Navigate Settings Options	Description
Navigation Mode	Toggles between Walk and Fly modes: • **Flight mode** - permits movements in all three dimensions • **Walk mode** - restricts movement to the xy-plane.
Current Drawing Settings	**Walk/Fly step size** - enter a number in drawing units; this is the distance the view travels with each press of the key **Steps per second** - specifies how quickly the view changes when a travel key is held down (This section refers to settings saved with the drawing, and so will be different in other drawings.)
Registry Settings	**Rotation speed factor** - specifies how quickly the view rotates **Keep horizon horizontal** - keeps the flight level; this option is grayed-out for Walk mode (This section refers to settings saved with the registry, and so will be the same in other drawings.)
OK	Exits the dialog box and returns to Navigate mode.

Figure 6.1471



Related Commands

RtRot (6.26.38) - rotates 3D drawings in real-time. RtWalk (6.26.43) - walk left / right and forward / backward in a 3D scene.

Figure 6.1472



Figure 6.1473



6.22.2 Nearest

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles Nearest entity snap mode; snaps to geometry on a 2D and 3D entity nearest to the cursor. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image335.png})

Accessing the Command

 command bar: **nearest** transparent: **nea** ribbon:**Home | Entity Snaps | Snap to Nearest** toolbar: **Entity snaps** |  : **nearest** Displays no prompts in the command bar. When nearest entity snap is on, the cursor snaps to the nearest geometry.This entity snap works with 2D and 3D entities.

Command Options

Option	Description
Nearest	Entering the command toggles Nearest entity snap. It snaps to the nearest geometry on entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode. Znearest (6.35.5) - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor.

Figure 6.1474



6.22.3 Netload

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Loads .NET applications.

Accessing the Command

command bar: **netload** : **netload** Displays a dialog box: select the application to be loaded, and then press the **OK** button. When the FILEDIA variable is off, **you are prompted in the command bar:** Assembly file name: **(Type the name of the application to be loaded, and then press Enter)**

Command Options

Option	Description
Assembly file name	1. Type the name of the application to be loaded. 2. Press**Enter**.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. Appload (6.4.14) - loads DRX, LISP, and SDS applications to run inside BricsCAD. Commands (6.9.37) - lists the names of internal and external commands. CuiLoad (6.9.56) and CuiUnload (6.9.57) - loads and unloads menu files. Customize (6.9.58) - customizes the user interface. MenuLoad (6.21.14) and MenuUnload (6.21.15) - loads and unloads menu files. Security (**??**) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not. VbaLoad (6.32.2) and -VbaLoad (6.32.3) - loads VBA projects. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaMan (6.32.4) - manages VBA projects. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.1475



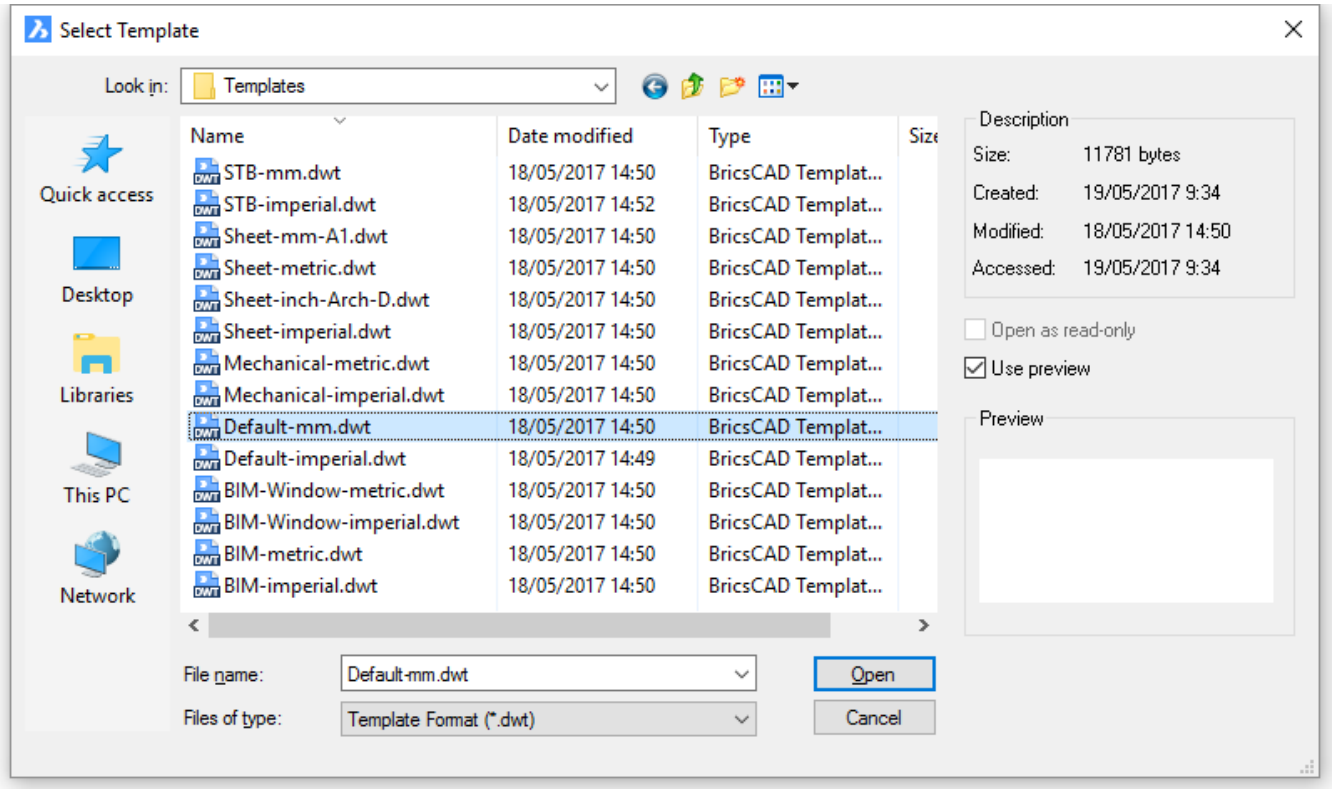
6.22.4 New

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Prompting you to select a DWT template file, and then starts a new drawing based on it. **Note:** The QNew (6.25.2) command starts new drawings without selecting a template.

Accessing the Command

 command bar: **new** shortcut: **Ctrl+N Note:** The ”New” options on the menu bar and ribbon run the QNew command. : **new** Opens Select Template dialog to the folder defined by the TemplatePath preference, by default **C:\Users\userid\AppData\Local\Bricsys\BricsCAD\V19x64\en_US\Templates**. (\vref {fig:C:/Dev/Task-2448/images/staticb Choose a DWT template file or DWG drawing file, and then click **Open. Note**: This command instead displays the New Wizard dialog box when the StartUp variable is on; seeNewWiz (6.22.6) command.

Figure 6.1476



Command Options

Option	Description
File name	Specifies the file name.To open more than one file at a time, hold down the **Ctrl** key while selecting file names.
Files of Type	Chooses the type of drawing file: • **DWT**- template files define the settings and content of new drawings; the program makes a copy of the DWT file and then opens a new DWG file based on it • **DWG**- drawing files are opened like the Open (6.23.8) command.
Open as read-only	Toggles read-only status of drawings: • **On** - opens drawings read-only; they can be edited but not saved to the same file name • **Off** - open drawings normally Use the SaveAs (6.27.3) command to save by another name.Template drawings are always opened as read-only.
Use Preview	Toggles previews of the drawing: • **On** - displays preview image in Preview area • **Off** - does not display preview image Many template files are empty, and so appear to not have a preview.
Open	Opens the file.

Procedures

- Working with document tabs (1.12.2)
- To create a drawing using a default template drawing (1.14.1)
- To set the default template drawing (1.14.1)
- To create a drawing from scratch (1.14.1)

Figure 6.1477



6.22.5 NewSheetSet

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a new sheet set by several methods:

- Starting from scratch
- Creating a new one from a sheet set template
- Borrowing settings from an existing sheet set
- Borrowing the structure from existing drawings

Note: A sheet set is a collection of drawings used in projects that can be viewed, edited, and published (printed) as a set.

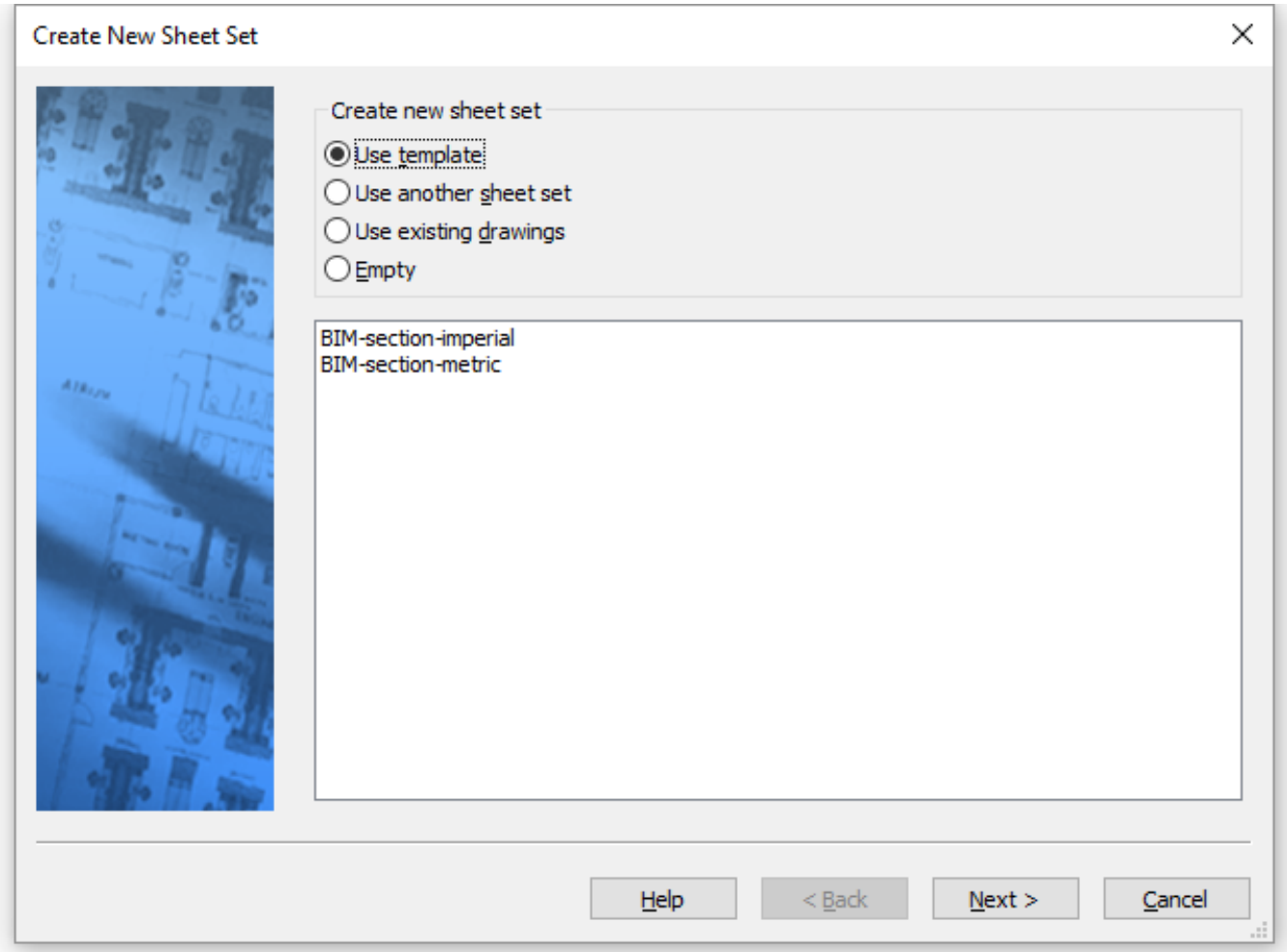
Accessing the Command

 command bar: **newsheetset** menu:**File | New Sheet Set** ribbon:**Home | File | New Sheetset** toolbar: **Drawing Explorer | Sheets |** : **newsheetset** Displays a dialog box that offers four ways to create the new sheet set:

- **Use template** - select a DST sheet set template file from the list shown in the dialog box
- **Use another sheet set -** borrow settings from another DST sheet set file
- **Use existing drawings** - borrow the structure from existing DWG drawing files
- **Empty** - create a new sheet set by starting from scratch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/CreateNewSheetSetDialog.png})

Figure 6.1478



Command Options

Option	Description
Use Template	Creates a new sheet set from a template. Sheet set temples are stored as DST filesin the folder set by the SheetSetTemplatePath (1.13.11)preference (default path is **C:\Users\<UserName>\AppData\Local\Bricsys\BricsCAD\V[Vers Sets**) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/ne template-1.png}) 1. Choose **Use Template.** 2. Select a template sheet set from the list shown in the dialog box 3. Click**Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands template-2.png}) 4. Type a name for the sheet set file in the **Save sheet set file as** field. 5. Click**Browse** .... 6. In the Save New Sheet Set As dialog box, select a folder in which to save the sheet set. **Important**: you cannot progress until you select a folder. 7. Click**Save**. 8. (Optional) Type a description in the **Description** field. 9. Click**Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands template-3.png}) 10. Review the summary of the sheet set that will be created, and then click**Finish**to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

Use another sheet set	Creates a new sheet set from the settings in an existing sheet set: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-dst-1.png}) 1. Choose **Use Another Sheet Set.** 2. Click **Browse**. 3. Select a sheetset file from Select a Sheet Set file dialog box. 4. Click **OK**. 5. Click**Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-dst-2.png}) 6. Type a name for the sheet set file in the **Save sheet set file as** field. 7. Click**Browse**. 8. In the Save New Sheet Set As dialog box, select a folder in which to save the sheet set. **Important**: you cannot progress until you select a folder. 9. Click**Save**. 10. (Optional) Type a description in the **Description** field. 11. Click**Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-dst-3.png}) 12. Review the summary of the sheet set that will be created, and then click**Finish**to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

Use existing drawings

Create a new sheet set from an existing set of drawings: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-dwg-1.png})

1. Choose **Use ExistingDrawings.**
2. Click **Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-dwg-2.png})
3. Click **Select Folder**.
4. Select a folder from the Choose a Folder dialog box. This is the folder in which the sheet set will be stored.
5. Click **Select Folder**. Notice that the dialog box fills with names of drawings and their layouts (see figure above).
6. Choose among the options; is is usually best to keep both options turned on: **Create subsets from folder structure** - uses the folders in the main folder to determine how the sheet set is structured **Prefix sheet titles with drawing file names** - adds the drawing file's name to the start of the sheet set titles, which makes it easier to track the source of the files
7. Click**Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-dwg-3.png})
8. Type a name for the sheet set file in the **Save sheet set file as** field.
9. Click**Browse** ....
10. In the Save New Sheet Set As dialog box, select a folder in which to save the sheet set. **Important**: you cannot progress until you select a folder.
11. Click**Save**.
12. (Optional) Type a description in the **Description** field.
13. Click**Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-dwg-4.png})
14. Review the summary of the sheet set that will be created, and then click**Finish**to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

Empty	Creates an empty sheet set" (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/ne empty-1.png}) 1. Choose **Empty.** 2. Click**Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands empty-2.png}) 3. Type a name for the sheet set file in the **Save sheet set file as** field. 4. Click**Browse**. 5. In the Save New Sheet Set As dialog box, select a folder in which to save the sheet set. **Important**: you cannot progress until you select a folder. 6. Click**Save**. 7. (Optional) Type a description in the **Description** field. 8. Click**Browse** next to **Default Template for New Sheets**. 9. In the Select Drawing Template file dialog box, select a DWT or DWG as the template file. **Important**: you cannot progress until you select a template. 10. Click**Open**. 11. Click**Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands empty-3.png}) 12. Review the summary of the sheet set that will be created, and then click**Finish**to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

Related Commands

SheetSet (6.27.31) - manages sheet sets. Opens the **Sheet Sets** panel. OpenSheetSet (6.23.9) - opens an existing sheet set.

Figure 6.1479

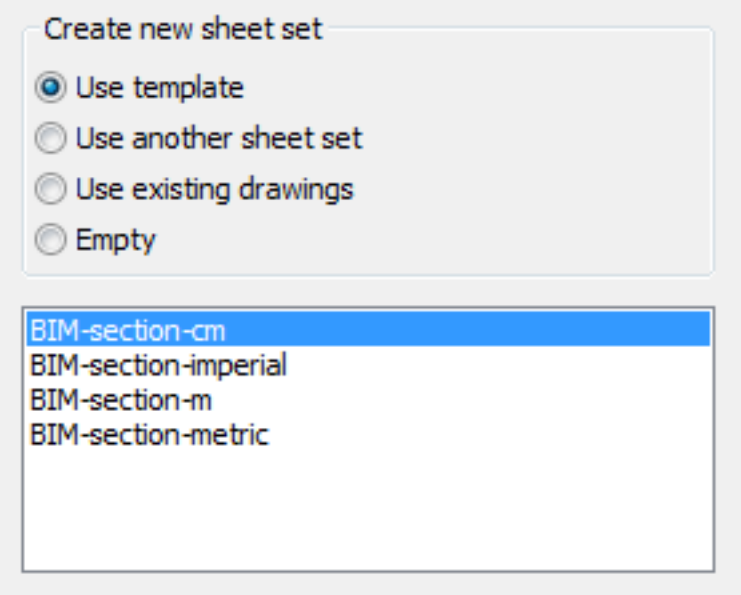


Figure 6.1480

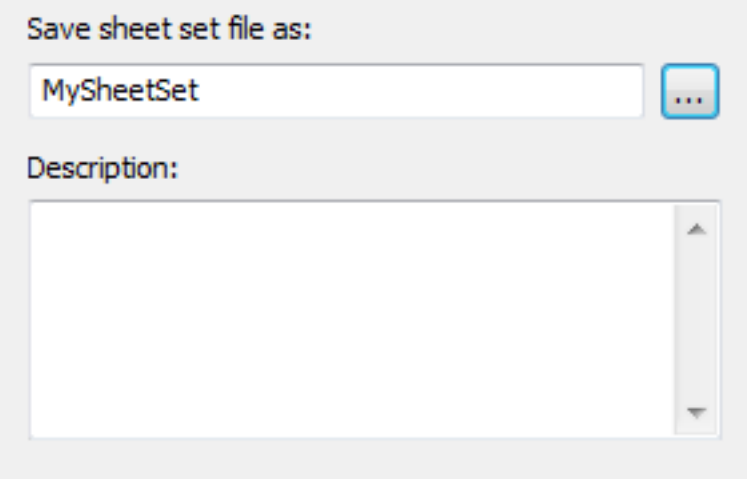


Figure 6.1481

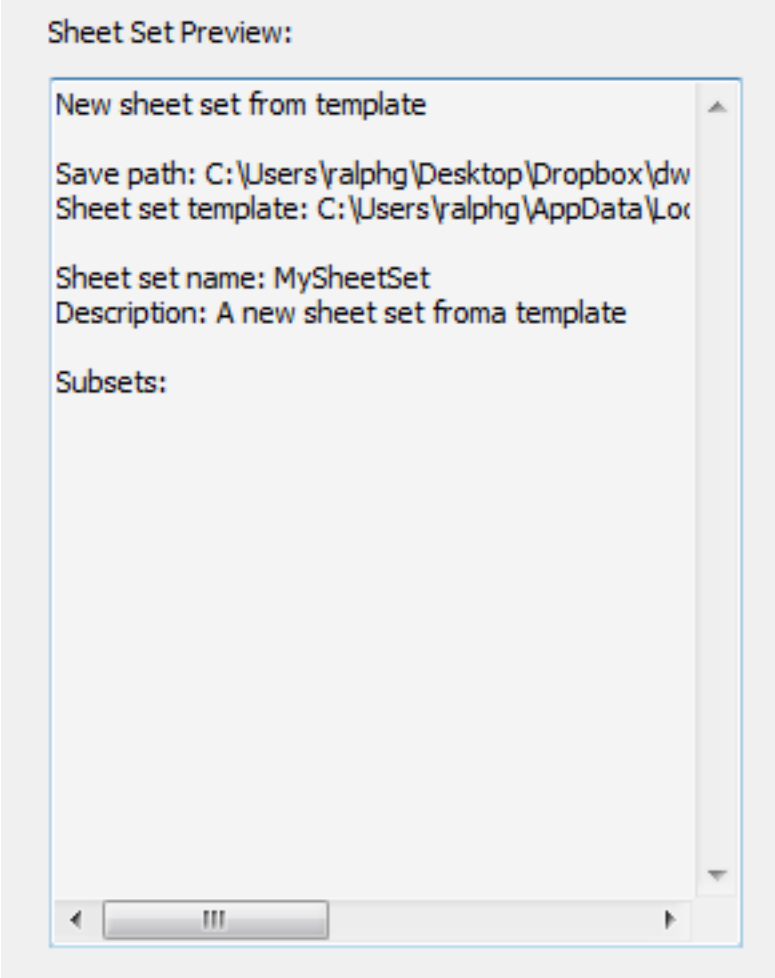


Figure 6.1482

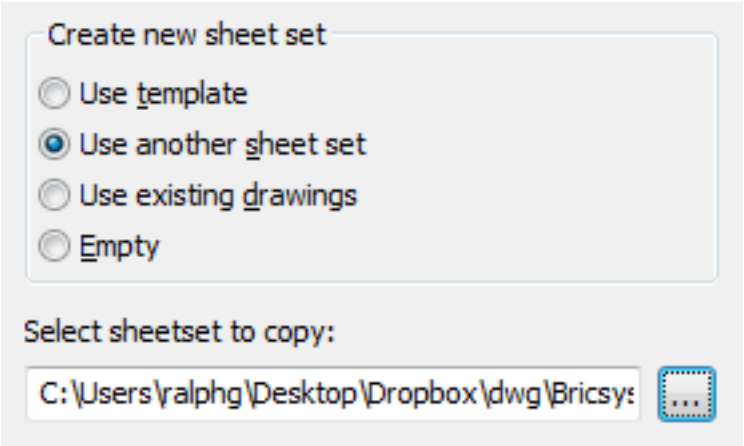


Figure 6.1483

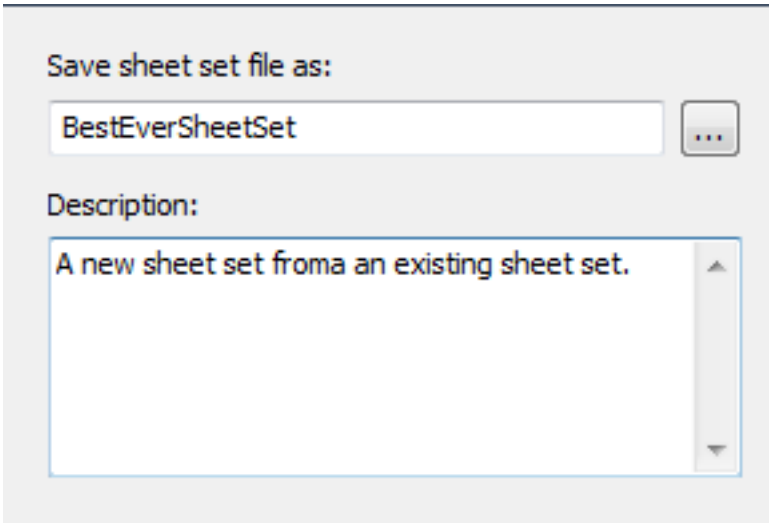


Figure 6.1484

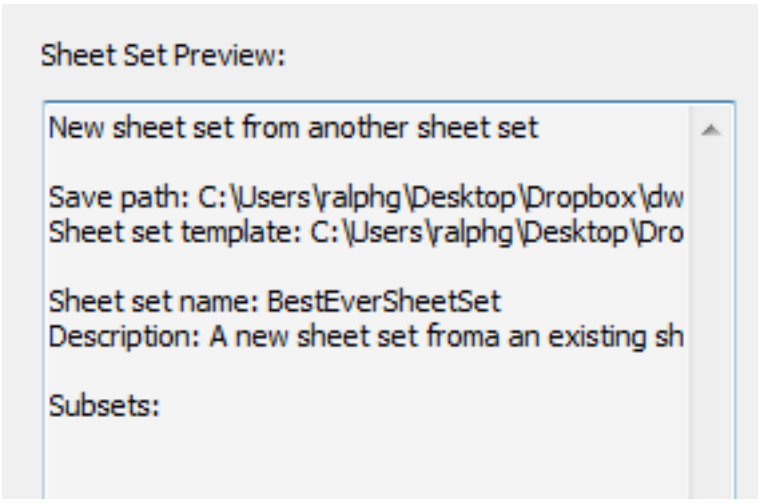


Figure 6.1485

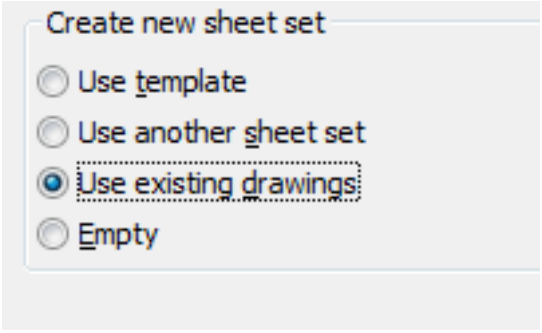


Figure 6.1486

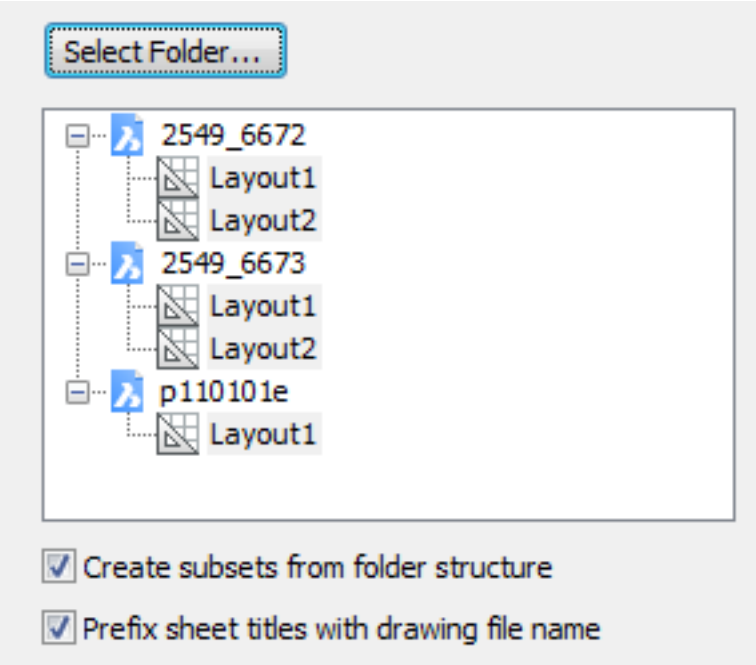


Figure 6.1487

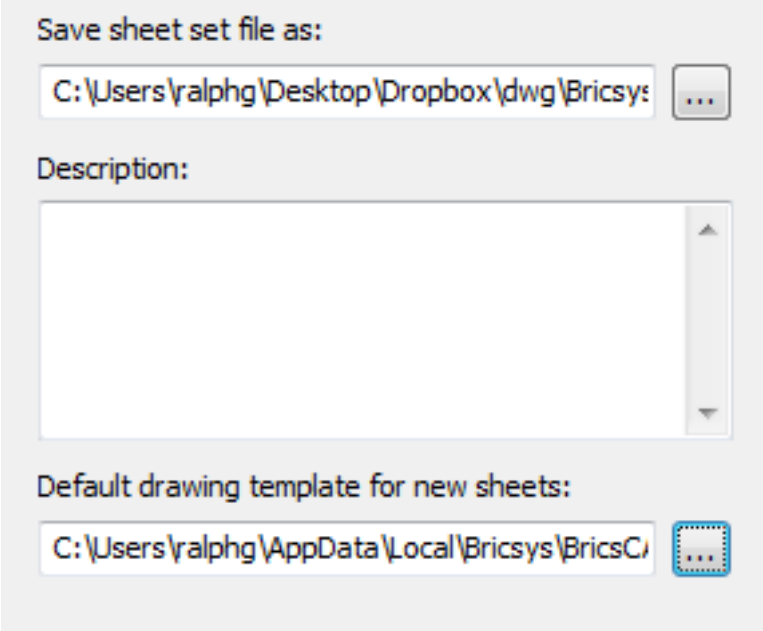


Figure 6.1488

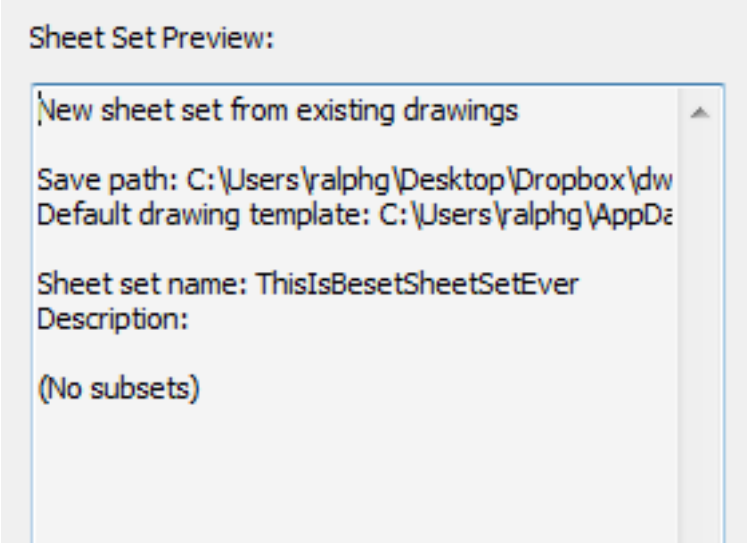


Figure 6.1489

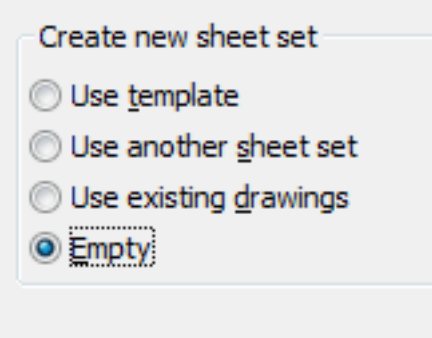


Figure 6.1490

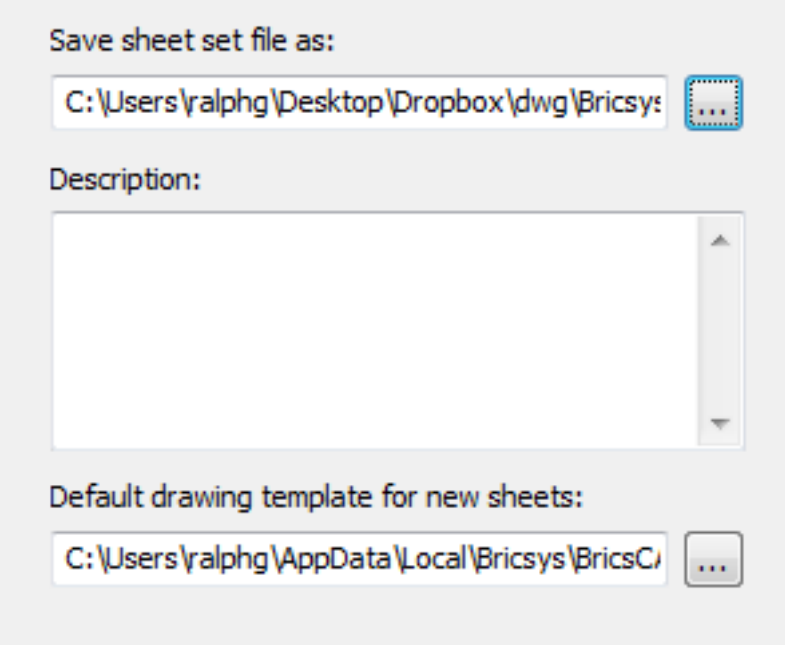


Figure 6.1491

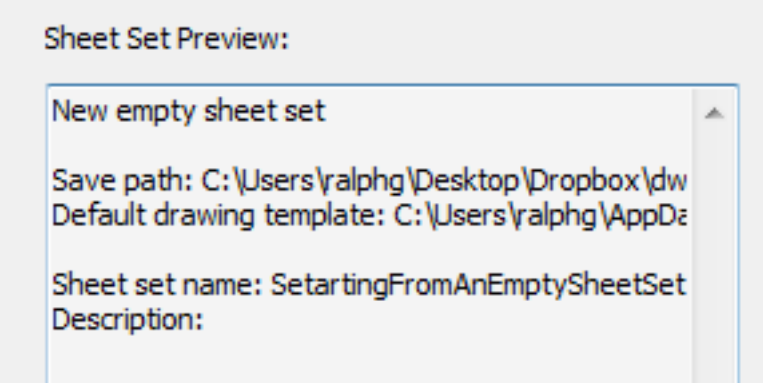


Figure 6.1492



6.22.6 NewWiz

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates new drawings by several methods with the New Drawing Wizard:

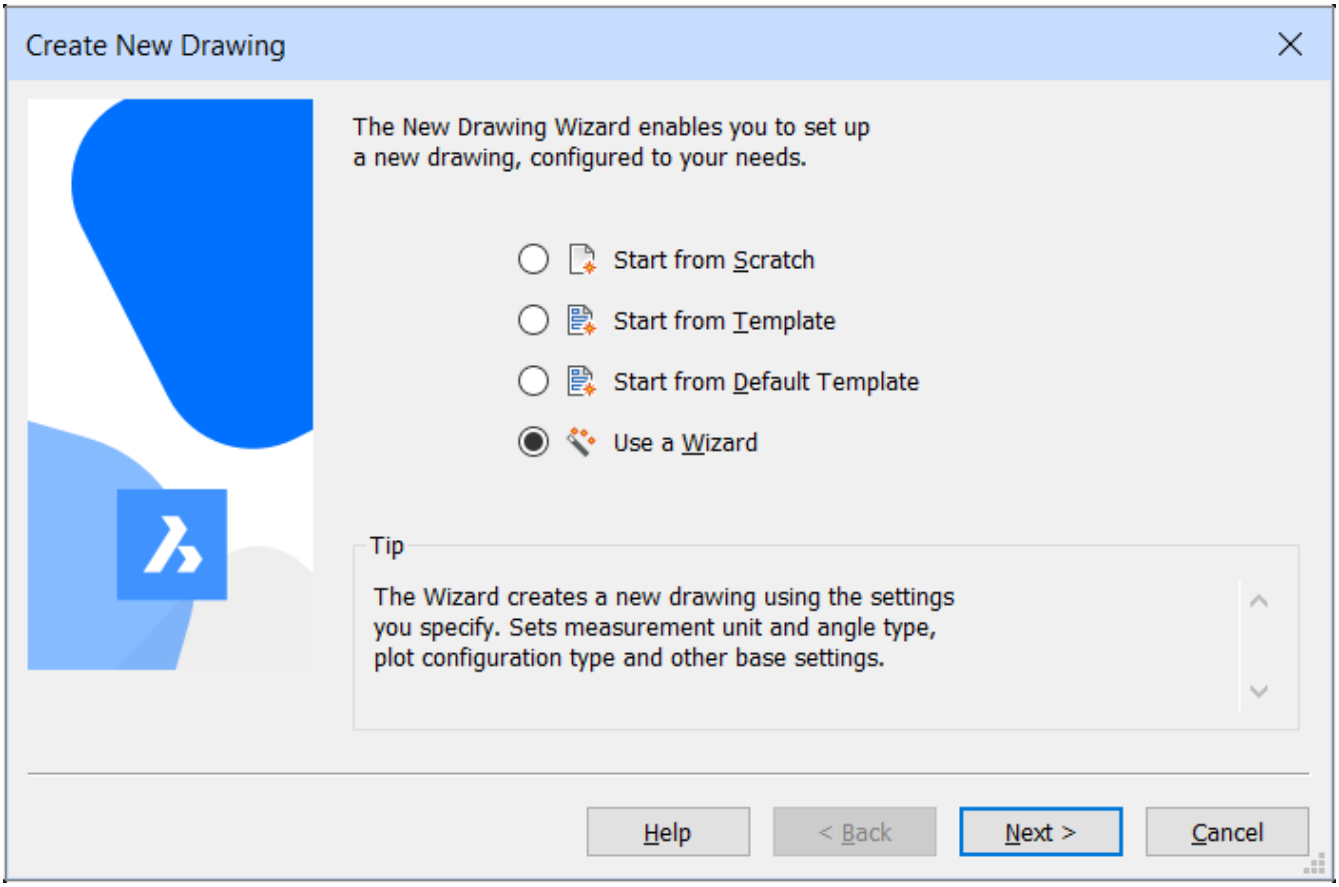
- Starting from scratch
- Creating a new drawing from a drawing template
- Starting with the default DWT template file
- Using a wizard to step you through the process

Note: A wizard is a series of dialog boxes that guide you through a complex setup process.

Accessing the Command


command bar: **newwiz** alias: **ddnew** menu bar: **File | New Wizard** ribbon:**Home | File | New Wizard** : **newwiz** Displays a dialogbox: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/newwiz-dlg.png}) Choose an option, and then click **Next**.

Figure 6.1493



Command Options

Option	Description
Start from Scratch	Starts the new drawing using the default template file specified by the BaseFile variable: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-scratch-1.png}) 1. Choose **Start from Scratch**. 2. Click **Next**. 3. Choose a measurement system: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-scratch-2.png}) • **Imperial (feet and inches)** - new drawing is based on the **default-imperial.dwt** file and so uses Imperial measurements. • **Metric** - new drawing is based on the **default-mm.dwt** file and so uses metric measurements. 4. Click **Finish**. Notice that the program opens a new drawing whose content depends on the default template file.

Start from Template	Starts the new drawing with a template file: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-template-1.png}) 1. Choose **Start from Template**. 2. Click **Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-template-2.png}) 3. Choose a file to act as the template in one of the following ways: • Under **Select a Template**, choose one of the DWT file names. These files are stored in the folder specified by the TemplatePath variable; the default location is**C:\Users\useid\AppData\Local\Bricsys\BricsCAD\V1** • Click **Browse**, and then navigate to a folder with drawing files; you can use the following kinds of file types for a template: **DWG** - drawing files **DWT** - template files **DXF** - drawing interchange files 4. Turn on**Use the selected template as default** option when you wan to use this file as the template from now on. If so, then the program updates the value of the BaseFile variable. 5. Click **Finish**. Notice that the program opens a new drawing whose content depends on the template file you selected.
Start from Default Template	Starts new drawings with the default template defined by the BaseFile variable: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-defaulttemplate-1.png}) 1. Choose **Start From Default Template**. 2. Click **Finish**. Notice that the program opens a new drawing whose content depends on the default template file.

Use a Wizard

Starts the new drawing by stepping through a wizard that guides you along the way: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-wizard-1.png})

1. Choose **Use a Wizard**.
2. Click **Next**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-wizard-2.png})
3. Choose a measurement system:
   - **Imperial (feet and inches)** - new drawing is based on the **default-imperial.dwt** file and so uses Imperial measurements.
   - **Metric** - new drawing is based on the **default-mm.dwt** file and so uses metric measurements.
4. Click **Next**.
5. Select a linear unit of measurement and display precision. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-wizard-3.png})
   - **Unit of Measurement** - choose one from the list. If you are unsure, choose Decimal; you can always change the units any time later with the Units (6.30.8) command.

     ```
     Unit Name            Measures
     ```

   - **Precision** - click the droplist and choose a level of precision.If you are unsure, choose the defaults, as you can always change the precision any time later with the Units (6.30.8) command. Note that this is a display precision: the program makes calculations internally always to eight decimal places.
6. Click **Next**.
7. Select an angular unit of measurement and display precision: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-wizard-4.png})
   - **Angle of Measurement** -choose one form the list. If you are unsure, choose Decimal/Degrees; you can always change the units any time later with the Units (6.30.8) command.

     ```
     Angle Name                  Measures In
     ```

   - **Precision** - click the droplist and choose a level of precision.If you are unsure, choose the defaults, as you can always change the precision any time later with the Units (6.30.8) command.
8. Click **Next**.
9. Determine which kinds of plot style the drawing should employ: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/N/new-wizard-5.png})

Procedures

- To create a drawing using a default template drawing (1.14.1)
- To set the default template drawing (1.14.1)
- To create a drawing from scratch (1.14.1)
- To set the Plot style policy (1.14.1)
- To create a drawing using a template drawing (1.14.1)
- To create a drawing using the New command (1.14.1)
- To set the Template Path variable (1.14.1)
- To create a drawing using a wizard (1.14.1)

Related Commands

Units (6.30.8) - sets units through the Settings dialog box. -Units (6.30.9) - sets units through the command bar. New (6.22.4) - starts new drawing in Imperial or metric units. QNew (6.25.2) - opens new drawings.

Figure 6.1494

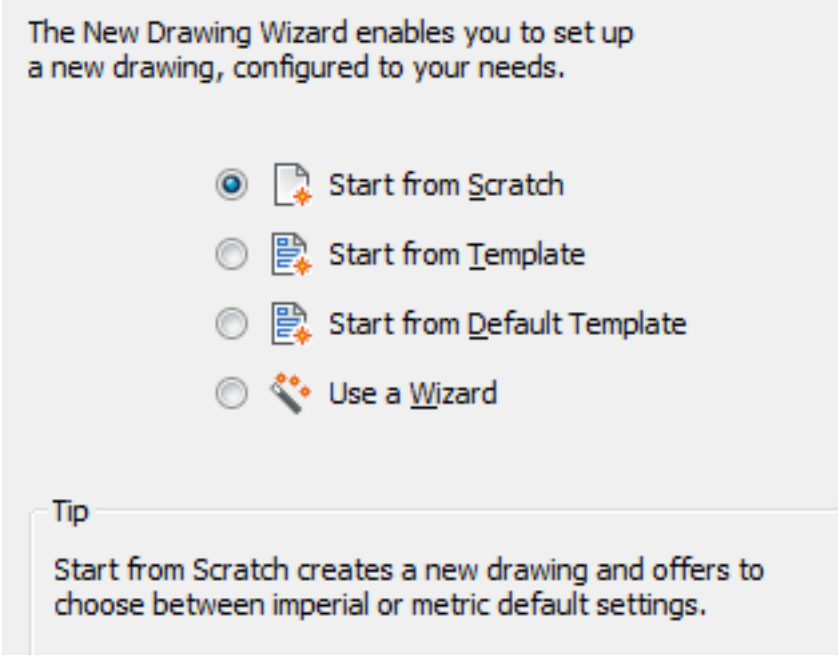


Figure 6.1495

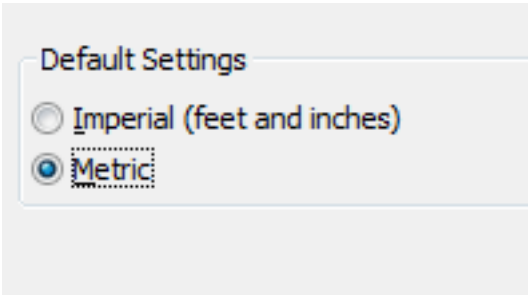


Figure 6.1496

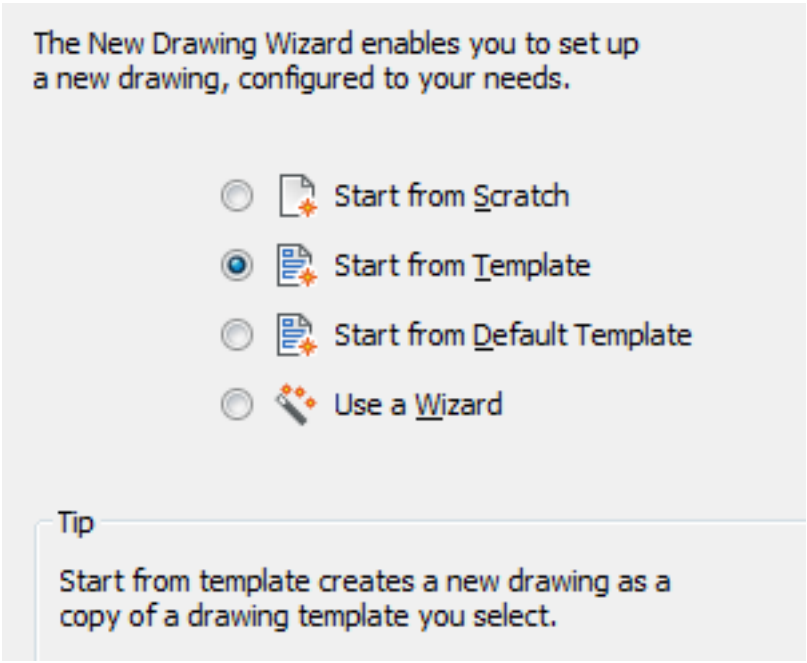


Figure 6.1497

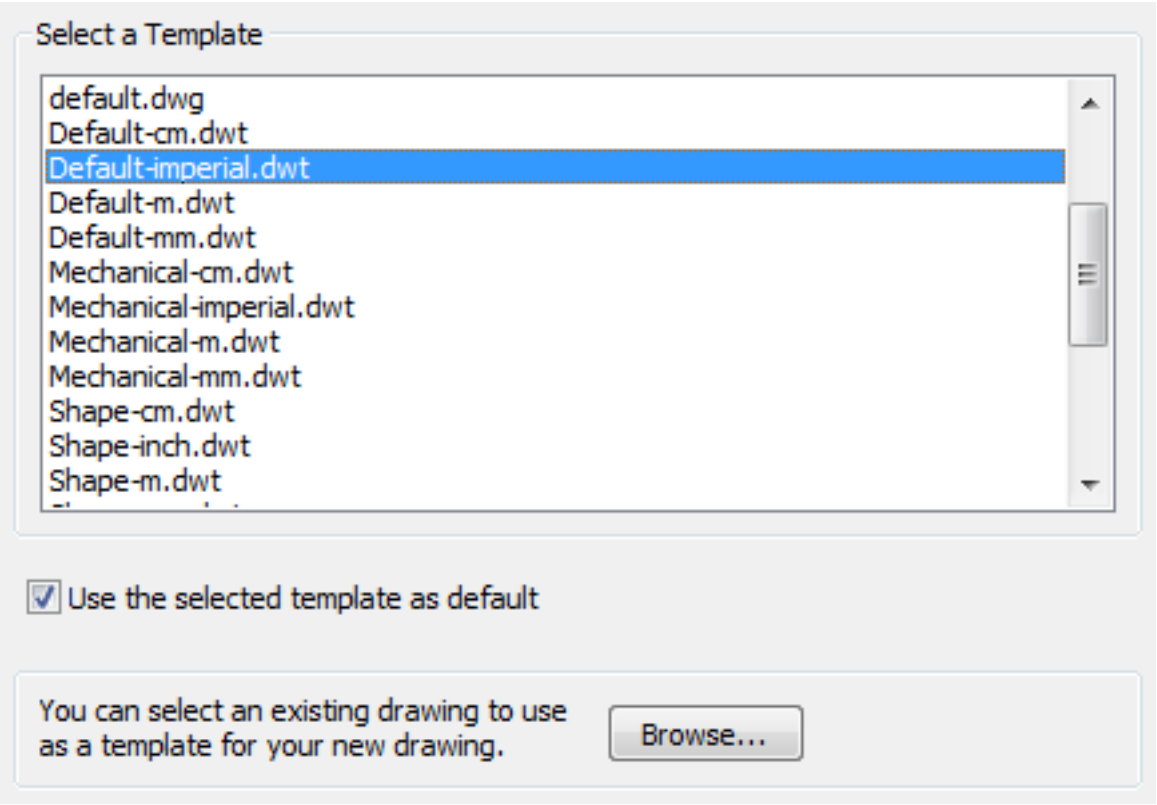


Figure 6.1498

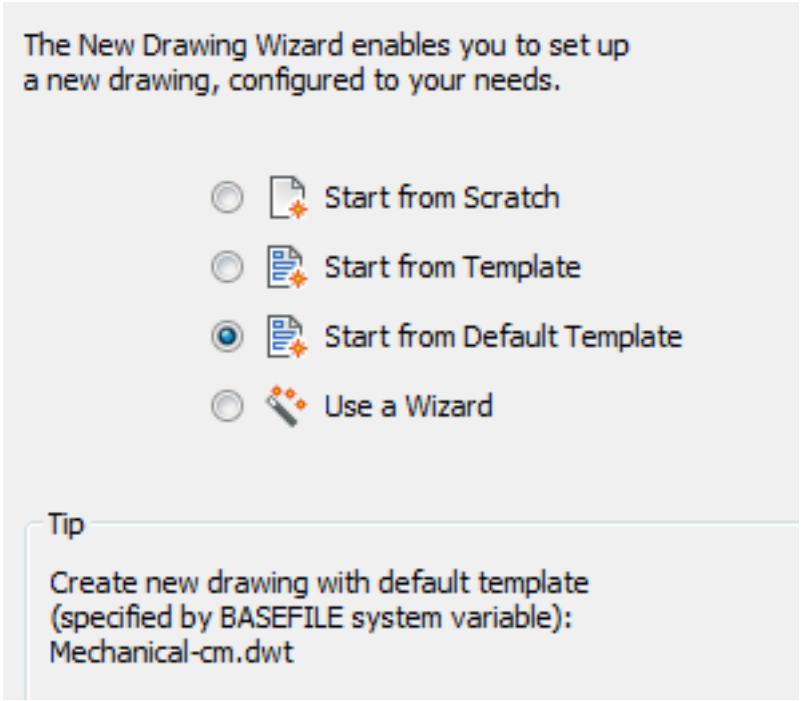


Figure 6.1499

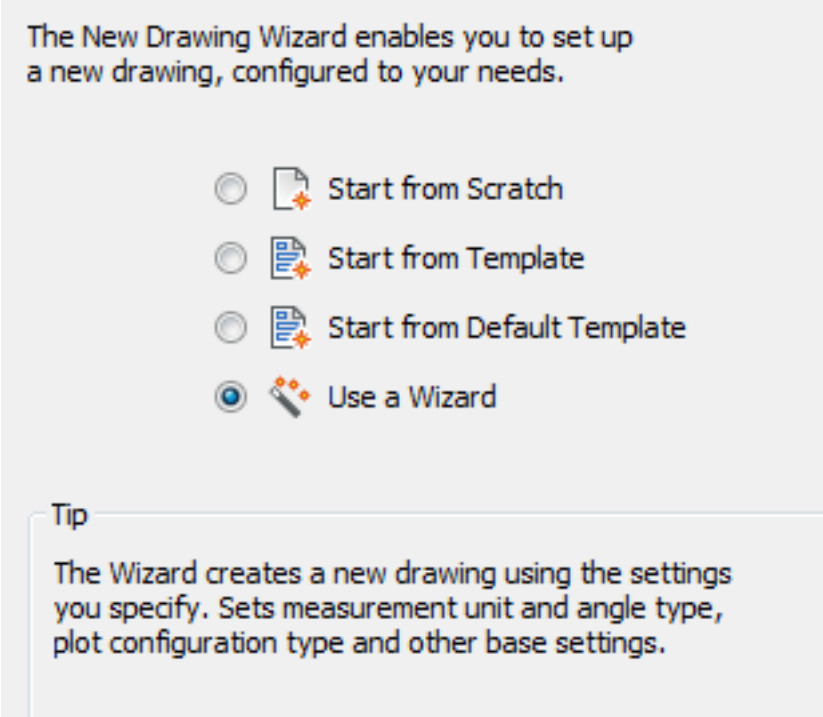


Figure 6.1500

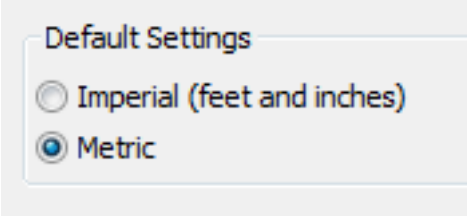


Figure 6.1501

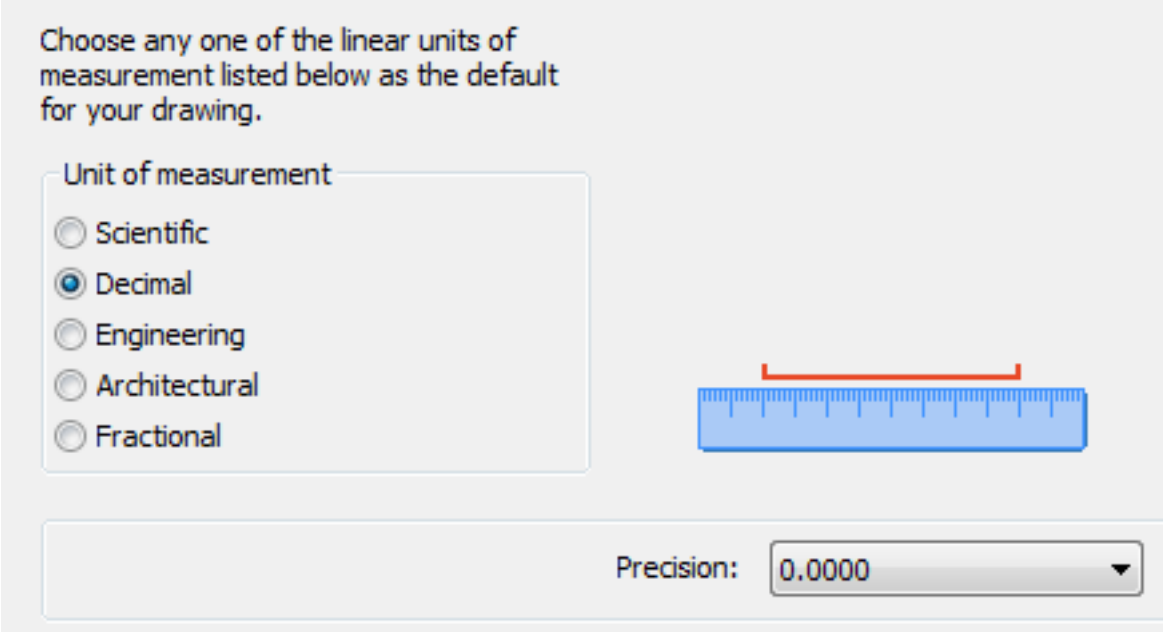


Figure 6.1502

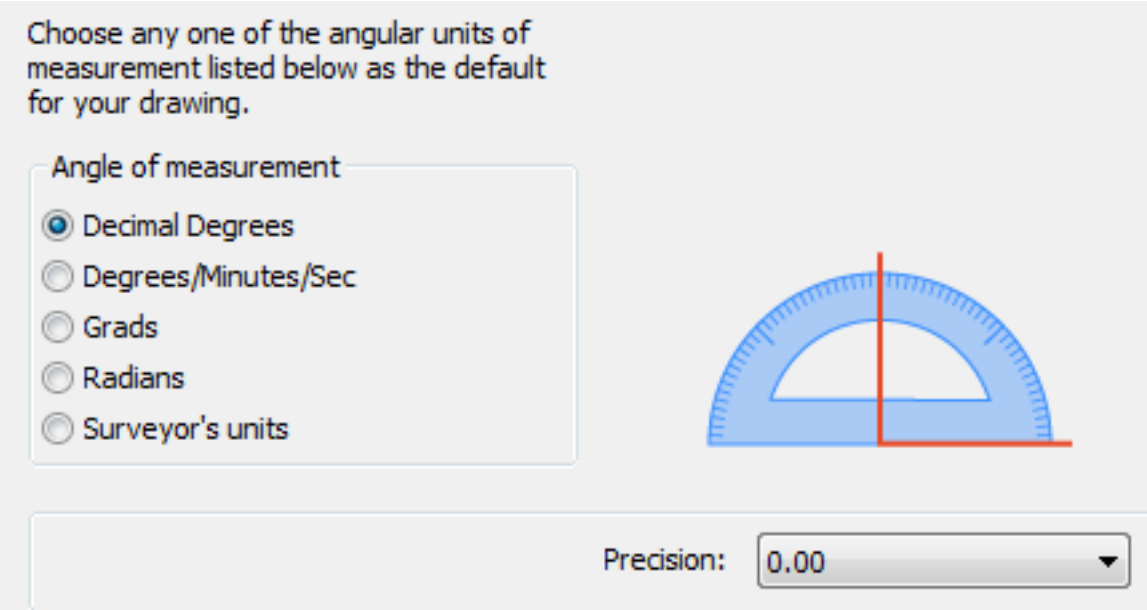


Figure 6.1503

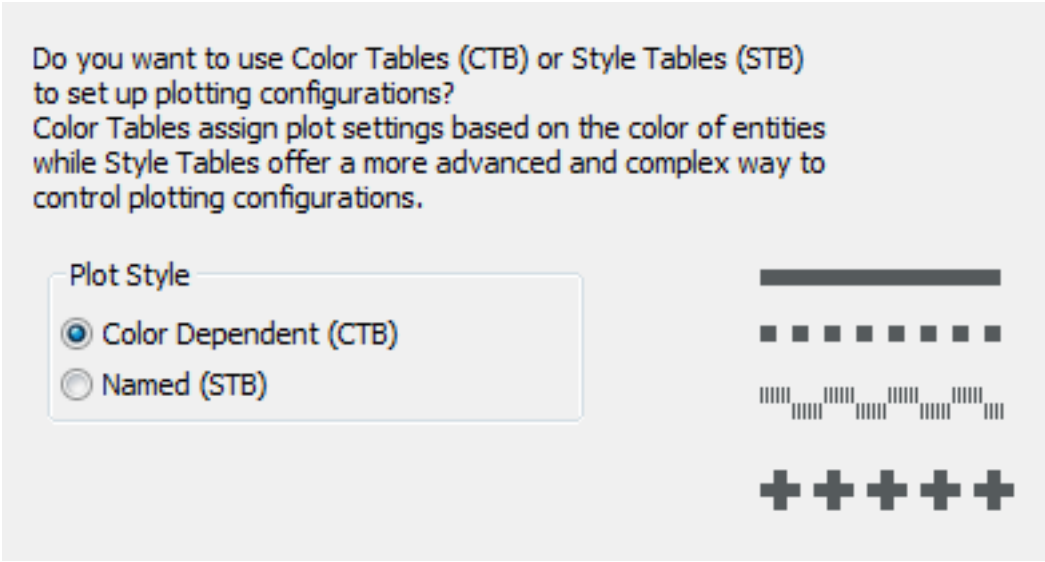


Figure 6.1504

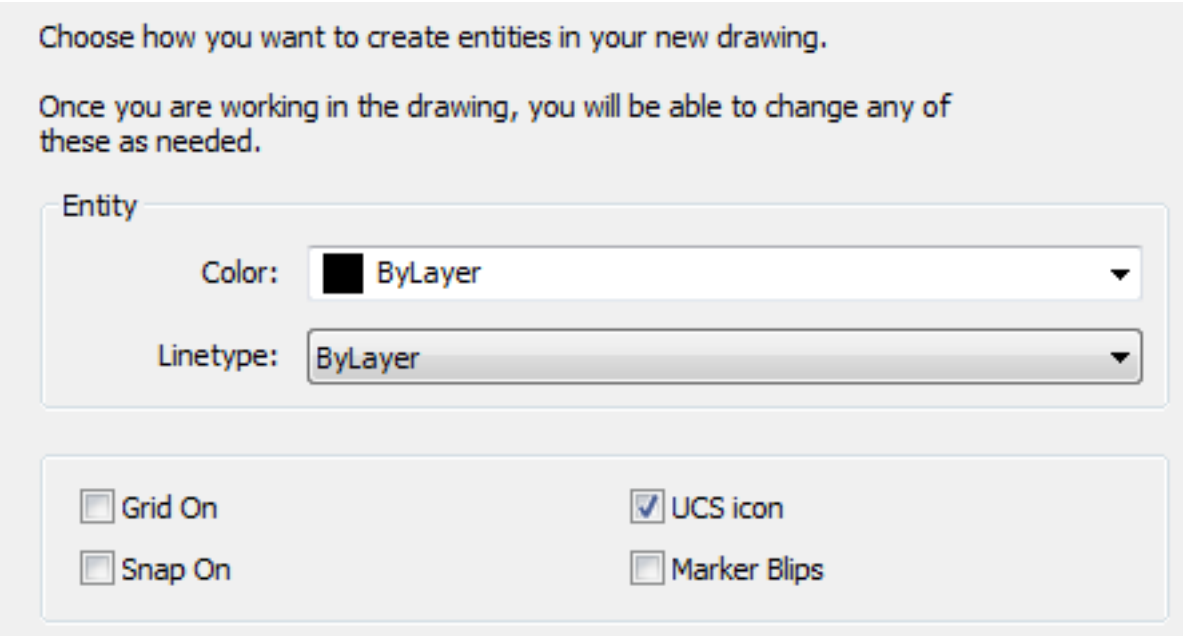


Figure 6.1505



Figure 6.1506



6.22.7 Node

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles Node entity snap mode, which snaps to the following entities with nodes:

- Point entities
- Text insertion points
- Mtext insertion points
- Attribute insertion points
- Table insertion points

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image334.png})

Accessing the Command

command bar: **node** transparent:**node** alias: **nod** ribbon:**Home | Entity Snaps | Snap to Node** status bar: right-click **ESNAP**, choose **Node** toolbar: **Entity Snaps |** : **node** Displays no prompts in the command bar. When node entity snap is on, the cursor snaps to point entities, including mtext entities.

Command Options

Option	Description
Node	Each time the command is entered, it toggles Node entity snap on and off. When on, it snaps to point, mtext, and other entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.1507



6.22.8 None

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Turns off all entity snap modes. **Note:**To turn on entity snap modes, see the ESnap command.

Accessing the Command

 command bar: **none** transparent: **non** ribbon:**Home | Entity Snaps | Clear Entity Snaps** status bar: right-click **ESNAP**, choose **Off** toolbar: **Entity Snaps |**  : **none** Displays no prompts in the command bar. Turns off all entity snap modes.

Command Options

Option	Description
None	Entering the command turns off all entity snap modes. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.1508



6.22.9 Number

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates incremental number tags for BIM entities.

Accessing the Command



Keyboard: **number : number Prompts you:** Select entities to number and press Enter: **(Select the entities to number.)** Entities in set: xx Select entities to number and press Enter: **(Select more entities or press Enter to stop.)** Provide First index or [Increment/Prefix/Suffix/Number style/Entities sorting/Overwrite Numbers] <1>: **(Choose an option or press Enter to accept the default.)** Total number of entities numbered is xx Total number of entities skipped is yy

Command Options

Option	Description
First Index	Sets the first index. Type an integer.
Increment	Sets the increment for the numbering. Prompst you: **Provide the increment** <1>: **Type an integer.**
Prefix	Specifies a prefix for the numbering. Prompts you: **Provide the prefix** <””>: **Type a string.**
Suffix	Specifies a suffix for the numbering. Prompts you: **Provide the prefix** <””>: **Type a string.**
Number Style	Defines the number style. Prompts you: **Select number formatting** [arabic (0)/roman (1)/roman lowercase (2)/letters (3)/letters lowercase (4)] <Arabic>: **Choose an option:** • **0**: arabic numbers (1, 2, 3, ...) • **1**: roman uppercase numbers (I, II, III, ...) • **2**:roman lowercase numbers (i, ii, iii, ...) • **3**: uppercase letters (A, B, C, ...) • **4**: lowercase letters (a, b, c, ...)
Entities Sorting	Allows to specify the sorting order. Prompts you: **Current sorting orde**r = **Select a coordinate to sort by or press Enter to stop [X/Y/Z/None] <None>: Choose an option.** • **None**: from oldest to most recent. • **X**: based on the value of the x-coordinate, from low to high. • **Y**:based on the value of the y-coordinate, from low to high. • **Z**:based on the value of the z-coordinate, from low to high.

Overwrite Numbers	Allows to overwrite or keep existing numbers. Prompts you: **Existing Number are** [Kept/Overwritten] <Overwritten>: **Choose an option.** • **Kept**: Existing numbers are kept. • **Overwritten**: Existing numbers are overwritten.

Procedures

Renumber grid labels (2.4.13)

Related Commands

BimGrid (6.7.12): Creates 2-dimensional rectangular and radial grids.

6.23 O

6.23.1 ObjectScale

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adds or removes scale factors used with annotative entities through a dialog box.
Note: Annotative scaling adjusts scale factors of annotative items, such as text and hatch patterns, to match the plot scale.

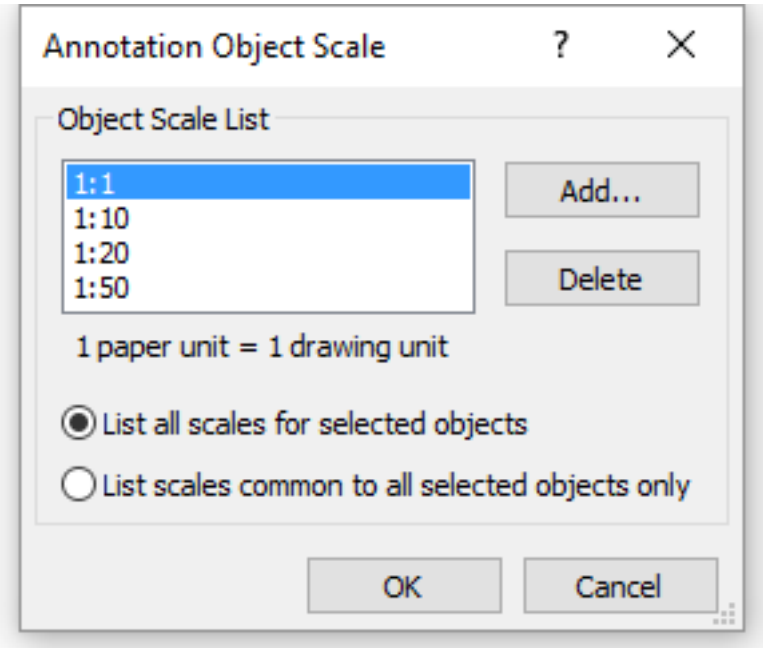
Figure 6.1509



Accessing the Command


command bar: **objectscale** ribbon: **Draw | Annotation Scaling | Add/Delete Scales** : **objectscale Prompts you in the command bar:** Select annotative entities: **(Select one or more entities)** Entities in set: 2 Select annotative entities: **(Select more entities, or else or press Enter to continue)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/AnnotationObjectScaleDialog.png})

Figure 6.1510



Command Options

Option	Description
Select annotative entities	Selects the entities whose annotative scales should be adjusted: • Select one or more items using any selection method. • Enter **all** to select all entities in the drawing; the program filters out ineligible entities automatically. • Press **Enter** to end entity selection.
Add	Adds annotation scale factors to the selected entities; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/A 1. Select a scale factor from the list To select more than one, hold down the **Ctrl** key. 2. Click **OK** when done.
Delete	Removes annotation scales from the selected entities; this option does not work when there is just one scale factor attached to the entities. 1. Select one or more scale factors; hold down the Ctrl key to select more than one. 2. Click**Delete**. Choose **No** in the **Annotative** field on the Properties Bar (6.24.49) to remove the annotative property.

Scales to list	Specifies which scale factors are displayed by the dialog box; choose one: **List all scales for selected entities** -lists all the scale factors **List scales common to all selected entities only** -lists only the scale factor shared by the selected entities; this option is meaningless when only one entity is selected
OK	Closes the dialog box and applies the changes.

Related Commands

-ObjectScale (6.23.2) - adds or removes supported scales for annotative entities at the command line. ScaleListEdit (6.27.7) - edits the scale list of a drawing, used in **Plot Scale** section of the **Print** and **Page Setup** dialogs and the **Standard scale** property of a paper space viewport. Defines the available annotation scales for annotative entities.

Figure 6.1511

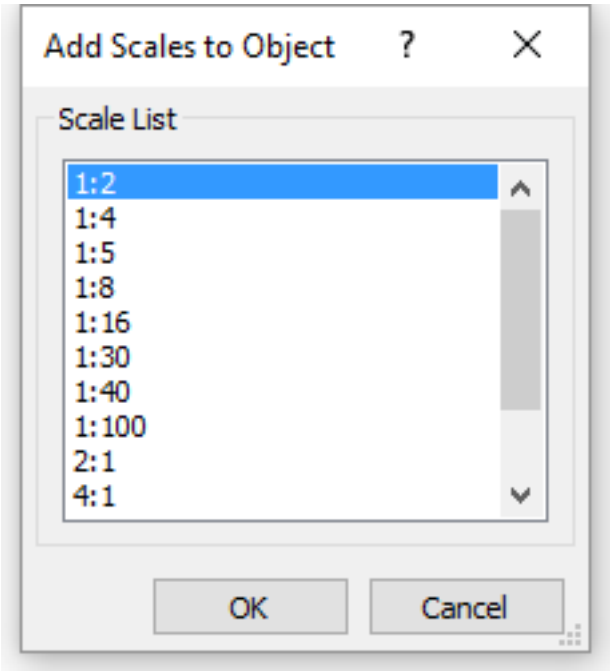


Figure 6.1512



6.23.2 -ObjectScale

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adds or removes scale factors used with annotative entities through the command line. **Note:** Annotative scaling adjusts scale factors of annotative items, such as text and hatch patterns, to match the plot scale.

Accessing the Command

command bar: **-objectscale** : **-objectscale Prompts you in the command bar:** Select annotative entities: **(Select entities)** Entities in set: 1 Select annotative entities: **(Select more entities, or else press Enter to continue)** Enter an option [Add/Delete/?] <Add>: **(Enter an option)**

Command Options

Option	Description
Select annotative entities	Selects the entities whose annotative scales should be adjusted: • Select one or more items using any selection method. • Enter **all** to select all entities in the drawing; the program filters out ineligible entities automatically. • Press **Enter** to end entity selection.
Add	Adds annotation scales to the selected annotative entities. When you enter a scale factor at the prompt, enter only the divisor. For instance, to add a scale of 1:5, type only the 5. 1. **Enter name scale to add or [? to list] <1:10>** - **type a scale, then press Enter x entities updated to support annotation scale '1:10'.** 2. **Enter name scale to add or [? to list]** - **type another scale or press Enter to stop** Enter **?** to list the annotative scale factors available, as listed below.
Delete	Deletes annotation scales from the selected annotative entities; prompts you: 1. **Enter name scale to delete or [?] <1:10>** - **type a scale factor, then press Enter x entities scale removed.** 2. **Enter name scale to delete or [?]**- **type another scale, or else press Enter to stop** Type **?** to display a list of all scales of the selected objects.Press **F2** to open the **Prompt History** window. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

?	Displays a list of available annotation scales as defined in the scale list (see the ScaleListEdit (6.27.7) command).Press **F2** to open the Prompt History window. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

Related Commands

ObjectScale (6.23.1) - adds or removes supported scales for annotative entities through a dialog box. ScaleListEdit (6.27.7) - Edits the scale list of a drawing, used in **Plot Scale** section of the **Print** and **Page Setup** dialogs and the **Standard scale** property of a paper space viewport. Defines the available annotation scales for annotative entities.

Figure 6.1513

	Scale Name	Paper Units	Drawing Units	Effective Scale
1:	1:1	1.0000	1.0000	1.0000
2:	1:20	1.0000	20.0000	0.0500
3:	1:50	1.0000	50.0000	0.0200

Figure 6.1514

	Scale Name	Paper Units	Drawing Units	Effective Scale
1:	100:1	100.0000	1.0000	100.0000
2:	10:1	10.0000	1.0000	10.0000
3:	1:1	1.0000	1.0000	1.0000
4:	1:10	1.0000	10.0000	0.1000
5:	1:100	1.0000	100.0000	0.0100
6:	1:125	1.0000	125.0000	0.0080
7:	1:16	1.0000	16.0000	0.0625
8:	1:2	1.0000	2.0000	0.5000
9:	1:20	1.0000	20.0000	0.0500
10:	1:25	1.0000	25.0000	0.0400
11:	1:250	1.0000	250.0000	0.0040
12:	1:30	1.0000	30.0000	0.0333
13:	1:4	1.0000	4.0000	0.2500
14:	1:40	1.0000	40.0000	0.0250
15:	1:5	1.0000	5.0000	0.2000
16:	1:50	1.0000	50.0000	0.0200
17:	1:8	1.0000	8.0000	0.1250
18:	2:1	2.0000	1.0000	2.0000
19:	4:1	4.0000	1.0000	4.0000
20:	8:1	8.0000	1.0000	8.0000

Figure 6.1515

Shape Classic Pro Platinum Mechanical BIM

6.23.3 Offset

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates parallel copies of 2D entities and 3D solid faces. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/C

Accessing the Command

command bar: **offset** alias: **o** menu bar: **Modify | Offset** toolbar: **Modify |** ribbon:**Change | Modify | Offset** Quad: **2D Editing |** : **offset Prompts you in the command bar:** Enter offset distance or [Through point/Erase/Layer] <20.0>: **(Enter a distance, pick two points, press Enter for the default distance, or else select an option)** Select entity/subentity or [Exit] <Exit>: **(Choose a single entity, or press Enter to exit the command)** Select side for parallel copy or [Both sides/Multiple]: **(Pick a point to place the copy, or else choose an option)** Select entity/subentity or [Undo/Exit] <Exit>: **(Choose another entity, type U to undo the previous parallel copy, or press Enter to exit this command) Note:** When curved entities are offset, the command changes the radii of the copies: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image341.png}) When polylines are being offset, theOFFSETGAPTYPE (6.23.3)variable determines how potential gaps between segments are treated: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/O/offset-eg.png})

- **0** - gaps are closed, with segments extended to close them
- **1** - gaps are filleted, with the radius of the arc segments the size of the offset distance
- **2** - gaps are chamfered, with the distance equals to the offset distance

When faces of 3D models are offset, theSELECTIONMODES variable determines how potential conflicts are handled:

- **1** - edges are selected and offset
- **2** - faces are selected and offset (figure below, at left)
- **4** - boundaries are detected and offset (boundary is in blue in the figure below, at right)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/OffsetFace.png}) (\vref {fig:C:/Dev/T
2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/OffsetBoundary1.png}) Boundaries are also detected by the Offset command for 2D entities, such as these overlapping squares: (\vref {fig:C:/Dev/Task-2448/images/staticbr

Figure 6.1516

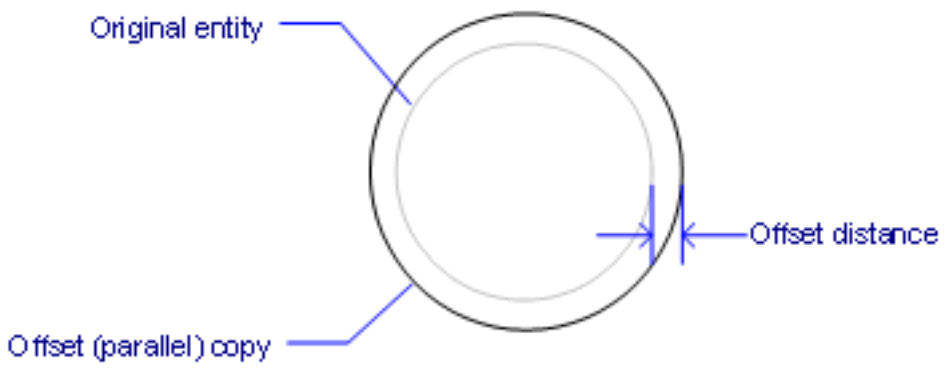


Figure 6.1517

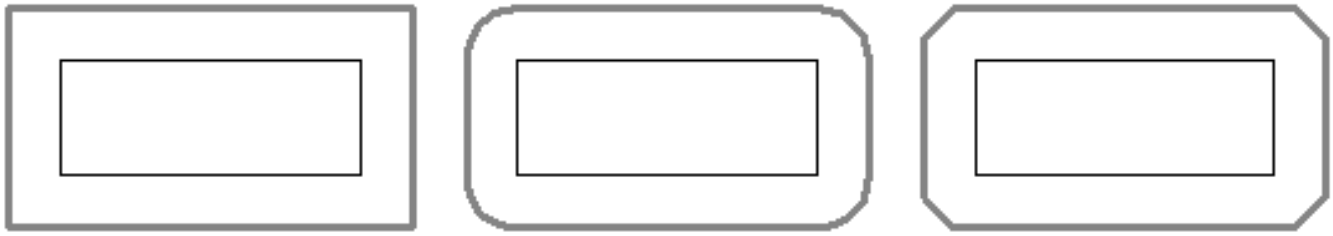


Figure 6.1518

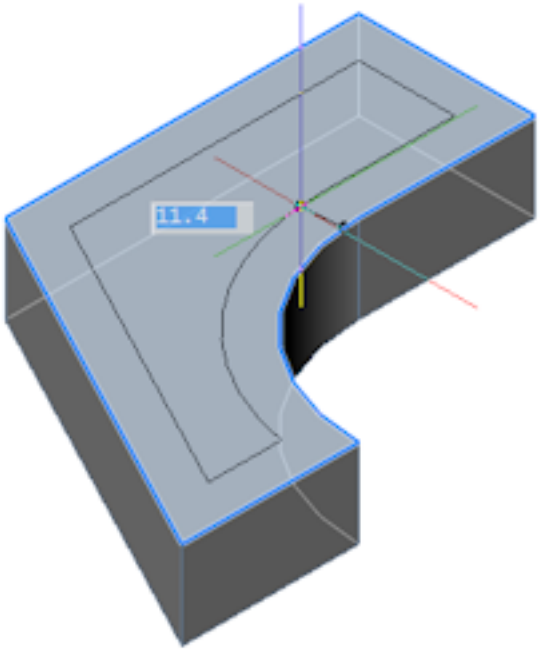


Figure 6.1519

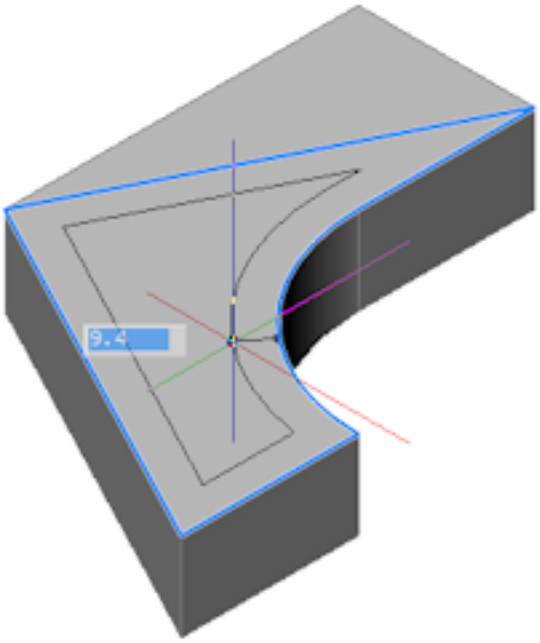
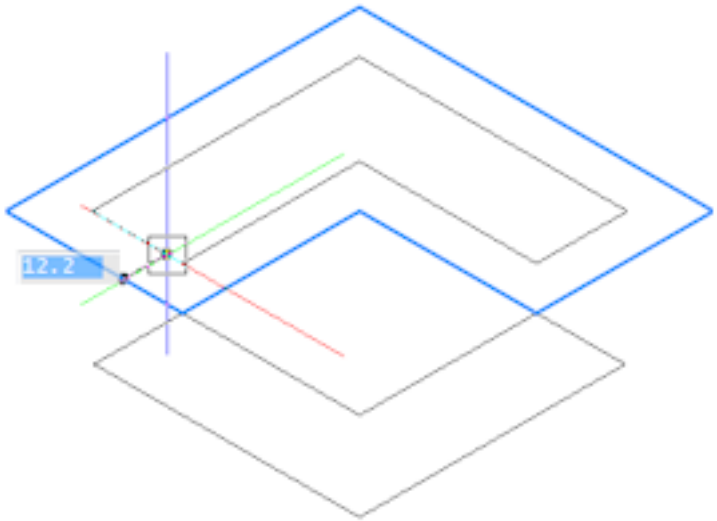


Figure 6.1520



Command Options

Option	Description
Enter offset distance	Specifies the distance between the original entity and its parallel copy: • Press **Enter** to accept the default distance. • Type a value. • Specify two points in the drawing, such as along another entity. The offset distance is saved to the OFFSETDISTvariable.
Through point	Specifies the offset distance by picking two points; prompts you: 1. **Select entity/subentity or [Exit] <Exit>** - enter an option: • **Select entity** - select a single entity • **Subentity** - select the face of a 3D solid • **Exit** - return to the previous prompt 2. **Select through point or [Multiple]** - enter an option: • **Select through point** - pick a point in the drawing • **Multiple** - repeats the 'Select next through point' prompt until you press Esc or Enter to return to the previous prompt; see below
Erase	Deletes the source entity following the offset operation. The status is saved to the OFFSETERASEvariable with the drawing, but is then reset (turned off) with the next drawing.

Layer	Specifies the layer on which to place the copied entity; prompts you: **Select layer for offset entities? [Current/Source] <Source>** - enter an option: • **Current** - places the the offset entity on the current layer. • **Source** - places the offset on the same layer as that of the source entity.
Select entity/subentity	Selects the entity of which to make the parallel copy: • **Entity** - 2D entities that lay in a plane; this means you can offset lines, 2D polylines, and splines, but not helices or 3D polylines. • **Subentity** - flat 2D parts of 3D solid model; this means you can offset the base of a cone but not its curved surface. Only one entity can be selected for offsetting at a time; the command repeats automatically so that you can offset additional entities.Press **Enter** or **Esc** to exit the command.
Side for parallel copy	Specifies the side on which to make the offset (parallel) copy. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i If the Through Point option was selected earlier, your pick point specifies the offset distance.
Both sides	Places offset copies on both sides of the entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Exit	Returns to the previous prompt of this command, or ends the command.
Undo	Undoes the previous offset operation.

Procedures

- Making a parallel copy at a specified distance (1.23.18)
- Making a parallel copy through a point (1.23.18)
- Making a parallel copy of a 3D solid edge (1.23.18)
- Making a parallel copy a 3D solid face (1.23.18)
- Making a parallel copy of a detected boundary (1.23.18)

Related Command

Copy (6.9.50) - makes copies of entities.

Figure 6.1521

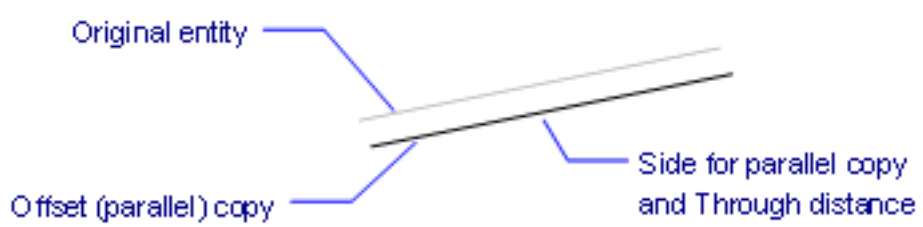


Figure 6.1522

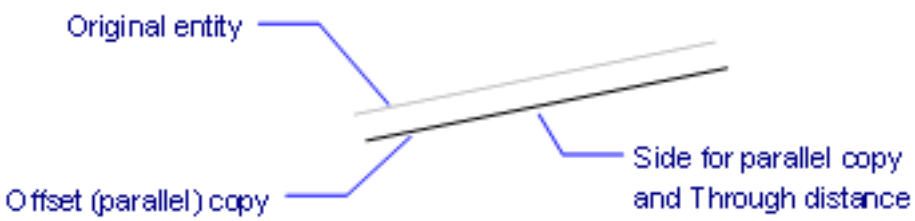


Figure 6.1523

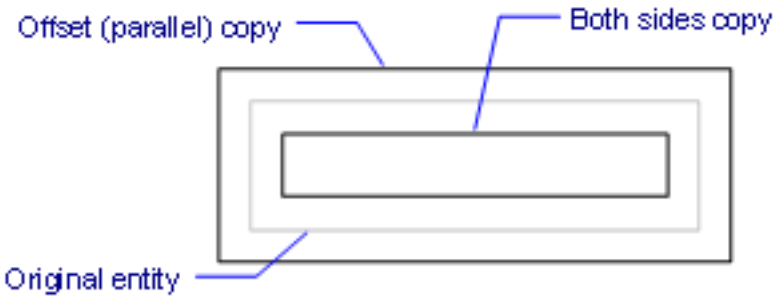


Figure 6.1524



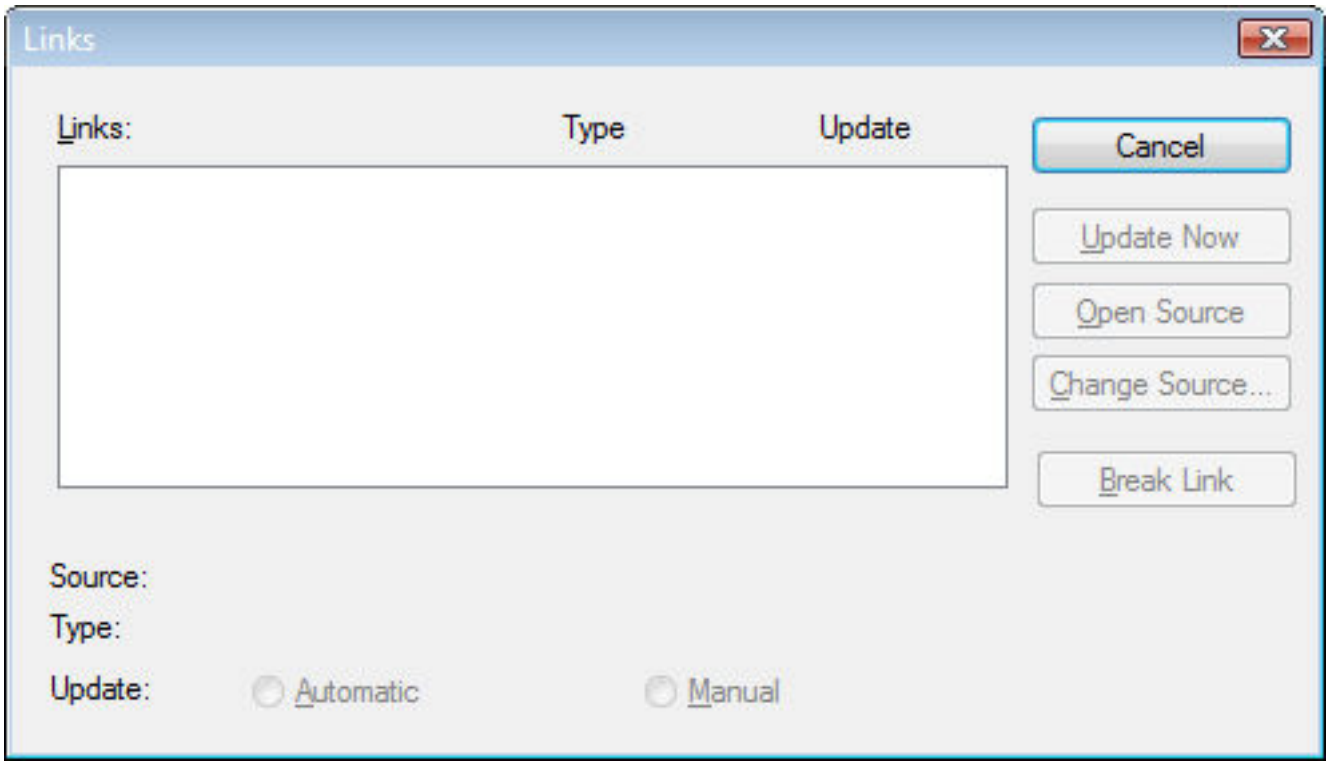
6.23.4 OleLinks

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Edits the links of OLE entities that were embedded or linked to drawings (short for ”object linking and embedding linking”). **Note:** Use the InsertObj (6.17.17) command to insert OLE objects in drawings.

Accessing the Command

command bar: **olelinks** menu bar: **Edit | Links** : **olelinks** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbrics
Change a setting for a link, and then click **Update Now**.

Figure 6.1525



Option	Description
Update Now	Updates linked OLE entities.When the source file changes, use this button to update the copy in the BricsCAD drawing.
Open Source	Opens the source file in the source application. This allows you to edit the source file.When done editing, choose **Update & Exit** from the File menu.
Change Source	Changes the source for the linked OLE entity.
Break Link	Breaks the link with the source file. When the link is broken, the OLE entity in the drawing no longer changes when the source file changes.
Source	Reports the path to the source file name.
Type	Reports the file type.
Update	Determines how the OLE entity in the drawing is updated, when the source changes: • **Automatic** - the OLE entity in the drawing changes whenever the source file changes. • **Manual** - the OLE entity in the drawing changes when you click the Update Now button.

Command Options

Related Commands

OleOpen (6.23.5) - opens linked and embedded objects in the appropriate application. PasteSpec (6.24.11) - pastes entities in several formats. InsertObj (6.17.17) - inserts foreign (non-BricsCAD) objects into drawings Insert (6.17.13) - inserts DWG files in drawings. ImageAttach (6.17.5) - attaches raster images to drawings. XAttach (6.34.1) - attaches DWG files to drawings.

Figure 6.1526



6.23.5 OleOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Opens the selected OLE object in the source application for editing. **Note:**Use the InsertObj (6.17.17) command to insert OLE objects in drawings.

Accessing the Command

command bar: **oleopen** : **oleopen** Prompts you: **Select OLE item to open: (Select an OLE object in the drawing)** The selected OLE object is opened.

Command Options

Option	Description
Select OLE item to open	Select an OLE object in the drawing. Alternatively, you can double-click the OLE object to open it in its source application. When you are finished editing the object in the source application, choose **File | Exit and Return to Drawing**. If the link between the drawing and the source application is broken, the OLE object cannot open.

Related Commands

OleLinks (6.23.4) - adjusts links of OLE entities embedded in or linked to drawings PasteSpec (6.24.11) - pastes entities in several formats. InsertObj (6.17.17) - inserts foreign (non-BricsCAD) objects into drawings

Figure 6.1527



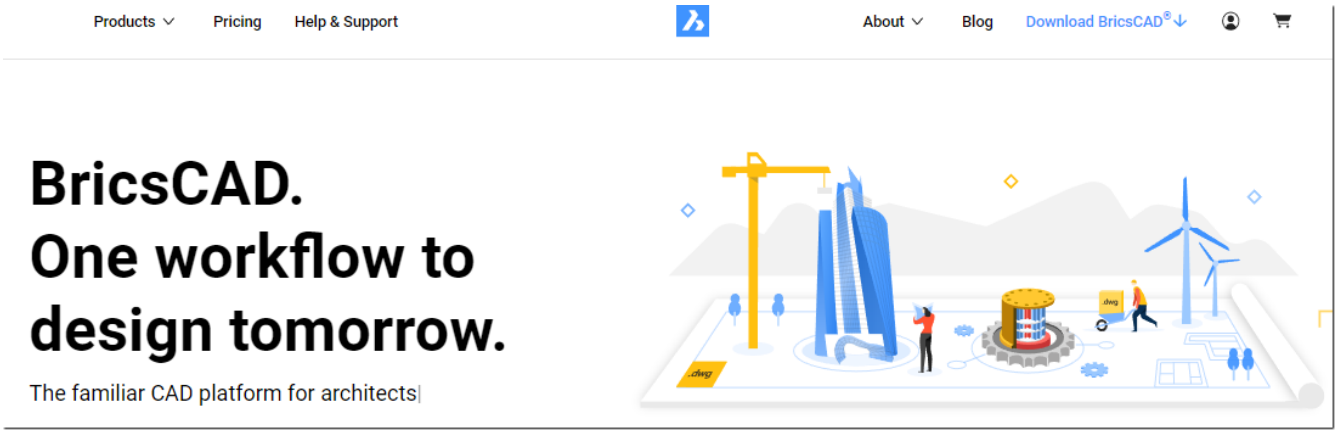
6.23.6 OnWeb

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Bricsys home page in your computer's default Web browser.

Accessing the Command

command bar: **onweb** : **onweb** Displays the Bricsys home page in a Web browser: (\vref {fig:C:/Dev/Task-2448/images/staticbr eg.png})

Figure 6.1528



Command Options

No options.

Related Commands

Browser (6.6.19) - opens the default Web browser. Url (6.30.11) - opens the default Web browser. Mail (6.21.1) - attaches the current drawing to a message in the default email client.

Figure 6.1529



6.23.7 Oops

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Unerases the last erased entity, including those erased by the **Block** command. **Note:** While the U command undoes the last action, this command returns the last entity erased, no matter how many commands earlier it occurred.

Accessing the Command

command bar: **oops** aliases: **undelete, unerase** : **oops** Does not prompt you in the command bar.If the drawing contains no erased entities, BricsCAD reports, "There is nothing to erase."

Command Options

No options.

Related Commands

U (6.30.1) - undoes the last command. Undo (6.30.5) - undoes one or more commands. Redo (6.26.8) - redoes the last undone command. Zoom (6.35.7) - utilizes the Previous option to show the previous view.

Figure 6.1530



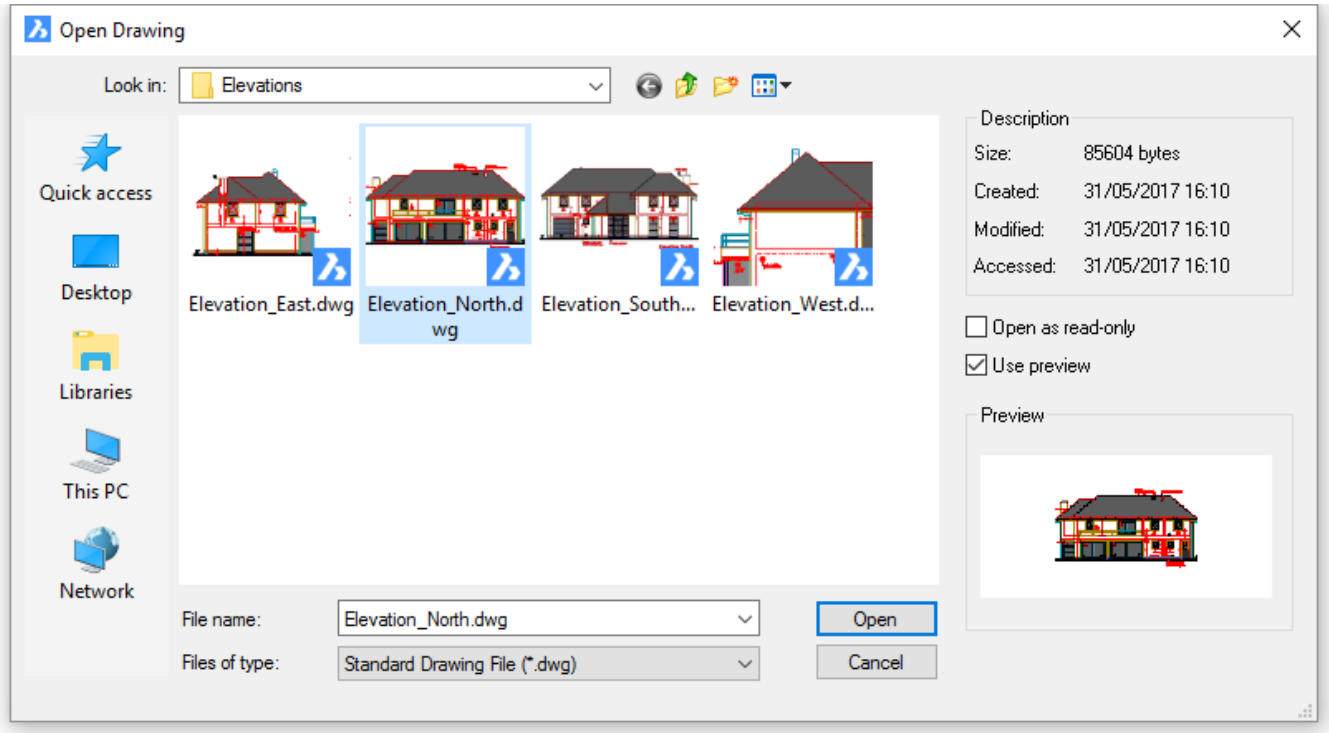
6.23.8 Open

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Opens drawing (DWG), template (DWT), and interchange (DXF) files in new document tabs. **Note:** To open a drawing in an existing drawing, use the Import (6.17.10) orInsert (6.17.3) commands.

Accessing the Command


command bar: **open** shortcut: **Ctrl+O** aliases: **op, im, import** menu bar: **File | Open** ribbon:**Home | File | Open** toolbar: **Standard |**  : **open** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/C
Choose a file, and then click **Open**.

Figure 6.1531



Command Options

Option	Description
Look in	Selects the drive and folder.
File name	Specifies the name of the file.
Files of type	Selects the type of file: • **DWG** - CAD drawing file • **DWT** - drawing template file • **DXF** - drawing interchange file
Open	Opens the file in a new document tab.
Cancel	Closes the dialog box without opening a file.
Open as read-only	Toggles opening the drawing in read-only mode • **Yes** - opens the file in read-only mode, and displays [Read-Only] on the program's title bar • **No** - opens the drawing normally The drawing cannot be saved to the same file name, so to save changes to the drawing, use the SaveAs (6.27.3) command to save it by a different name.
Use Preview	Toggles the display of the the preview image: • **Yes** - displays a preview image in this dialog box • **No** - does not display the image

Procedures

- Working with document tabs (1.12.2)

Related Commands

Close (6.9.25) - closes the current drawing. Quit (6.25.8) - exits BricsCAD after closing open drawings. New (6.22.4) - starts new drawings. SecurityOptions (6.27.15) - sets a password to protect a drawing WhoHas (6.33.8) - reports the ownership of a drawing file. WorkSets (6.33.14) - creates and loads a set of drawing files. Allows to work with a shared group of drawings.

Figure 6.1532



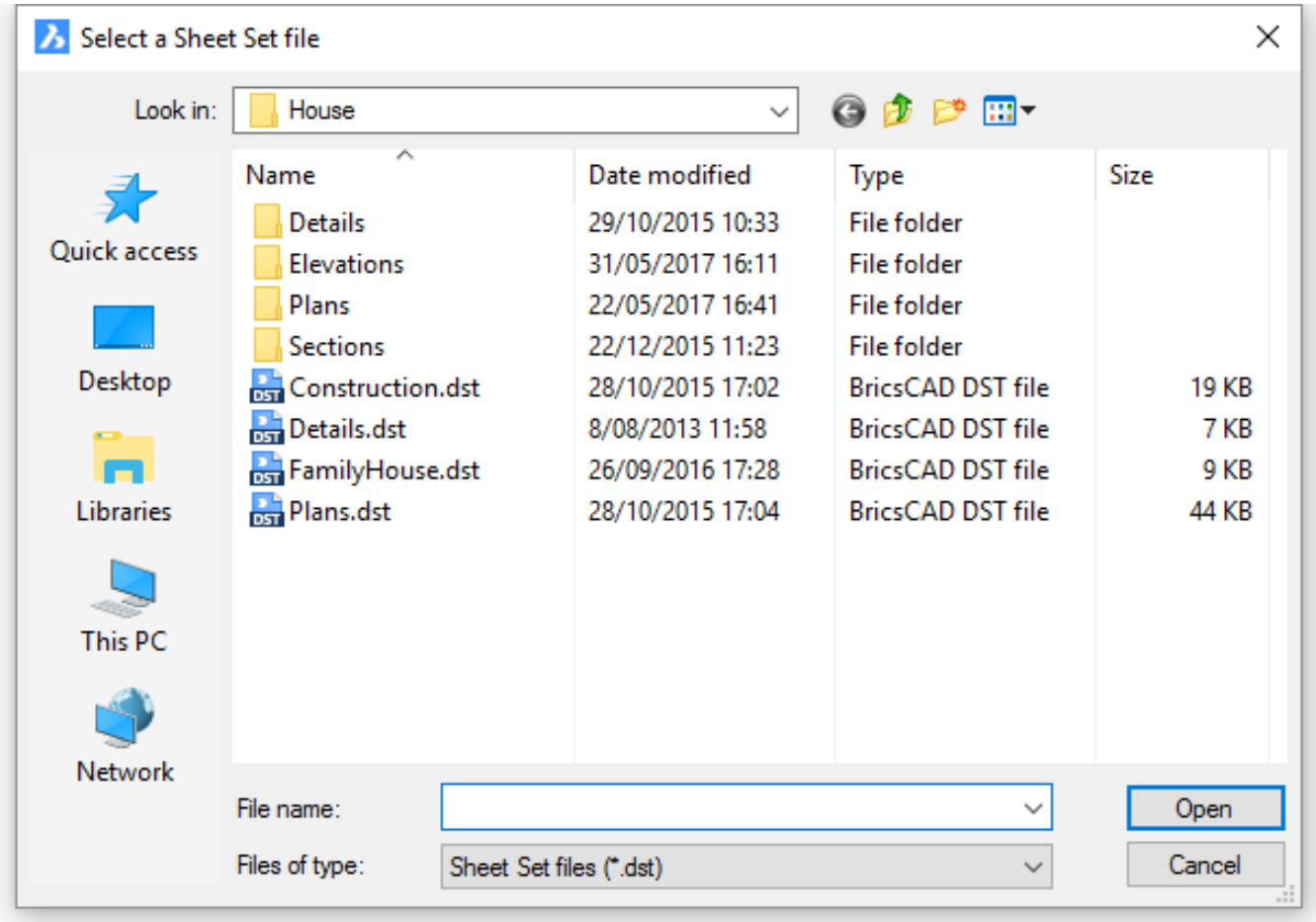
6.23.9 OpenSheetSet

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens an existing sheet set through a file dialog box. **Note:** A sheet set is a collection of drawings used in projects that can be viewed, edited, and published (printed) as a set.

Accessing the Command


command bar: **opensheetset** ribbon:**Home | File | Open Sheetset** toolbar: **Drawing Explorer - Sheets |**  : **opensheetset** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ Select a sheet set file (*.dst), and then click**Open**. The Sheet section of theDrawing Explorer dialog box opens; see the SheetSet (6.27.31) command.

Figure 6.1533



Command Options

No options.

Related Commands

NewSheetSet (6.22.5) - creates a new sheet set. -OpenSheetSet (6.23.10) - opens an existing sheet set; prompts you in the command bar. SheetSet (6.27.31) - manages sheet sets. Opens the **Sheet Sets** panel.

Figure 6.1534



6.23.10 -OpenSheetSet

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens an existing sheet set; prompts you in the command bar. **Note:** A sheet set is a collection of drawings used in projects that can be viewed, edited, and published (printed) as a set.

Accessing the Command

command bar: **-opensheetset** transparent:**-opensheetset** : **-opensheetset Prompts you in the command bar:** Enter sheet set file name: **(Type the path and file name of the .dst sheetset file)** The Sheets section of theDrawing Explorerdialog box displays; see the SheetSet (6.27.31) command.

Command Options

Option	Description
Enter sheet set file name	Type the path and file name of the .dst sheetset file, such as **C:\dwg\architectural\project.dst**

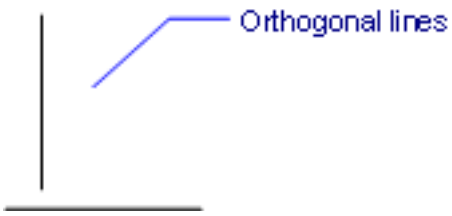
Related Commands

OpenSheetSet (6.23.9) - opens an existing sheet set; displays a file dialog box. NewSheetSet (6.22.5) - creates a new sheet set. SheetSet (6.27.31) - manages sheet sets. Opens the **Sheet Sets** panel.

Figure 6.1535



Figure 6.1536



6.23.11 Orthogonal

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Toggles orthographic (ortho) mode, which constrains drawing and editing to 90-degree angles. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image339.png})

Accessing the Command

command bar: **orthogonal** transparent: **'orthogonal** shortcut: **F8 Ctrl+L** keyboard shortcut: hold down **Shift** key while moving the cursor during editing and drawing aliases: **or ortho** menu bar: **Settings | Draw Orthogonal** ribbon:**Settings | Draw Orthongal** toolbar: **Settings |** status bar: **ORTHO** : **orthogonal Prompts you in the command bar:** ORTHOMODE is currently off: ON/Toggle/<Off>: **(Enter an option)**

Command Options

Option	Description
ON	Turns on orthogonal mode.
Toggle	Toggles orthogonal mode: • if on, turns it off • if off, turns it on
OFF	Turns off orthogonal mode.

Status Bar Menu

Right-click ORTHO on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image791.png})

Shortcut Menu	Description
On	Turns on orthographic mode.
Off	Turns off orthographic mode.
Settings	Opens the Coordinate Input section of the Settings dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Orthogonal mode	Toggles orthogonal mode: • **On** - turns on ortho mode • **Off** - turns off ortho mode

Figure 6.1537

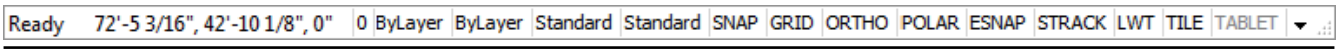
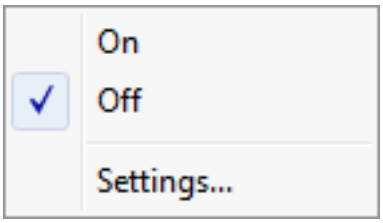


Figure 6.1538



Related Commands

Snap (6.27.38) - sets the ortho angle. Settings (6.27.23) - sets the parameters for ortho mode.

Figure 6.1539

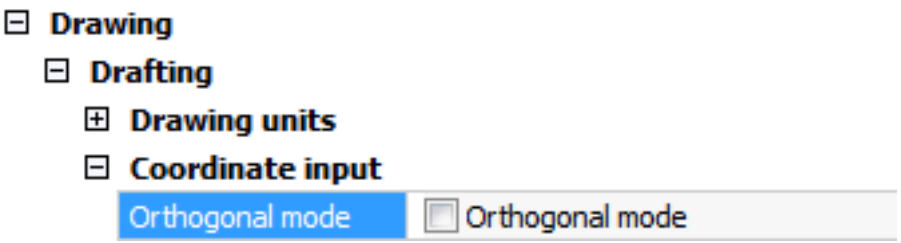


Figure 6.1540



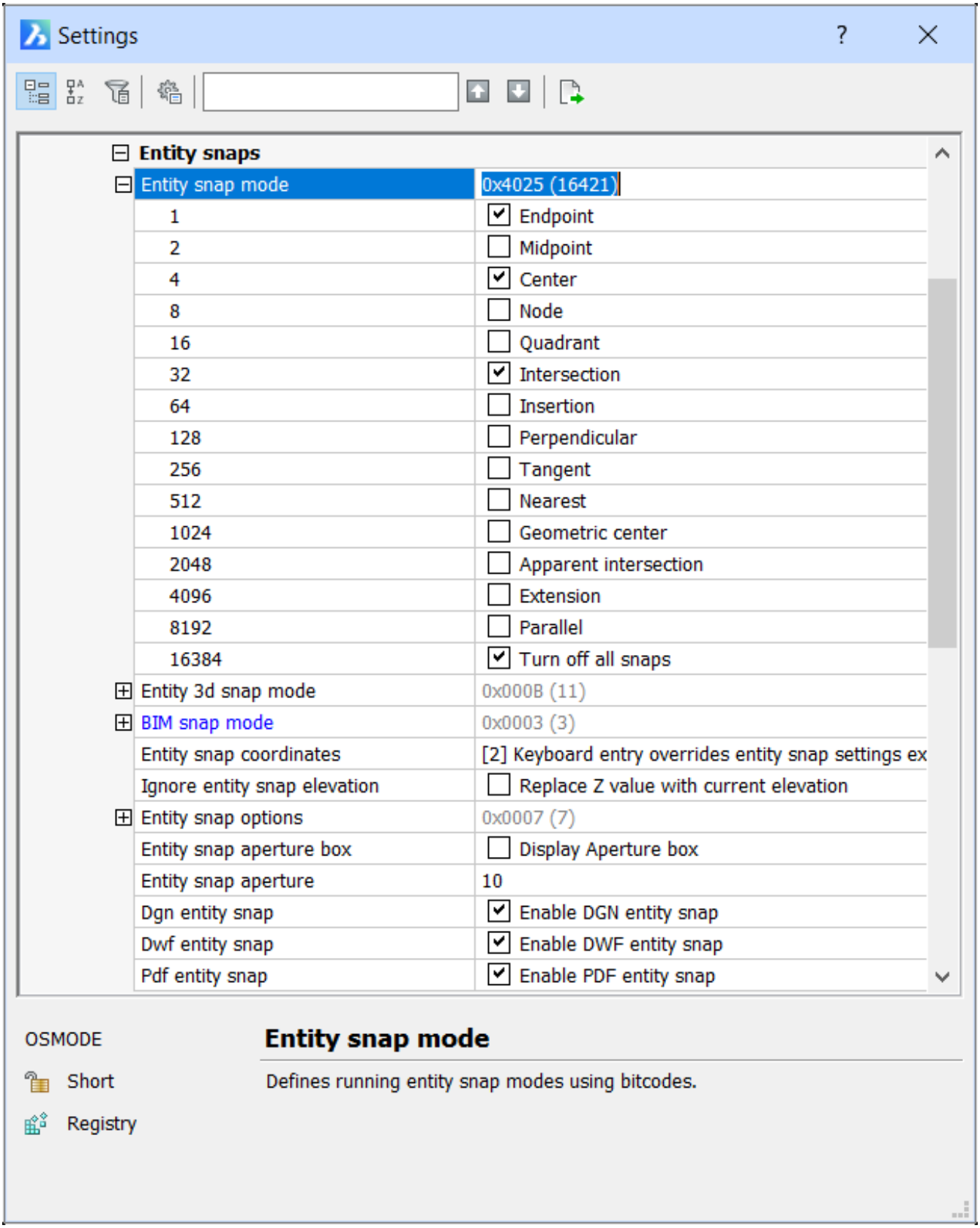
6.23.12 OSnap

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets entity snaps through the Settings dialog box (short for "object snap"). **Note:** Entity snap help you draw and edit precisely by snapping the cursor to the nearest geometric feature, as listed below. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image313.png})

Accessing the Command

command bar: **osnap** transparent: **'osnap** shortcut: **F3** alias: **ddosnap ddesnap os setesnap** menu bar: **Settings | Entity Snap Settings** ribbon:**Settings | Entity Snap Settings** toolbar: **Settings |** status bar: **ESNAP** : **osnap** Displays the Entity Snaps section of the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V dlg.png}) Change a setting, and then click **X**.

Figure 6.1541



Command Options

Option	Description
1 - Endpoint	Toggles endpoint entity snap mode. Snaps to the end points of open entities, such as lines,arcs, open polylines,and open splines. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Endpoint (6.13.7) command.
2 - Midpoint	Toggles midpoint entity snap mode. Snaps to the midpoint of open entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Midpoint (6.21.16) command.
4 - Center	Toggles center entity snap mode. Snaps to the center of circular entities, such as circles, arcs, and polyarcs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Center (6.9.3) command.
8 - Node (Point)	Toggles node entity snap mode. Snaps to point entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Node (6.22.7) command.
16 - Quadrant	Toggles quadrant entity snap mode. Snaps to the quadrants points of circular entities, such as circles, arcs, and polyarcs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Quadrant (6.25.7) command.
32 - Intersection	Toggles intersection entity snap mode. Snaps to the intersection of any pair of entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Intersection (6.17.20) command.
64 - Insertion	Toggles insertion entity snap mode. Snaps to the insertion point of blocks and text. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Insertion (6.17.16) command.
128 - Perpendicular	Toggles perpendicular entity snap mode. Snaps to the perpendicular of one entity to another. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Perpendicular (6.24.22) command.
256 - Tangent	Toggles Tangent entity snap mode. Snaps to the tangencies of circular entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Tangent (6.29.9) command.
512 - Nearest	Toggles nearest entity snap mode. Snaps to the nearest point on any entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Nearest (6.22.2) command.

1024 - Geometric Center	Toggles geometric center entity (GCenter) snap mode. Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids. Can also be toggled with the GCenter (6.15.4) command.
2048 - Apparent Intersection	Toggles Apparent entity snap mode. Snaps to the apparent intersection of entities in 3D space. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Apparent (6.4.11) command.
4096 - Extension	Toggles extension entity snap mode. Snaps to the intersection of open entities, as if they were extended to the point of intersection. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Extension (6.13.20) command.
8192 - Parallel	Toggles parallel entity snap mode. Snaps to the parallel point of entities. Allows to draw lines, polyline segments, xlines and rays parallel to another entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/p Can aso be toggled with the Parallel (6.24.4) command.
16384 - Toggle all snaps	Turns all entity snap modes off. Modes can also be turned off with the None (6.22.8) command.

Status Bar Menu

Right-click ESNAP on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image791.png})

Shortcut Menu	**Description**
On	Turns on entity snaps.
Off	Turns off entity snaps.
Settings	Opens the Entity Snaps section of the Settings dialog box.

Figure 6.1542

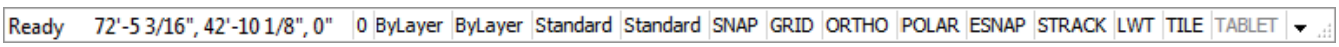
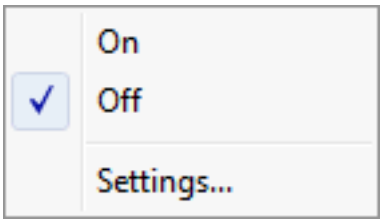


Figure 6.1543



Related Commands

-OSnap (6.23.13) - toggles entity snap modes in the command bar. -3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. 3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.1544

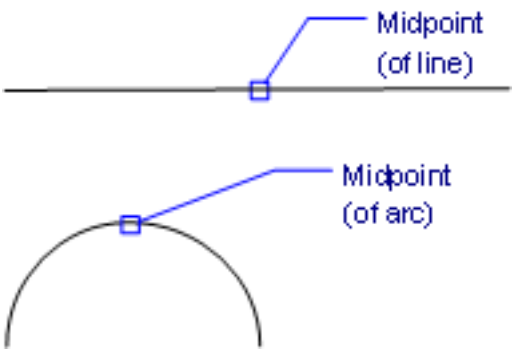


Figure 6.1545

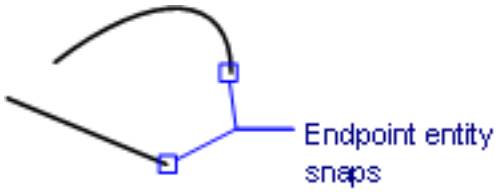


Figure 6.1546

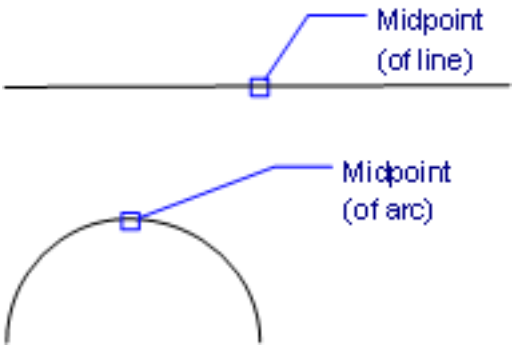


Figure 6.1547

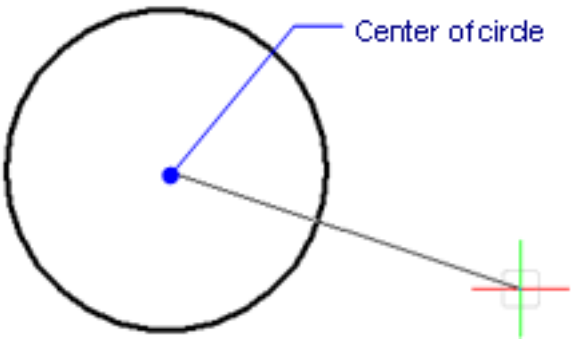


Figure 6.1548



Figure 6.1549

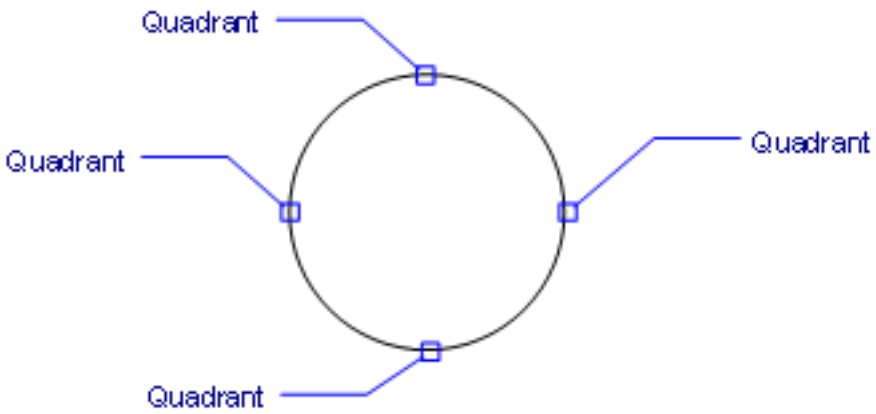


Figure 6.1550

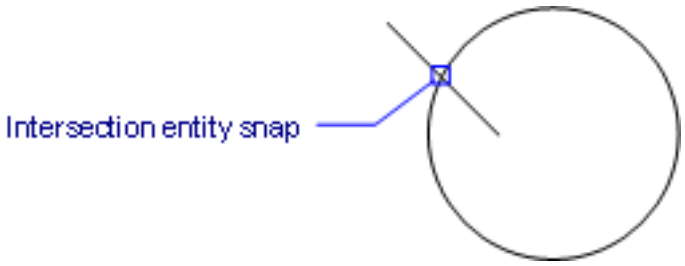


Figure 6.1551



Figure 6.1552

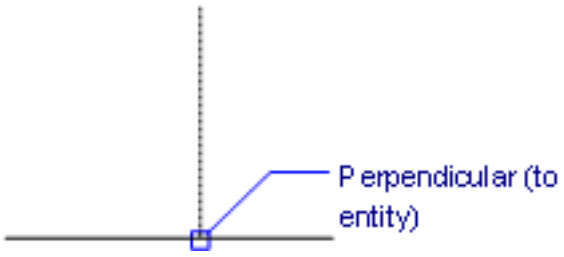


Figure 6.1553

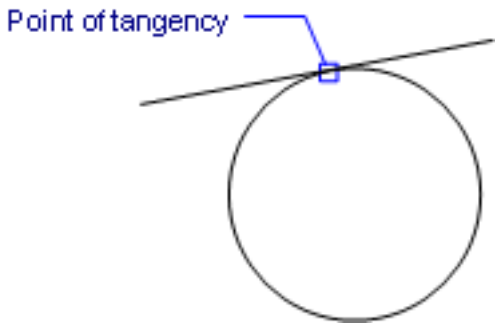
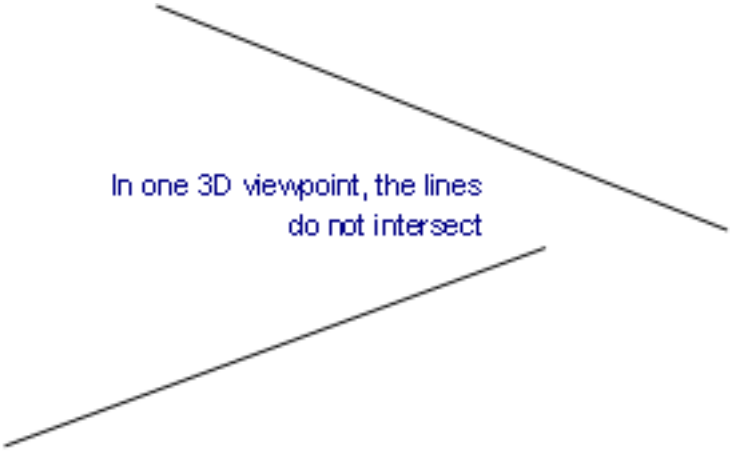


Figure 6.1554



Figure 6.1555



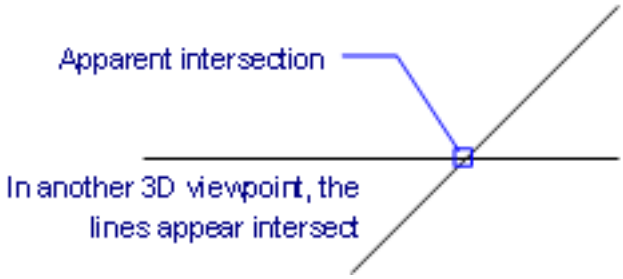


Figure 6.1556

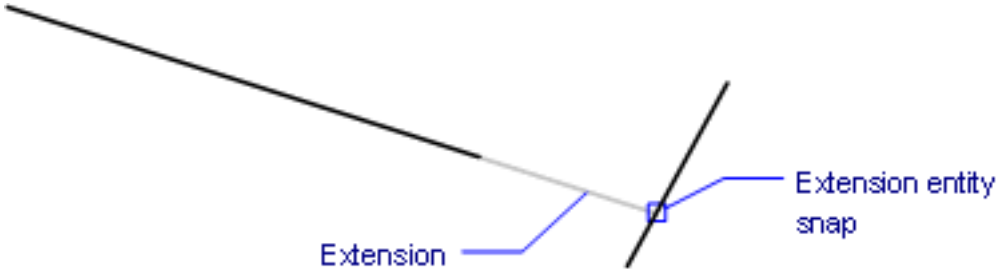


Figure 6.1557

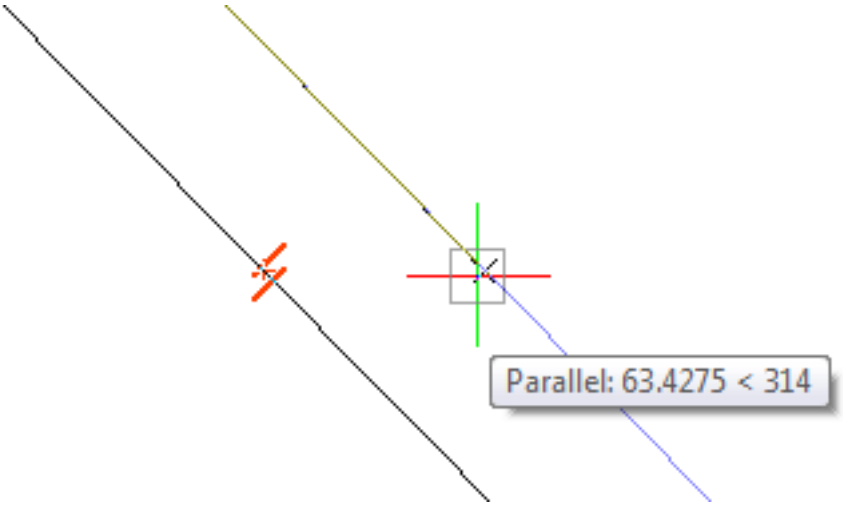


Figure 6.1558



6.23.13 -OSnap

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles entity snap modes at the command line (short for "object snap"). **Note:** Entity snap help you draw and edit precisely by snapping the cursor to the nearest geometric feature, as listed below. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image605.png})

Accessing the Command

command bar: **-osnap** transparent: **'-osnap** shortcut: **F3** aliases: **esnap -os** status bar: **ESNAP** mouse button: **middle** : **-osnap Prompts you in the command bar:** Snaps: NEArest/ENDpoint/MIDpoint/CENter/ GeometricCEnter/ PERpendicular/ TANgent/QUAdrant/INSertion/POInt/INTersection/APParent/ON/<OFF>: **(Enter an option, or press Enter to toggle all entity snaps on or off)** The following entity snaps work with the vertices and edges of 3D solid models:

- Endpoint
- Midpoint
- Nearest

Command Options

Option	Description
NEArest (6.22.2)	Toggles nearest entity snap mode. Snaps to the nearest point on any entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Nearest (6.22.2) command.
ENDpoint (6.13.7)	Toggles endpoint entity snap mode. Snaps to the endpoints of open entities, such as lines, arcs, open polylines, and open splines. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Endpoint (6.13.7) command.
MIDpoint (6.21.16)	Toggles midpoint entity snap mode. Snaps to the midpoint of open entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Midpoint (6.21.16) command.
CENter (6.9.3)	Toggles center entity snap mode. Snaps to the center of circular entities, such as circles, arcs, and polyarcs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Center (6.9.3) command.
GeometricCEnter (6.15.4)	Toggles geometric center entity (GCenter) snap mode. Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids. Can also be toggled with the GCenter (6.15.4) command.
PERpendicular (6.24.22)	Toggles perpendicular entity snap mode. Snaps to the perpendicular of one entity to another. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Perpendicular (6.24.22) command.

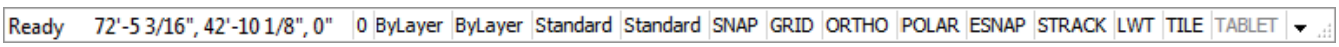
TANgent (6.29.9)	Toggles Tangent entity snap mode. Snaps to the tangencies of circular entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Tangent (6.29.9) command.
QUAdrant (6.25.7)	Toggles quadrant entity snap mode. Snaps to the quadrant points of circular entities, such as circles, arcs, and polyarcs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Quadrant (6.25.7) command.
INSertion (6.17.16)	Toggles insertion entity snap mode. Snaps to the insertion point of blocks and text. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Insertion (6.17.16) command.
POInt (6.24.31) (Node)	Toggles node entity snap mode. Snaps to point entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Node (6.24.31) command.
INTersection (6.17.20)	Toggles intersection entity snap mode. Snaps to the intersection of any pair of entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Intersection (6.17.20) and 3dIntersection (6.3.6) commands.
APParent (6.4.11)	Toggles Apparent entity snap mode. Snaps to the apparent intersection of entities in 3D space. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Apparent (6.4.11) and 2dIntersection (6.2.1) commands.
EXTension (6.13.20)	Toggles extension entity snap mode. Snaps to the intersection of open entities, as if they were extended to the point of intersection. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Can also be toggled with the Extension (6.13.20) command.
PARallel (6.24.4)	Toggles parallel entity snap mode. Snaps to the parallel point of entities. Allows to draw lines, polyline segments, xlines and rays parallel to another entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/p Can aso be toggled with the Parallel (6.24.4) command.
ON	Turns all entity snap modes on.
OFF	Turns all entity snap modes off. Modes can also be turned off with the None (6.22.8) command.

Status Bar Menu

Right-click ESNAP on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/ (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image791.png})

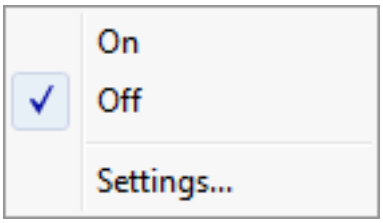
Shortcut Menu	Description

Figure 6.1559



On	Turns on entity snaps.
Off	Turns off entity snaps.
Settings	Opens the Entity Snaps section of the Settings dialog box.

Figure 6.1560



Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. 3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.1561

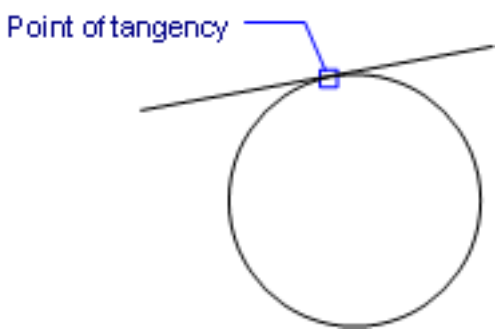


Figure 6.1562



Figure 6.1563

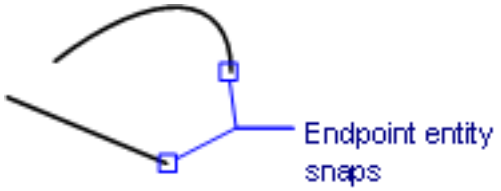


Figure 6.1564

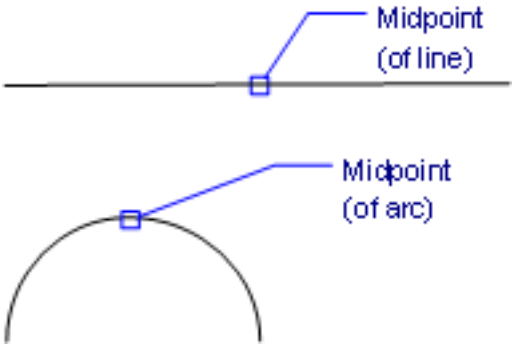


Figure 6.1565

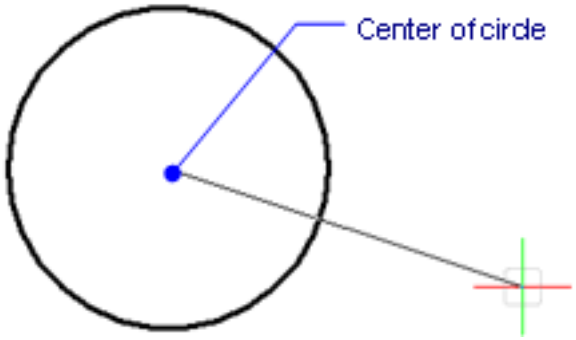


Figure 6.1566

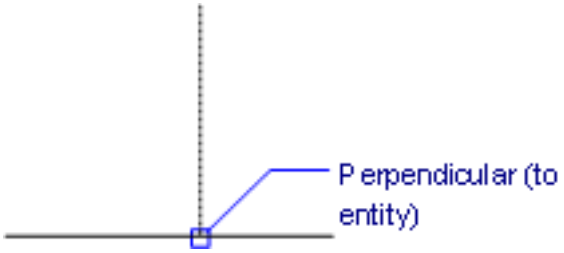


Figure 6.1567

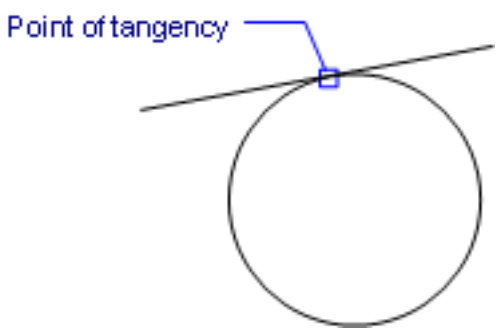


Figure 6.1568

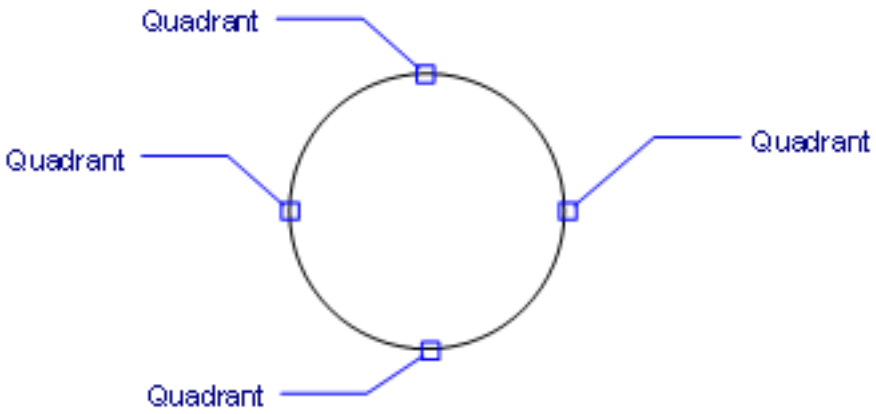


Figure 6.1569



Figure 6.1570



Figure 6.1571

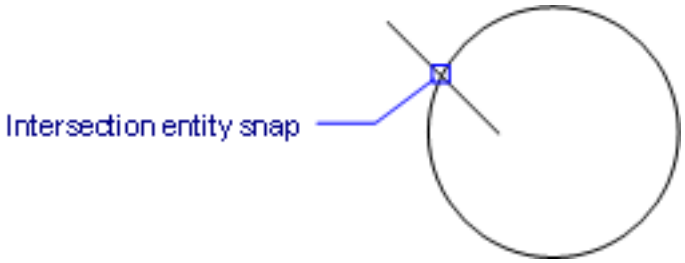


Figure 6.1572

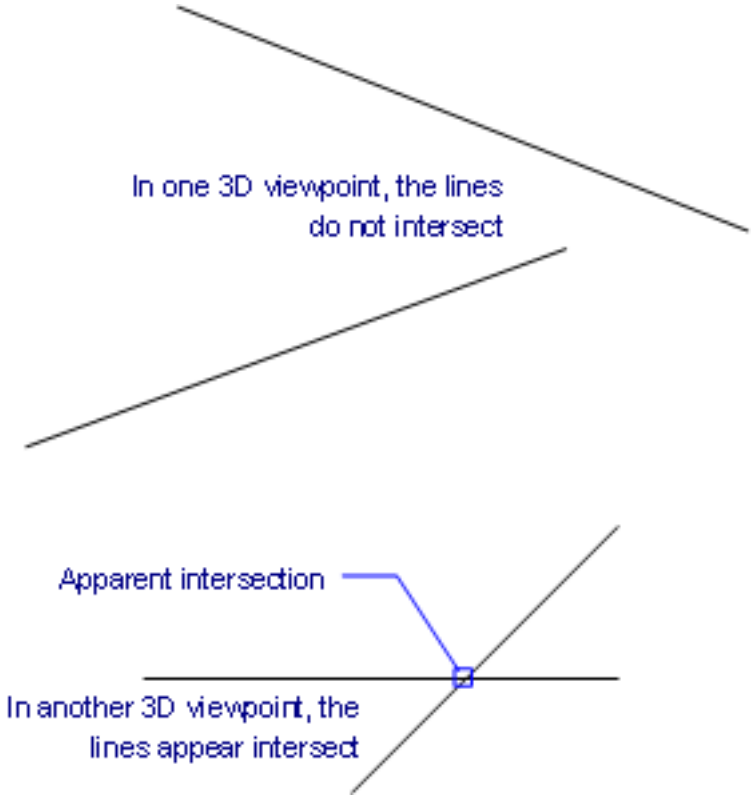


Figure 6.1573

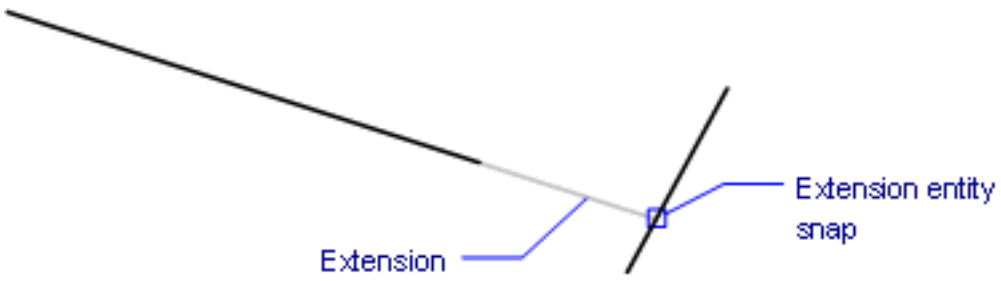


Figure 6.1574

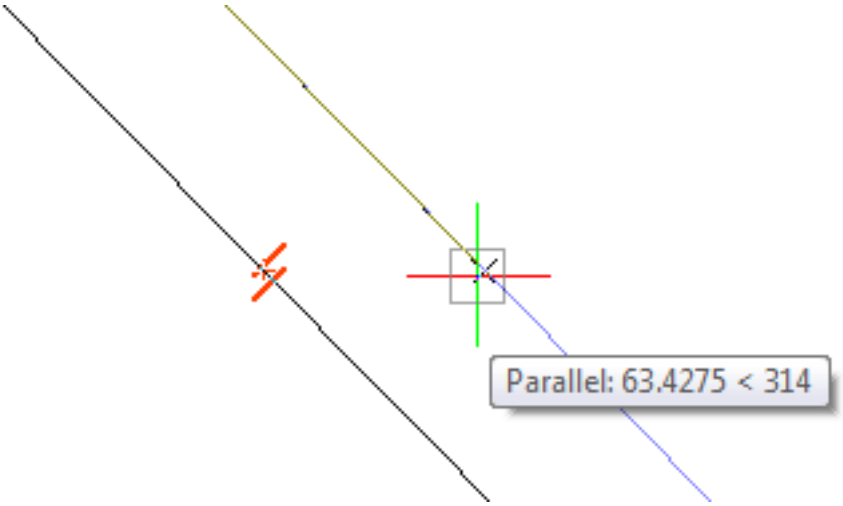


Figure 6.1575



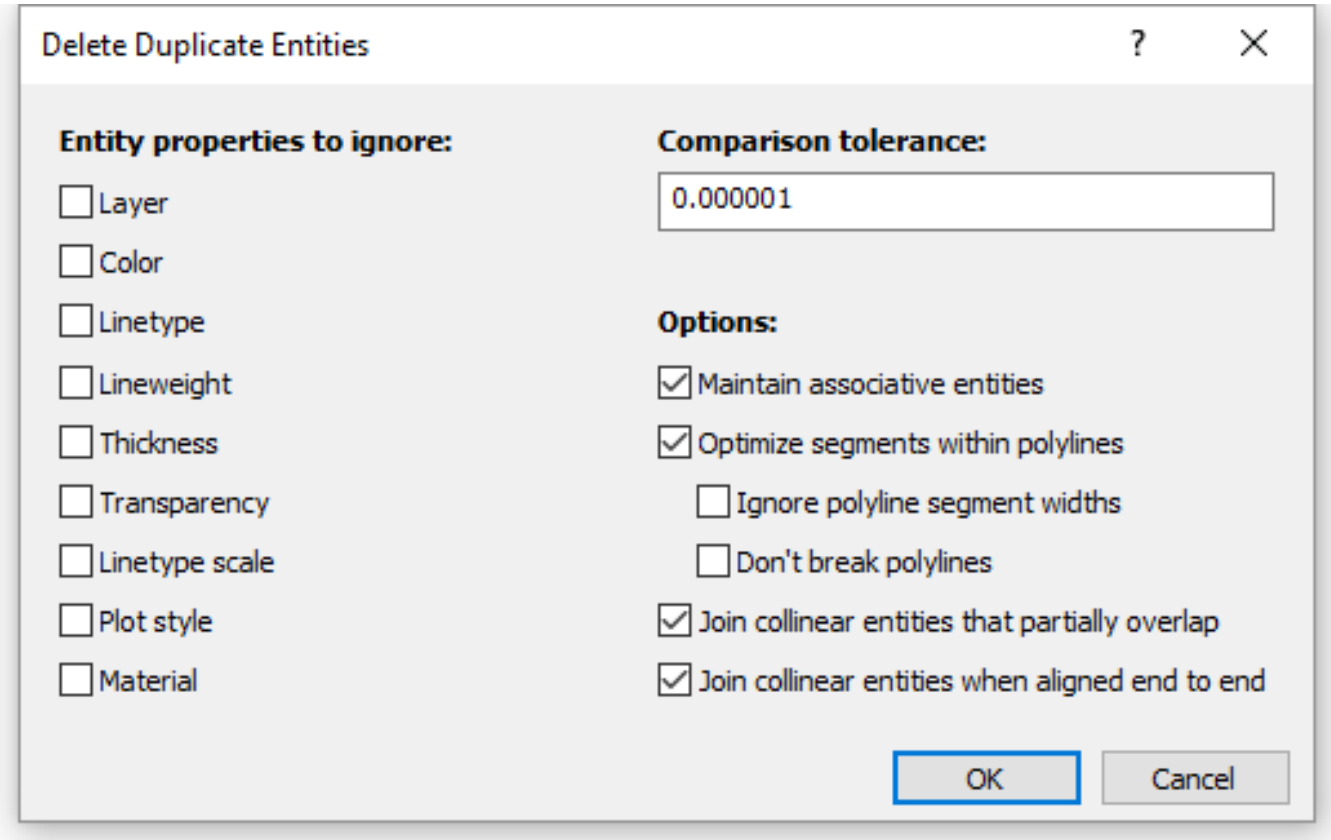
6.23.14 Overkill

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Deletes duplicate entities and overlapping lines, arcs, and polylines, and unifies partly overlapping or contiguous ones; displays a dialog box for choosing options.

Accessing the Command

command bar: **overkill** transparent: **'overkill** menu: **Edit | Delete Duplicate Entities** : **overkill Prompts you in the command bar:** Select entities:**(Select entities to check) Select entities: (Select more entities to check, or press Enterto continue)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou

Figure 6.1576



Command Options

Option	Description
Select entities	Select the entities to check for overlap: • Select one or more entities • Enter **all** to select all entities in the drawing • Press **Enter** to end entity selection The command opens the Delete Duplicate Entities dialog box, in which you specify which ones should be erased.
Entity properties to ignore	Selected properties are ignored when comparing duplicate or overlapping entities.
Comparison tolerance	Sets the tolerance for the comparison process. If set to 0 (zero), entities must match completely before being evaluated in the overkill process.

Options	Determines how special cases are handled: • **Maintain associative entities** - associative entities are not processed. • **Optimize segments within polylines** - segments of selected polylines are evaluated individually, duplicate vertices and segments are removed. When lines or arcs duplicate a polyline segment, one of them is deleted, which might break a polyline, unless the 'Don't break polylines option' is checked. • **Ignore polyline segment widths** - ignores the Width property of overlapping polyline segments • **Don't break polylines** - polyline entities are kept intact • **Join collinear entities that partially overlap** - partially overlapping entities are unified into a single entity • **Join collinear entities when aligned end to end** - collinear lines and polylines having coinciding endpoints are unified into a single entity or segment

Related Commands

-Overkill (6.23.15) - deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones; prompts you in the command bar. Break (6.6.18) - erases a portion of an entity. Erase (6.13.8) - erases entities from the drawing. Join (6.18.1) - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints. Oops (6.23.7) - returns erased entities. Purge (6.24.57) - removes unused named entities from drawings, such as unused layers and linetypes. Trim (6.29.26) - removes portions of entities. U (6.30.1) - unerases erased entities. Undo (6.30.5) - unerases erased entities.

Figure 6.1577



6.23.15 -Overkill

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Deletes duplicate entities and overlapping lines, arcs, and polylines, and unifies partly overlapping or contiguous ones; displays options on the command line.

Accessing the Command

command bar: **-overkill** : **-overkill Prompts you in the command bar:** Select entities: **(Select entities to check) Select entities: (Select more entities to check, or press Enterto continue)** Enter an option to change [Done/Ignore/tOlerance/optiPlines/combine parTial overlap/combine Endtoend/Associativity] <done>: **(Enter an option or press Enter to run the command)**

Command Options

Option	Description
Done	Executes the overkill procedure.
Ignore	Sets the properties to ignore; prompts you: **[None/All/Color/LAyer/Ltype/ltScale/LWeight/Thickness/TRanspa <None>** - **(enter an option)** • **None** - properties are taken into account to determine if overlapping entities should be combined • **All** - all properties are taken into account • **Color**, etc - the colors of overlapping entities are taken into account: if the colors match, then they are combined or deleted
tOlerance	Sets the tolerance for the comparison process; prompts you: **Specify tolerance for entity comparison precision: <0.000001>** - **(specify a number)** When set to 0 (zero), entities must match completely before being evaluated in the overkill process.
optimize Plines	Segments of selected polylines are evaluated individually, duplicate vertices and segments are removed. If lines or arcs duplicate a polyline segment, one of them is deleted, which might break a polyline, unless the **Don't break polylines** option is checked.Prompts you: **Optimize segments within plines [segment wiDth/Break polyline/Yes/No] <Yes>** - **(enter an option):** • **Segment width** - prompts you: **Ignore segment widths [Yes/No] <No>** - **(if Yes , ignores the Width property of polyline segments.)** • **Break polylne** - prompts you: **Break polylines [Yes/No] <Yes>** - **(if No, polyline entities are kept intact)** • **Yes** - optimizes polyline entities • **No** - polyline entities are not optimized

combine parTial overlap	Partially overlapping entities are unified into a single entity; prompts you: **Combine co-linear entities that partially overlap [Yes/No] <Yes>** - **(when yes,partially overlapping entities are unified into a single entity)**
Combine Endtoend	Collinear lines and polyline segments having coinciding endpoints are unified into a single entity or segment; prompts you: **Combine co-linear entities when aligned end to end [Yes/No] <Yes>** - **(when yes,collinear lines and polylines having coinciding endpoints are unified into a single entity or segment)**
Associativity	Associative entities are not processed; prompts you: **Maintain associative entities [Yes/No] <Yes>** - **(when yes, associative entities are not processed and so keep their associativity)**

Related Commands

Overkill (6.23.14) - deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones; displays a dialog box. Break (6.6.18) - erases a portion of an entity. Erase (6.13.8) - erases entities from the drawing. Join (6.18.1) - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints. Oops (6.23.7) - returns erased entities. Purge (6.24.57) - removes unused named entities from drawings, such as unused layers and linetypes. Trim (6.29.26) - removes portions of entities. U (6.30.1) - unerases erased entities. Undo (6.30.5) - unerases erased entities.

Figure 6.1578



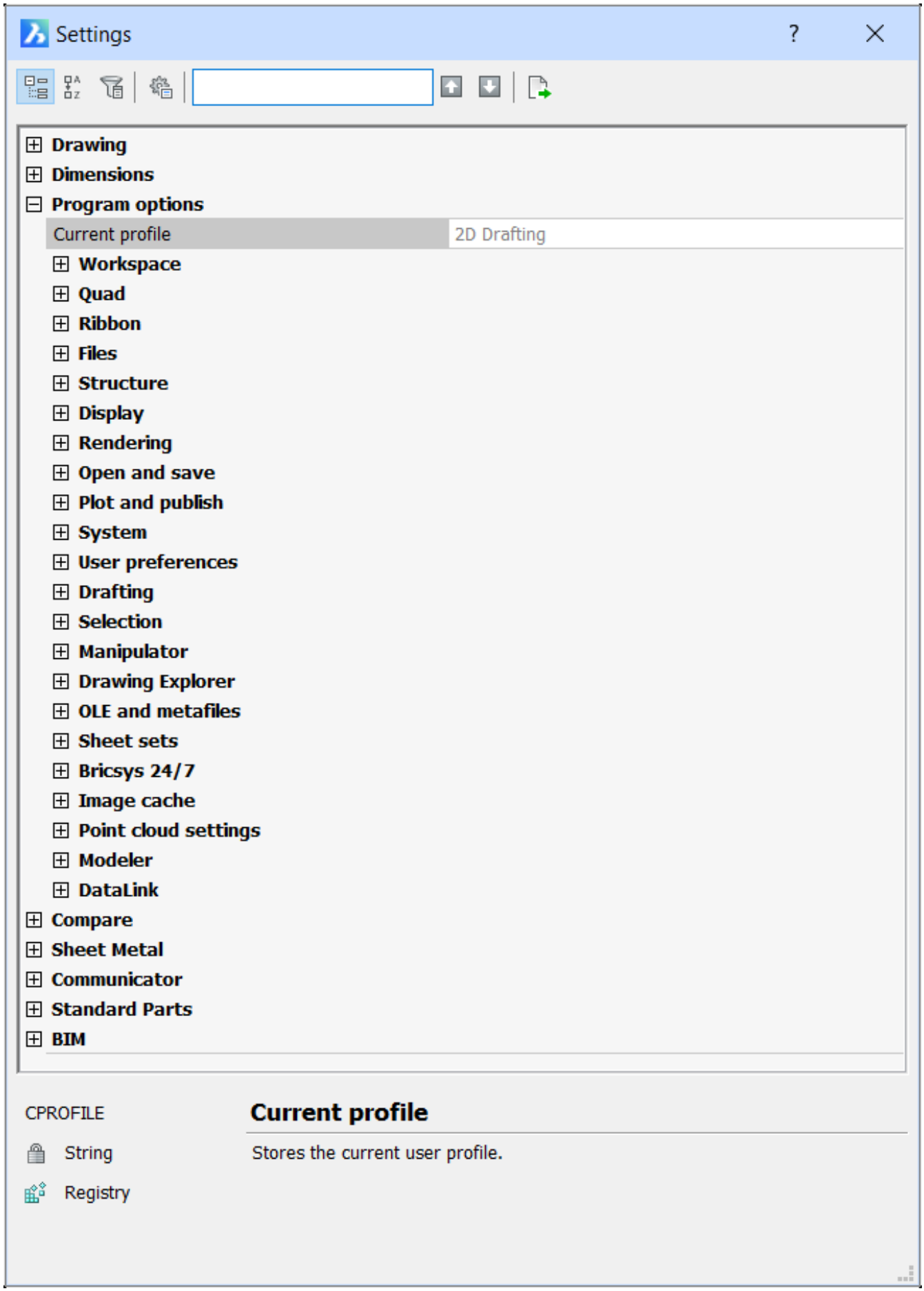
6.23.16 Options

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets program options through the Settings dialog box.

Accessing the Command


command bar: **options** aliases: **cfg, config, preferences, prefs, settings** : **options** Displays the Program Options of the Settings dialog box; see the Settings **(6.27.23)**command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h dlg.png}) Change a setting, and then click **X**.

Figure 6.1579



Command Options

See the Settings (6.27.23)command.

Related Commands

Settings (6.27.23) - changes the values of system variables and user preferences through a dialog box.

6.24 P

6.24.1 PageSetup

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and edits page setups for presetting plotting options in the Drawing Explorer.

Figure 6.1580



Accessing the Command


command bar: **pagesetup** menu bar: **File | Page Setup** toolbar: **Layout |**  : **PageSetup** Displays the Page Setups section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cm
Do one of the following:

- Click the **New** icon () to create a new page setup.
- Double click the icon () of a ***Layout*** or ***Model*** to edit the page setup of a layout or the model space.
- Select one or more layouts () then click the **Delete** button () to remove layouts.
- Double click the icon () of a model space page setup to edit the definition.
- Double click the icon () of a paper space page setup to edit the definition.

When creating a new page setup, choose between **Model space** or **Paper Space**, and then click the **Create**button. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/NewPageSetupDialog.png})
The Page Setup dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/
Yes, this looks very similar to the Print dialog box, but there are some differences. Changes options in it, and then click **OK**. You are returned to the Page Setups dialog box.

Figure 6.1581

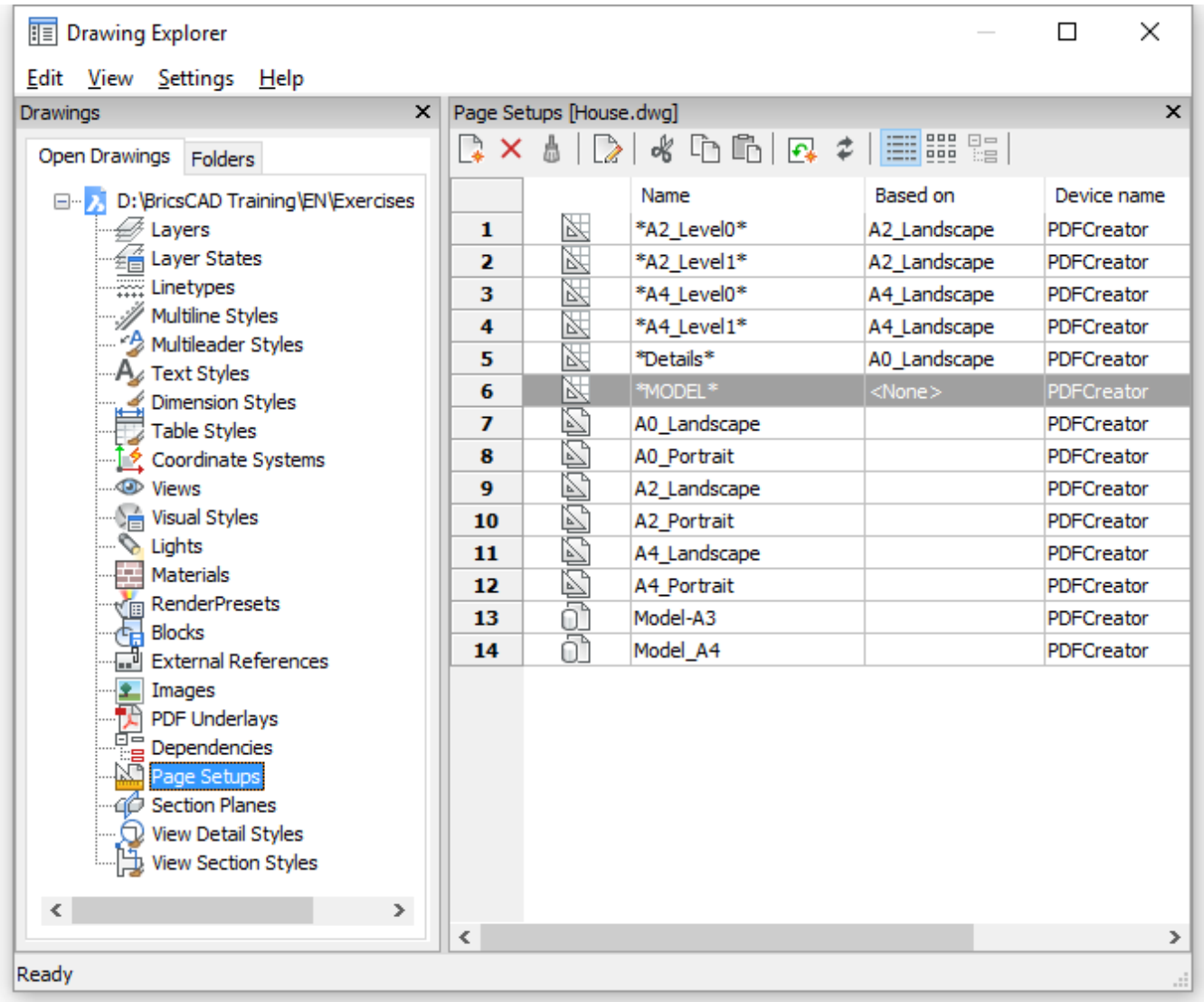


Figure 6.1582

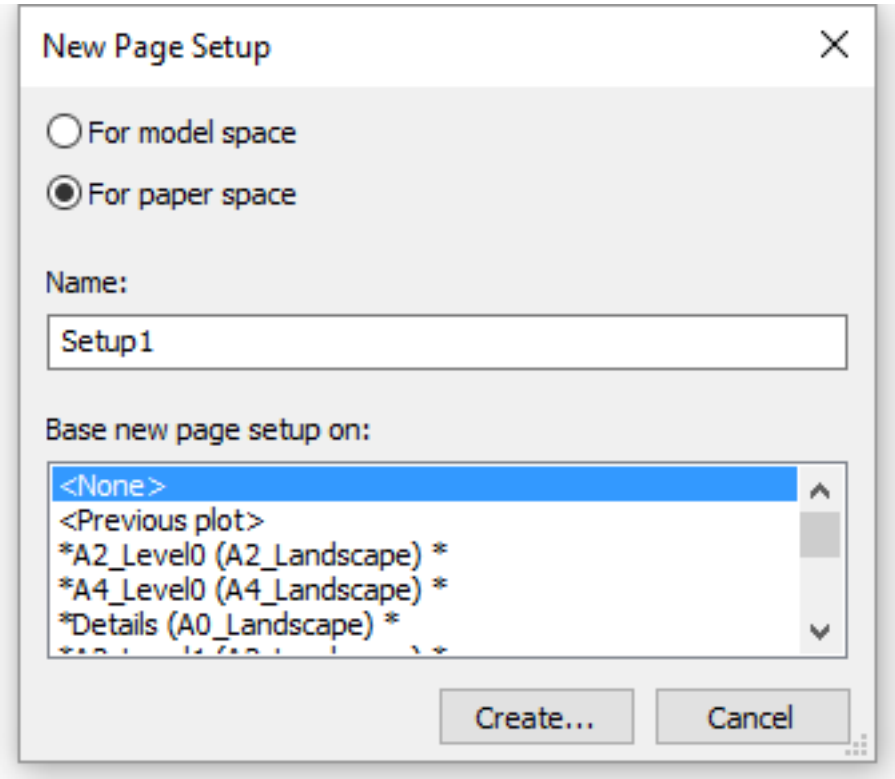
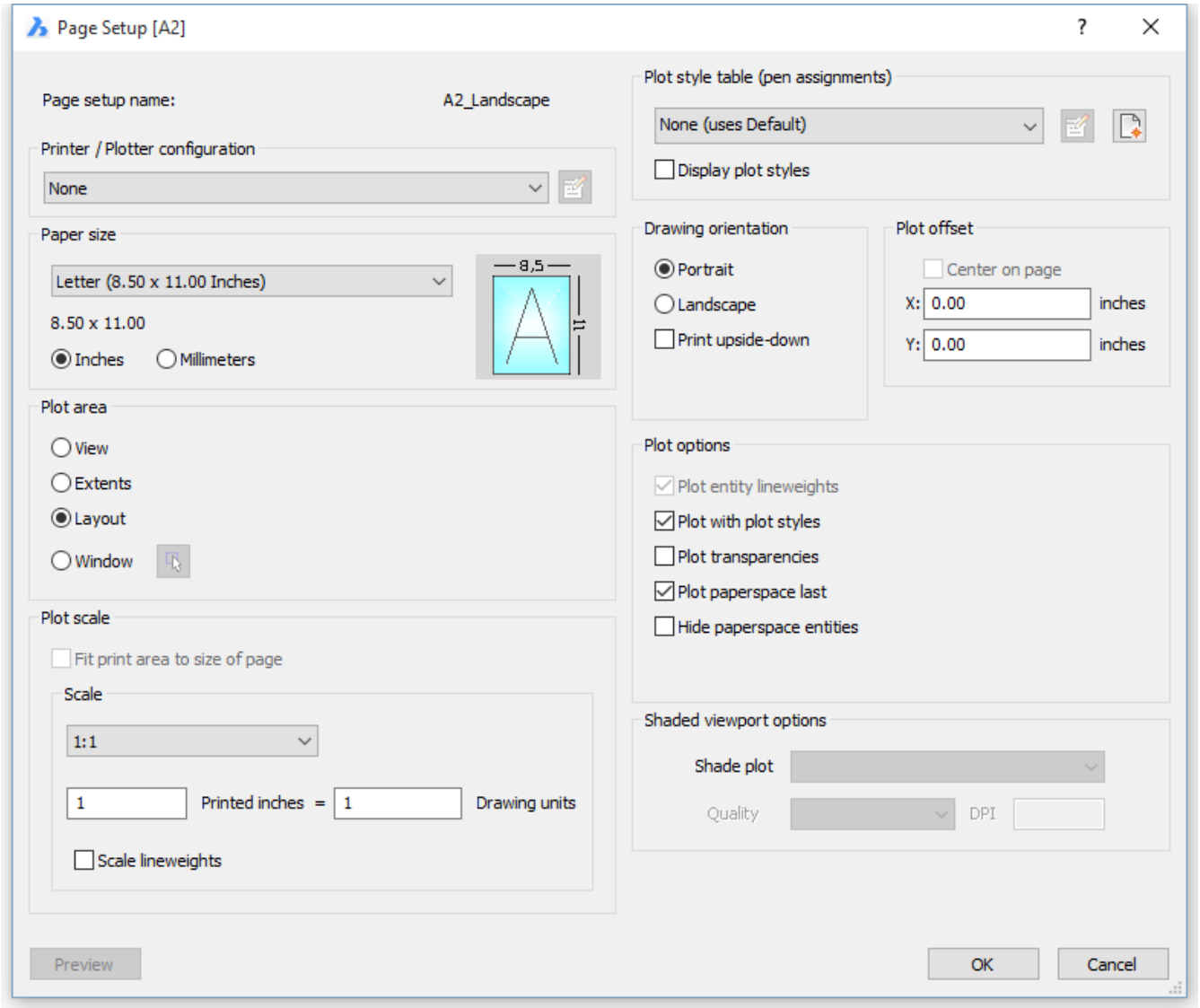


Figure 6.1583

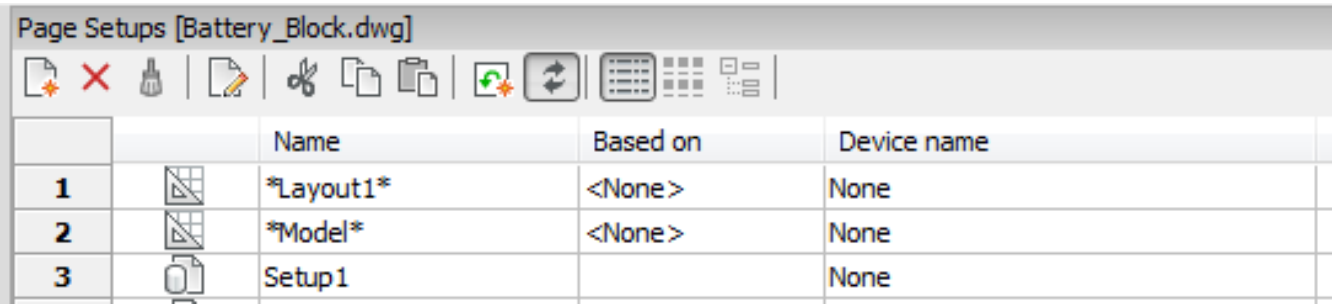


Command Options for Page Setups

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pasgesetup-dlg.png})

Page Setups Options	Description
New	Creates a new page setup; see Command Options for New Page Setup dialog box below
Delete	Removes the page setup from this drawing; displays a warning dialog box when you try to delete page setups with asterisks, such as *Layout1*
Edit	Edit the selected page setup; displays the Page Setup dialog box; see below
Cut / Copy / Paste	Cuts or copies the selected page setup for pasting into another drawing

Figure 6.1584

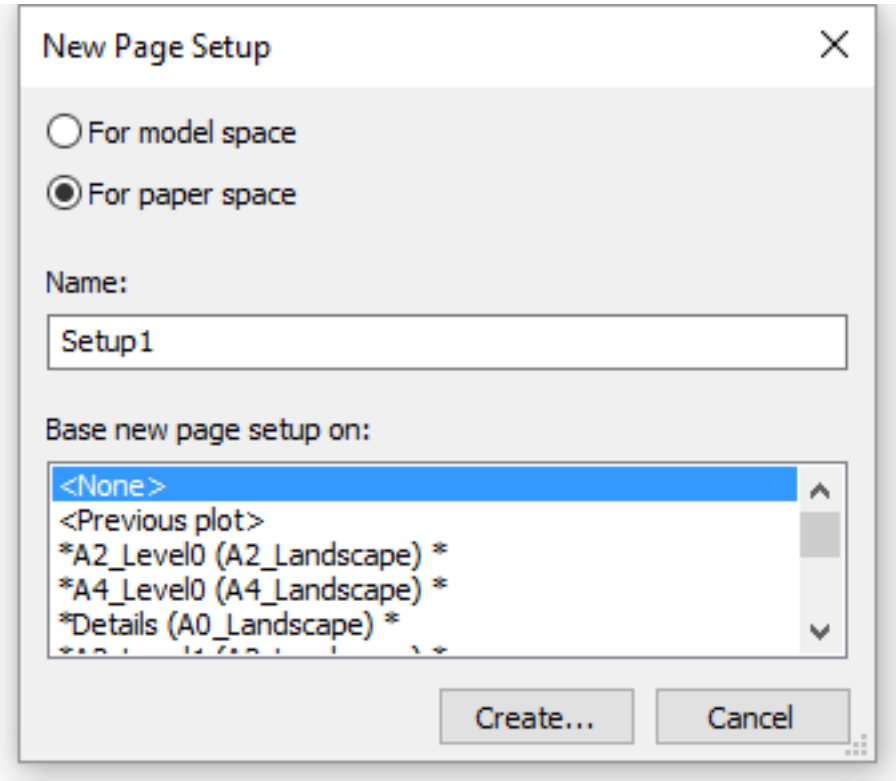


Command Options for New Page Setup Dialog Box

This dialog box appears when you click **New** in Page Setups: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS

New Page Setup Options	Description
For Model Space	Lists model space setups in the list
For Paper Space	Lists paper space setups in the list
Name	Specifies the name for this new setup
Base New Page Setup On	Lists available page setups in this drawing; choose one: • **<None>** - creates a fresh page setup • **<Previous Plot>** - uses the settings from the last time the Print command was used • **Name** - bases the new setup on the settings in a named setup
Create	Moves on to the Page Setup dialog box; see below.
Cancel	Returns to the Page Setups section of the Drawing Explorer without creating the new setup

Figure 6.1585



Command Options for Page Setup Dialog Box

This dialog box appears when you click **Create**in New Page Setup:

Page Setup Option	**Description**
Printer/Plotter Configuration	Selects the printer or plotter. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/s The program works with any output device installed on the system, including networked printers, print to file, faxes, and PostScript devices -- as well aspredefined printer parameters stored in PC3 files.
Edit plotter configuration	Opens the printer's Properties dialog box for customizing the printer's parameters and creating PC3 files; see thePlotterManager (6.24.30)command.

Paper Size	Specifies the size of paper. You can select standard sizes listed by the droplist; these are sizes supported by the printer. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/s While printer may appear to support many different sizes, you should select only the size of paper that is actually in the printer. IfUser Definedappears in the list, the paper's dimensions are specified by the printer's Properties dialog box. The image indicates the size, position and orientation of the current plot area on the selected paper size. • **Inches** - uses imperial units for measurements involving printing • **Millimeters** -uses metric units for measurements involving printing **Manage Custom Sizes** -(PDF output only.) Manages custom paper sizes whenPrint as PDF.pc3is selected in thePrinter/Plotter Configurationlist; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C
Plot Area	Specifies which area of the drawing to print: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/s • **View**- prints the current view or a named view. Choose a view from the droplist; use the View (6.31.1) command to create named views. • **Extents**- prints the extents of the drawing, which ensures every visible entity is printed. Entities on frozen layers are not taken into account when calculating the extents. • **Limits**- prints the limits of the drawing, as specified by the Limits command. • **Window**- prints a rectangular area of the drawing. You define the rectangular area by entering x,y coordinates or by clicking the**Select Area to be Printed**  button: 1. Click the button; you are prompted: 2. **Select first corner of window- (specify one corner of the rectangle by picking points orentering x,y coordinates.)** 3. **Select second corner of window- (specify the other corner.)** The Print dialog box returns, with the pair of x,y coordinates added to the Windows Plot Area boxes; you can edit the values: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/v

Plot Scale

Scales the drawing to fit it to the paper: **Fit print area to the size of paper** -scale is calculated automatically by the program, taking into account the print area of the drawing. When this option is turned on, you cannot specify the scale factor.:

- **Printable area** is the size of the paper, minus the margins.
- **Margins** are the strips along the four edges that the printer uses for handling the paper.

Scale - allows you to specify the scale factor to use for plotting; choose a scale factor from the droplist, or choose **Custom**. The scale factor shown can be edited withtheScaleListEdit (6.27.7)command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/s

- For scale factors like 1:5, the drawing is printed smaller.
- For scale factors like 5:1, the drawing is printed larger.

Custom - lets you enter in your own scale factors: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P **Scale lineweights** - when on,lineweights are scaled with respect to the plot scale.

Plot Style Table (Pen Assignment)

Specifies the plot style table to use for the printed output, which assigns properties to ”pens,” colors, and entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/s

- When the drawing does not use plot styles, then only CTB (color-based table) files are listed.
- When the drawing uses plot styles, then only STB (style-based table) files are listed.

Display plot styles?- **(enter an option)**:

- **Yes**- displays plot styles at plot time
- **No**- does not.

 Edit Plot Style

Opens the Plot Style Table editor dialog box; see thePlotStyle (6.24.29)command.This button is available only when the drawing uses STB plot styles.

 Create New Plot Style

Creates new plot styles; runs the Add Plot Style Table wizard; see theStylesManager (6.27.55)command.

- When the drawing does not use named plot styles, then the wizard creates new color-dependent tables (CTB files).
- If the drawing uses named plot styles, then the wizard creates new named plot style tables (STB files).

Drawing Orientation	Specifies the orientation of the drawing on rectangular paper: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/s • **Portrait**- The drawing or layout x-axis is aligned with the shortest edge of the selected paper size. • **Landscape**- The drawing or layout x-axis is aligned with the longest edge of the selected paper size. • **Print upside down**- Prints the drawing upside-down. This is useful when paper with a drawing border is loaded backwards in the printer.
Plot Offset	Specifies the offset distance for the print. **Center On Page** - centers the print on the page, taking into account margins.Together with**Fit Print Area to Size of Page**, this option is excellent for prints where scale is unimportant. **X** and **Y** - **(enter positive or negative distances to move the drawing in the x and/or y distances)**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/c • **Positive values** - moves the drawing up and to the right. • **Negative values** - moves the drawing down and to the left. The lower left corner of the drawing is moved by the specified distance. This is useful when the paper has a title block area that might interfere with the drawing.

Plot Options	**Plot entity lineweights** - toggles the use the lineweights; this option is unavailable whenPlot with Plot Stylesis turned on\: • **On**- mimics lineweights assigned to the drawing in the print. • **Off**- ignores lineweights. **Plot with Plot Styles** - toggles the use of plot styles • **On**- uses plot styles to determine the look of the printed drawing, andoverrides lineweight settings. • **Off**- does not use plot styles. **Plot Transparencies** -plots entities who have been assigned the theTransparency (1.17.8)property, either by-entity or by layer. **Plot Paperspace Last** - specifies theprinting order;this option is available only when printing layouts, and so is unavailable in model tab: • **On**- prints model space entities, followed by paper space entities. • **Off**- prints paper space entities first, followed by entities in model space. **Hide Paperspace Entities** - when on, removes hidden lines from 3D entities in paper space.This option is disabled when printing model space.
Shaded Viewport Options	Override the visual style of the current view when printing model space: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S **Shade Plot** - **(choose a shade style plot the drawing with.)** This option is disabled when printing a paper space layout.TheShade Plotmode of a paper space viewport is defined in theShade Plotproperty of the viewport.The quality of theRenderedoption is defined through the current render preset; see theRenderPresets (6.26.24)command.
OK	Saves the changes, and then returns to the Page Setup dialog box.
Cancel	Returns to the Page Setup dialog box without printing the drawing.

Related Commands

CPageSetup (6.9.55) - edits the page setup of the current layout or model space in a dialog box. ConvertCtb (6.9.42) - converts CTB plot style files to STB files. ConvertPStyles (6.9.46) - converts drawings from CTB to STB plot styles. ExportPdf (6.13.17) - exports layouts in a drawing to a pdf docucment. PsetupIn (6.24.52) - Imports page setup definitions from another drawing through a dialog box. -PsetupIn (6.24.53) - imports page setup definitions from another drawing through the command bar. PlotStyle (6.24.29) - sets the current plot style. Print (6.24.46) - plots drawings with plot styles. Properties (6.24.49) - changes plot styles assigned to entities. Publish (6.24.55) - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file. -Publish (6.24.56)- prints a saved sheet list (= a list of model space or paper space layouts). ScaleListEdit (6.27.7) - edits the scale list of a drawing, used in **Plot Scale** section of the **Print** and **Page Setup** dialogs and the **Standard scale** property of a paper space viewport. Explorer (6.13.14) - Opens the **Drawing Explorer** dialog box.

Figure 6.1586

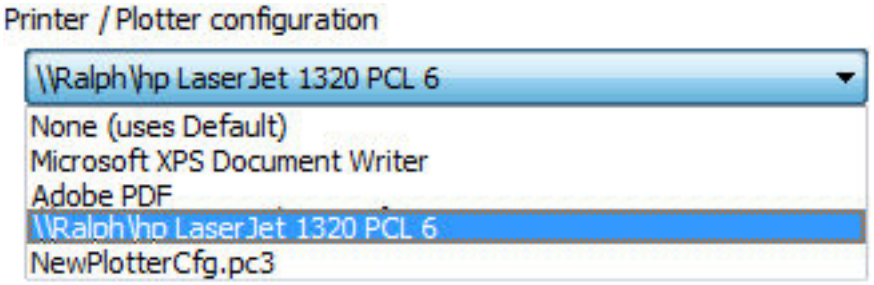


Figure 6.1587

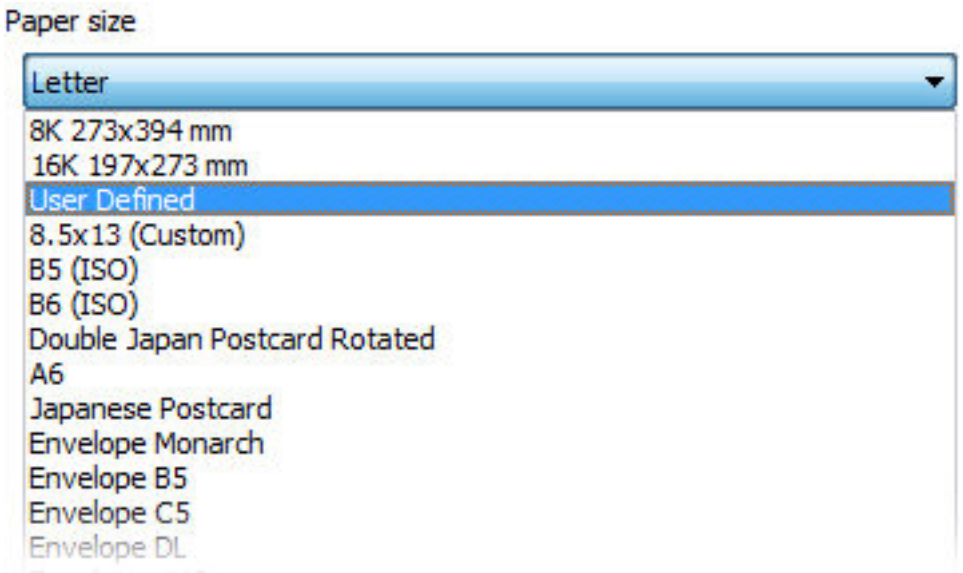


Figure 6.1588

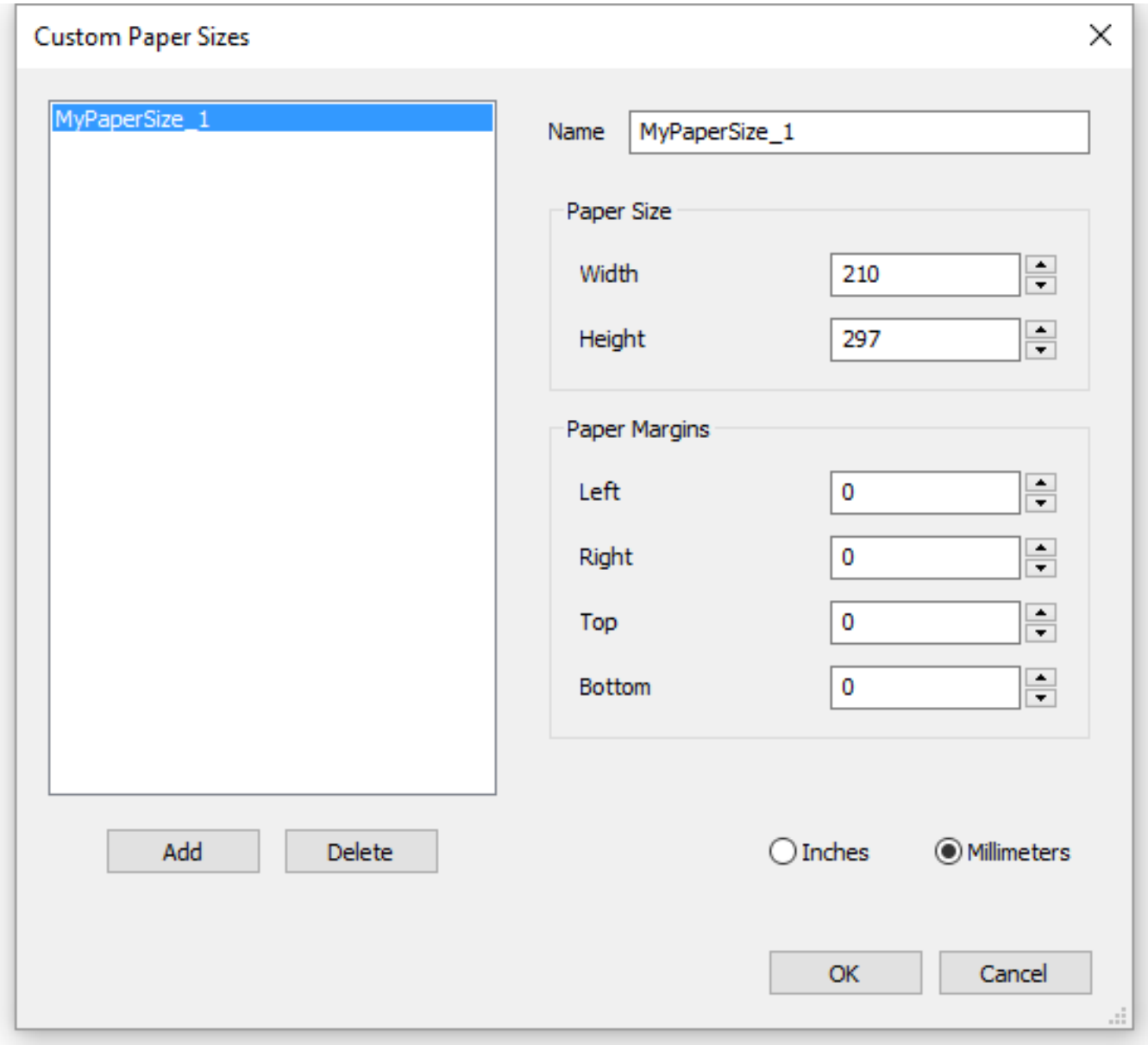


Figure 6.1589

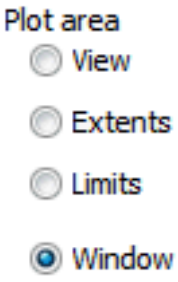


Figure 6.1590

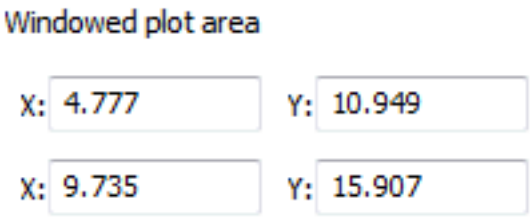


Figure 6.1591

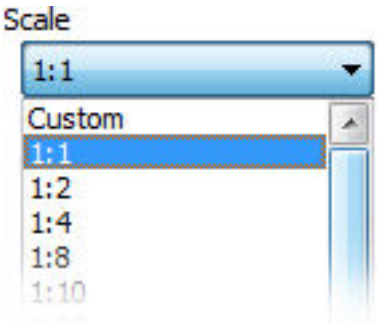


Figure 6.1592

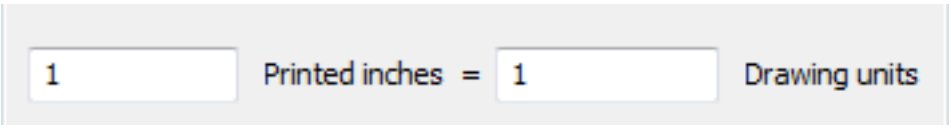


Figure 6.1593

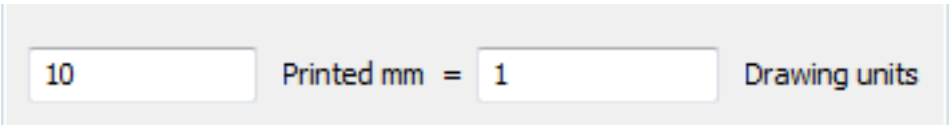


Figure 6.1594

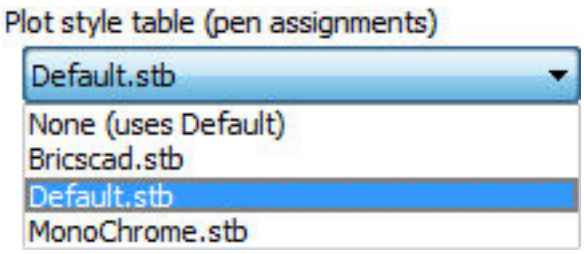


Figure 6.1595

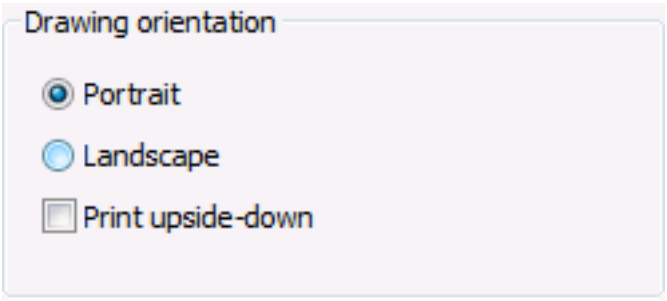


Figure 6.1596

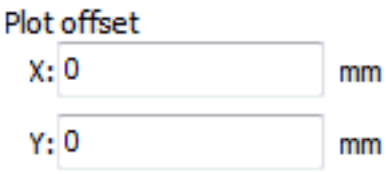


Figure 6.1597

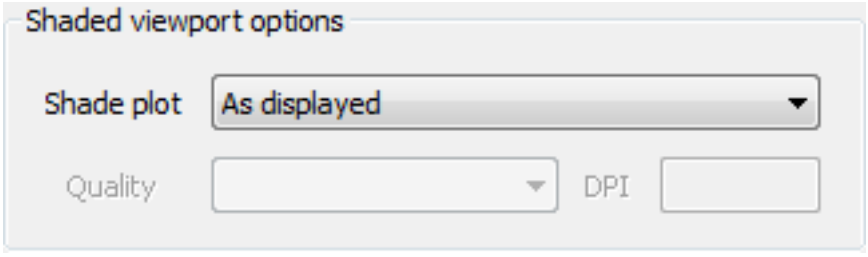


Figure 6.1598



6.24.2 Pan

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Moves the entire drawing in real-tme within the current viewport. **Note:** "Real-time" means that the drawing pans as you move the mouse.

Accessing the Command

 command bar: **pan** transparent: **'pan** alias: **p** menu bar: **View | Zoom| Pan** toolbar: **View | Zoom |**  mouse button: hold down **middle** key and move the mouse : **Pan** >>Press ENTER or Esc to complete, or right-click to display context menu...**(Hold down the left mouse key and then move the mouse to pan the drawing; pressEsc when done)**

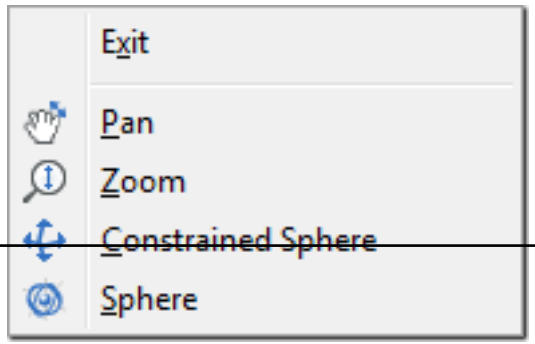
Command Options

Option	Description
Press ENTER or Esc to complete, or right-click to display context menu	Pans the drawing, notice that the cursor changes to a hand icon: 1. Hold down the left mouse key and then move the mouse to pan the drawing 2. Right-click to display a menu that lets you change modes; see below 3. Press**Esc** when done.

Note: You can also pan the drawing with the scroll bars; turn them on with the ScrollBar command. During the Pan command, right-click to see this menu that changes the viewing mode: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/conext-mnu.png})

Menu Option	Description
Exit	Exits the command
Pan	Switches back to panning mode; see the RtPan (6.26.39) command
Zoom	Switches to real-time zoom mode; see the RtZoom (6.26.44) command
Constrained Sphere	Switches to real-time 3D orbiting mode; see the RtRot (6.26.38) command
Sphere	Switches to real-time 3D orbiting mode; see the RtRotf (6.26.48) command

Figure 6.1599



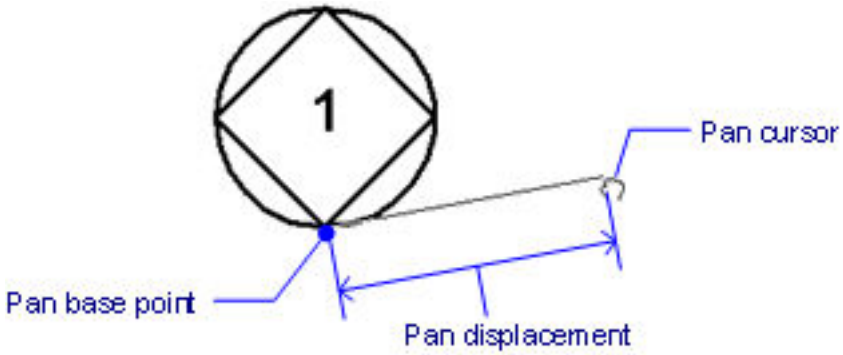
Related Commands

-Pan (6.24.3) - moves the drawing within the current viewport. RtPan (6.26.39) - pan the drawing in realtime. RtRot (6.26.38) - rotates 3D drawings in real-time (short for ”real time rotation”). RtRotf (6.26.48) - rotates 3D drawings in real-time (short for ”real time rotation”). RtZoom (6.26.44) - performs real-time zooming of the drawing. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport.

Figure 6.1600



Figure 6.1601



6.24.3 -Pan

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Moves the entire drawing within the current viewport. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comma

Accessing the Command

command bar: -**pan** transparent: **'-pan** alias: **-p** shortcuts:**PgUp PgDn** : **-Pan Prompts you in the command bar:** Left/Right/Up/Down/PGLeft/PGRight/PGUp/PGDown/<Pan base point>: **(Pick a point, or else enter an option)** Pan displacement point: **(Pick another point)**

Command Options

Option	Description
Pan base point	Specifies the pan distance's starting point.
Pan displacement point	Specifies the ending point.
Left	Pans the drawing 5% to the right.The drawing is panned by 5% of the width of the current viewport.
Right	Pans the drawing 5% to the left.
Up	Pans the drawing 5% downwards.
Down	Pans the drawing 5% upwards.
PaGe Left	Pans the drawing 100% to the right.You can also press the **Shift+Left** cursor key.
PaGe Right	Pans the drawing 100% to the left.You can also press the **Shift+Right** cursor key.
PaGe Up	Pans the drawing 100% upwards.You can also press the **Shift+Up** cursor key.
PaGe Down	Pans the drawing 100% downwards.You can also press the **Shift+Down** cursor key.

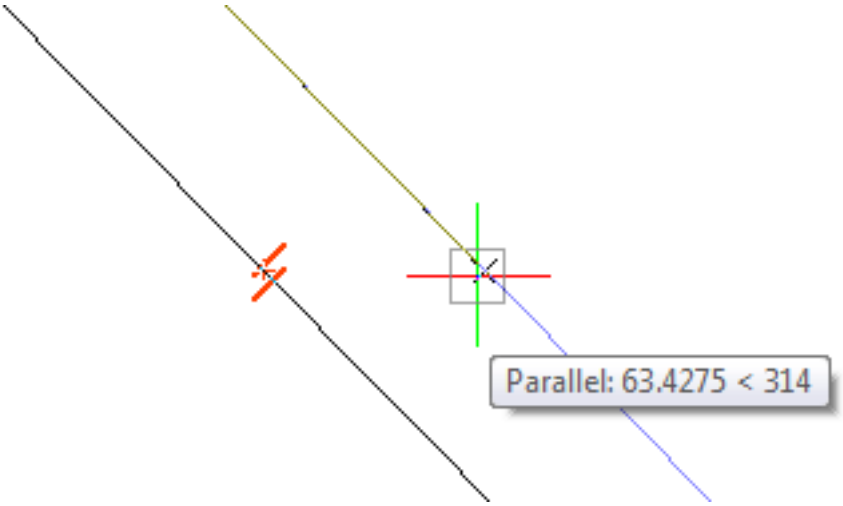
Related Commands

Pan (6.24.2) **-** moves the drawing within the current viewport. RtPan (6.26.39) - pan the drawing in realtime. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44) - performs real-time zooming of the drawing.

Figure 6.1602



Figure 6.1603



6.24.4 Parallel

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles parallel entity snap; draws lines, polyline segments, xlines, and rays parallel to another entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/parallelEntitySnap.pr

Accessing the Command



command bar: **parallel** transparent: **par** toolbar: **Entity Snaps |**  : **Parallel Displays no prompts in the command bar.** When drawing a line, polyline segment, xline or ray:

1. Hover the cursor over the entity tow which you want to draw parallel
2. The parallel snap marker is displayed on the entity, but wait until a small cross indicates the entity is identified
3. The program displays a tracking line when the entity you are drawing is parallel to the selected entity

Command Options

Option	Description
Parallel	Entering this command toggles Parallel entity snap. It draw lines parallel to another entity. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Perpendicular (6.24.22) - toggles perpendicular snap mode Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.1604



6.24.5 -Parameters

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Manages dimensional constraint parameters through the command line. **Note:** Use the ParametersPanelOpen (6.8.30) command to manage parameters through the Parameters panel.

Accessing the Command

command bar: **-parameters** : **-Parameters Prompts you in the command bar:** Enter parameter option [New/Edit/Rename/Dele
<**?**>: **(Enter an option, or else press Enter to display a list of the parameters defined in the current drawing)**

Command Options

Option	Description
New	Creates a user variable; prompts you: 1. **Enter name for new user parameter** - **(type a name for the new user variable. such as PipeLength.)** 2. **Enter expression** - **(enter a valid mathematical expression or a fixed value, such as 20.)**
Edit	Edits the expression for the dimensional constraint or user variable; prompts you: 1. **Enter parameter name** - **(type the name of a dimensional constraint parameter or a user variable,such as PipeRadius.) Old Expression = OutsideDiameter/2.0, Value = 30.15** 2. **Enter expression** - **(type a new mathematical expression or press Enter to confirm the current expression.)**
Rename	Renames a dimensional constraint or user variable. Prompts you: 1. **Enter old parameter name** - **(type the name of an existing dimensional constraint or user variable, such as PipeRadius.) Old Expression = OutsideDiameter/2.0, Value = 30.15** 2. **Enter new parameter name** - **(type a new name for the dimensional constraint or user variable.)**
Delete	Deletes a dimensional constraint or user variable. Prompts you: **Enter parameter name to delete** - **(type the name of an existing dimensional constraint or user variable.)** The parameter is erased with no warning.

?	Lists all dimensional constraints and user variables in the current drawing, showing the name, expression and current value of each dimensional constraint or user variable, such as:

Related Commands

CleanUnusedVariables (6.9.23) - purges variables not used in constraint expressions and not linked to dimensions. -BmParameters (6.8.19) - allows to list and edit the parameters of inserted components. -DesignTableEdit (6.10.27) - allows to edit designtables. Parametrize (6.24.6) - adds constraints and parameters to pure geometric models. DcAligned (6.10.5) - constrains the distance between two points. DcAngular (6.10.6) - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. DcConvert (6.10.7) - converts an associative dimension to a dimensional constraint. DcDiameter (6.10.8) - constrains the diameter of a circle or an arc. DcHorizontal (6.10.10) - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. DcLinear (6.10.39) - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. DcRadius (6.10.11) - constrains the radius of a circle or an arc. DcVertical (6.10.12) - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.

Figure 6.1605



6.24.6 Parametrize

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Adds constraints and parameters to pure geometric models.

Accessing the Command

 Keyboard: **parametrize** Menu: **Parametric | 3D Constraints | Parametrize** Toolbar: **3D Constraints** Quad Tab: **Constraints** Ribbon Tab | Panel: **Parametric | 3D Constraints : Parametrize Prompts you:** Select entities to auto constrain: **(Select entities)** Entities in set: 1 Select entities to auto constrain: **(Select more entities or press Enter to stop.)** Design intent setting "Perpendicular planes" enabled* * See Design Intent Recognition (1.18.3) The parameters and constraints can be edited in the Mechanical Browser (6.8.29). Parameters can be edited in theParameters Panel. (6.24.8)

Command Options

No command options.

Procedures

Parametrize a component automatically (2.4.18) Editing the parameters and constraints in the parameters panel (2.4.18)

Related Commands

MechanicalBrowseOpen (6.8.29) - opens the Mechanical Browser panel. ParametersPanelOpen (6.24.8) - opens the Parameters panel. CleanUnusedVariables (6.9.23) - purges variables not used in constraint expressions and not linked to dimensions. -BmParameters (6.8.19) - allows to list and edit the parameters of inserted components. -DesignTableEdit (6.10.27) - allows to edit designtables. -Parameters (6.24.5) -Manages dimensional constraint parameters in the current drawing. DcAligned (6.10.5) - constrains the distance between two points. DcAngular (6.10.6) - constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. DcConvert (6.10.7) - converts an associative dimension to a dimensional constraint. DcDiameter (6.10.8) - constrains the diameter of a circle or an arc. DcHorizontal (6.10.10) - constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. DcLinear (6.10.39) - constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. DcRadius (6.10.11) - constrains the radius of a circle or an arc. DcVertical (6.10.12) - constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. ConstraintBar (6.9.62) - controls the display of the constraint bar of entities. Delconstraint (6.10.24) - removes all dimensional and geometrical constraints from an entity. Dimconstraint (6.10.29) - applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions. Geomconstraint (6.15.16) - applies geometric relationships between entities and/or valid constraint points on entities.

Figure 6.1606



6.24.7 ParametersPanelClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Closes the Parameters panel.

Accessing the Command

Keyboard:**parameterspanelclose ：ParametersPanelClose Displays no prompts.**

Command Options

No command options.

Related Commands

ParametersPanelOpen (6.24.8) - Opens the Parameters Panel. -ToolPanel (6.29.21) -Allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Figure 6.1607



6.24.8 ParametersPanelOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-platinum-mechanical-bim.png}) Opens the Parameters Panel.

Accessing the Command



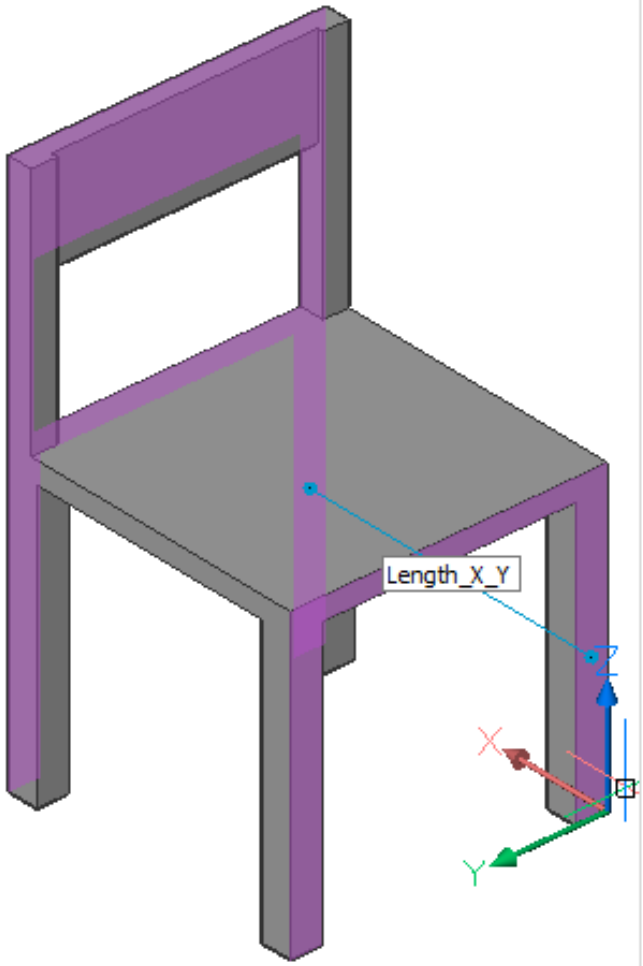
command bar:**parameterspanelopen** menu bar:**Assembly | Parameters Panel** and**Parametric | Parameters Panel** toolbar:**Assembly**and**Parameters** ribbon:**Assembly | Tools** context menu*:**Parameters** * Place the cursor over a toolbar or a ribbon panel, then right click. :**ParametersPanelOpen** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/ParametersPanel.png}) The Parameters Panel allows navigation on the list of parameters, change parameters values and expressions. The context menu allows to animate parameters. When you select a parameter, the controlled geometry highlights in the model. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/ParametersPanelSelect.png})

Figure 6.1608



Name	Expression	Value
CX	Length_X_Y	500
CX_14	p_4	50
CX_15	p_4	50
CX_18	p_4	50
CX_19	p_4	50
CX_21	p_1	10
CX_22	p_1	10
CX_23	p_1	10
CY	Length_X_Y	500
CY_1	p_3	40
CY_12	p_4	50
CY_13	p_4	50
CY_16	p_4	50
CY_17	p_4	50
CY_2	p_3	40
CZ	Length_Z	885
CZ_10	CZ*ratio_1	484.998494833
CZ_11	(CZ-CZ_20)*ratio_2	160
CZ_20	p_2	35
CZ_7	Length_Z	885
Length_X_Y	500	500
Length_Z	885	885
p_1	10	10
p_2	35	35
p_3	40	40
p_4	50	50
ratio_1	0.548020898116	0.54802089811
ratio_2	0.188235294118	0.18823529411

Figure 6.1609



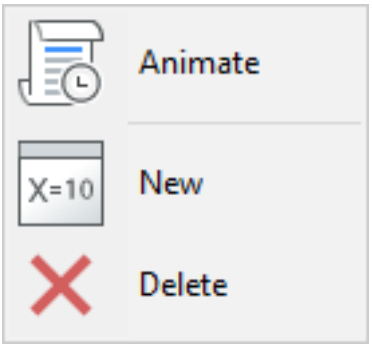
Search

Name	Expression	Value
CX	Length_X_Y	500
CX_14	p_4	50
CX_15	p_4	50
CX_18	p_4	50
CX_19	p_4	50
CX_21	p_1	10
CX_22	p_1	10
CX_23	p_1	10
CY	Length_X_Y	500
CY_1	p_3	40
CY_12	p_4	50
CY_13	p_4	50
CY_16	p_4	50
CY_17	p_4	50
CY_2	p_3	40
CZ	Length_Z	885
CZ_10	CZ*ratio_1	484.998494833
CZ_11	(CZ-CZ_20)*ratio_2	160
CZ_20	p_2	35
CZ_7	Length_Z	885
Length_X_Y	500	500
Length_Z	885	885
p_1	10	10

Command Options

Option	Description
Context menu	Right-click on any parameter: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/Pa • **Animate** : Allows to check the correctness of assigned parameter and see corresponding geometry modifications dynamically. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/Pa • **New** : Create a new parameter. A new parameter using default values is added at the bottom of the list. • **Delete** :Deletes a dimensional constraint or user parameter.
	Click to create a new parameter. A new parameter using default values is added at the bottom of the list.
	Deletes the selected parameter(s). To select multiple parameters: • Click the first parameter of a series, then hold down the Shift key and select the last parameter. • Hold down the Ctrl key to add parameters to the selection.
	Shows the parameter(s) driven by the selected parameter. Select a parameter, then click the Show Driven Parameters button. Click the button again to display the complete list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/Pa
	Shows the driving parameter(s) of the selected parameter. Select a parameter, then click the Show Driving Parameters button. Click the button again to display the complete list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/Pa
	Filters the parameters list as you type. Only parameters which contain the character string display.
Name Expression Value	Column headings allow to sort the parameters list: 1. Click the column heading. A small triangle displays. The list is sorted alphabetically or small to large. 2. Click again to sort in reverse order. 3. Click again to sort in creation order. The small triangle disappears.

Figure 6.1610



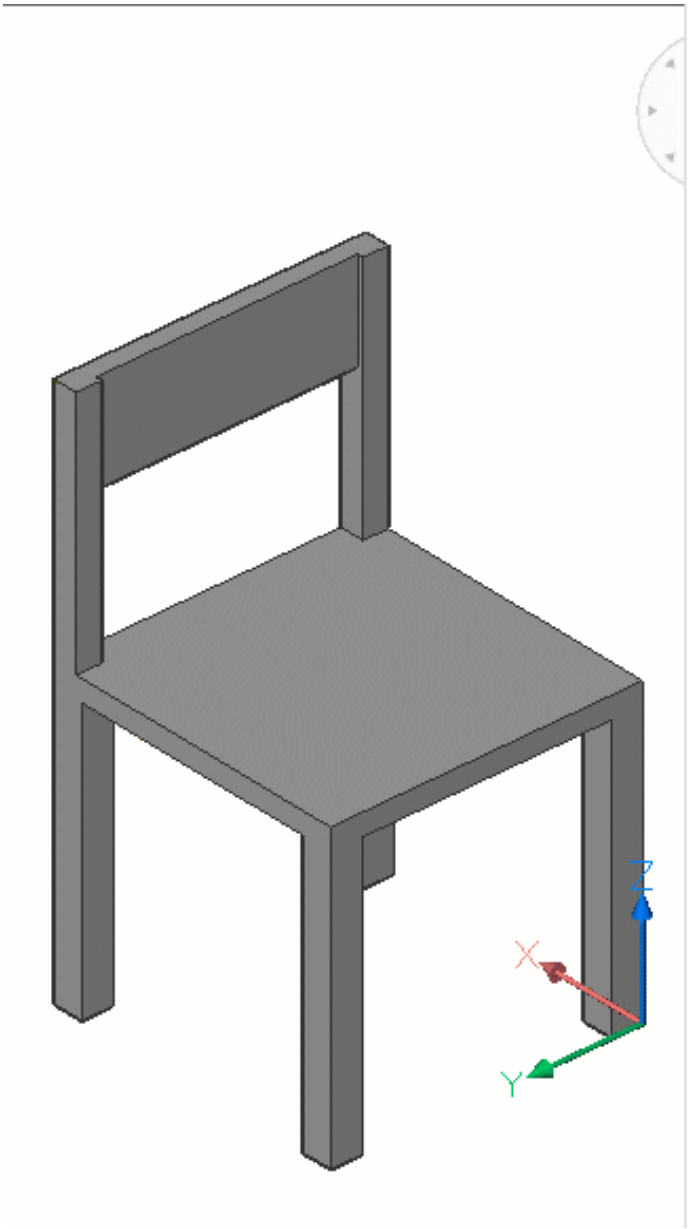
Procedures

Editing the parameters and constraints in the parameters panel (2.4.18)

Related Commands

CleanUnusedVariables (6.9.23)- purges variables not used in constraint expressions and not linked to dimensions. -BmParameters (6.8.19)- allows to list and edit the parameters of inserted components. -Parameters (6.24.5)- manages 2D dimensional constraint parameters and user variables in the current drawing. -DesignTableEdit (6.10.27)- allows to edit designtables. MechanicalBrowserOpen (6.24.8) - opens the Mechanical Browser panel.

Figure 6.1611



Search

Name	Expression	Value
ratio_2	0.188235294118	0.18823529411
ratio_1	0.548020898116	0.54802089811
p_4	50	50
p_3	40	40
p_2	35	35
p_1	10	10
Length_Z	885	885
Length_X_Y	500	500
CZ_7	Length_Z	885
CZ_20	p_2	35
CZ_11	(CZ-CZ_20)*ratio_2	160
CZ_10	CZ*ratio_1	484.998494833
CZ	Length_Z	885
CY_2	p_3	40
CY_17	p_4	50
CY_16	p_4	50
CY_13	p_4	50
CY_12	p_4	50
CY_1	p_3	40
CY	Length_X_Y	500
CX_23	p_1	10
CX_22	p_1	10
CX_21	p_1	10
CX_19	p_4	50
CX_18	p_4	50
CX_15	p_4	50
CX_14	p_4	50
CX	Length_X_Y	500

Figure 6.1612

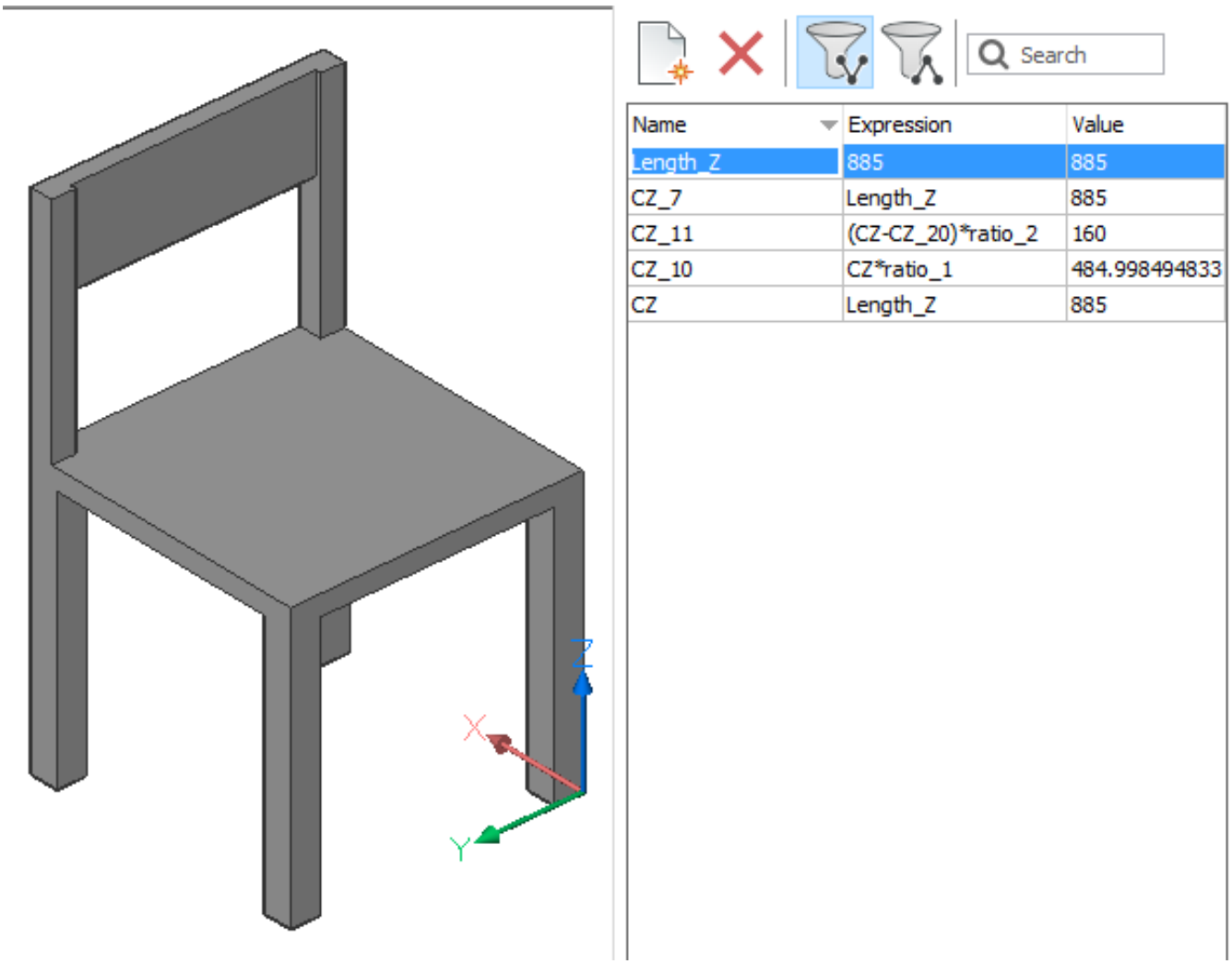


Figure 6.1613

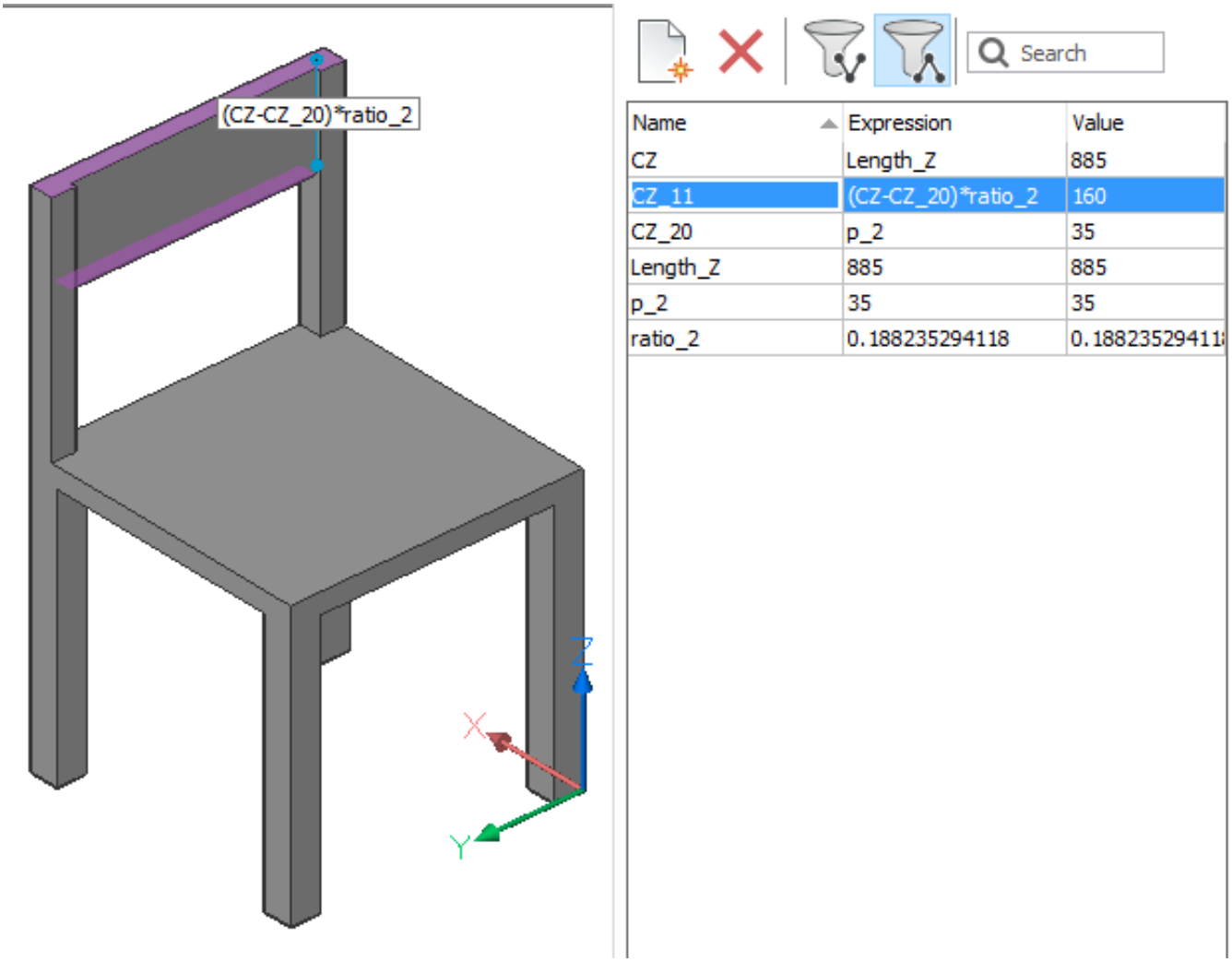


Figure 6.1614



6.24.9 PasteBlock

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Pastes CAD entities from the Clipboard as a block. **Note:**Before you can use this command, first use the the CopyClip (6.9.52) orCopyBase (6.9.51) commands to copy entities from the current drawing, another drawing, or even from another CAD program.

Accessing the Command

 command bar: **pasteblock** shortcut: **Ctrl+Shift+V** menu: **Edit | Paste as Block** ribbon:**Home | Edit | Paste | Base As Block** : **PasteBlock** When the Clipboard contains entities copied with the CopyClip (6.9.52) command, **you are prompted in the command bar:** Select insertion point: **(Pick a point, or enter x,y coordinates)** When the Clipboard contains entities copied withCopyBase (6.9.51), entities are pasted at their base point, and so you are not prompted in the command bar. This command can only paste CAD entities into the drawing as entities; other Clipboard content may be pasted as OLE entities.

Command Options

Option	Description
Select insertion point	Specifies the insertion point for the entities pasted from the Clipboard: • Pick a point • Enter x,y coordinates

Procedures

- To paste a selection set as a block (1.23.17)
- Paste a selection as a block in the target drawing (1.23.17)
- Move a selection as a block (1.23.11)

Related Commands

CopyBase (6.9.51)- copies selected entities to the Clipboard with a base point. CopyClip (6.9.52) - copies selected entities to the Clipboard. CopyHist (6.9.54) - copies all text from the Text window to the Clipboard. CutClip (6.9.59) - cuts (moves) selected entities to the Clipboard. PasteClip (6.24.10) - pastes entities from the Clipboard into the current drawing. PasteOrig (6.24.41) - pastes entities from the Clipboard at the original coordinates. PasteSpec (6.24.11) - displays the Paste Special dialog box for controlling the format of pasted entities.

Figure 6.1615



6.24.10 PasteClip

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Pastes entities from the Clipboard into the drawing (short for "paste clipboard"). **Note:**Before you can use this command, first use the the CopyClip (6.9.52) orCopyBase (6.9.51) commands to copy entities from the current drawing, another drawing, or even from another CAD program.

Accessing the Command


command bar: **pasteclip** shortcut: **Ctrl+V** menu bar: **Edit | Paste** ribbon:**Home | Edit | Paste** toolbar: **Standard |**  : **PasteClip** When the Clipboard contains BricsCAD entities (including BricsCAD text), **this command prompts you in the command bar:** Select insertion point: **(Pick a point in the drawing, or specify x,y coordinates)** When the Clipboard contains non BricsCAD entities

- Non-BricsCAD text (including LISP and Diesel code) is pasted in the command bar, and the program execute the text as commands
- Non-BricsCAD entities, such as raster images, are pasted as OLE entities in the upper-left corner of the viewport

Command Options

Option	Description
Select insertion point	Specifies the insertion point for the entity: • Pick a point • Enter x,y coordinates

Procedures

- Using Copy and Paste to copy entities between drawings (1.23.17)
- Using Copybase and Paste to copy entities between drawings (1.23.17)

Related Commands

ChSpace (6.9.19) - moves entities from paper space to model space and vice versa. CopyBase (6.9.51)- copies selected entities to the Clipboard with a base point. CopyClip (6.9.52) - copies selected entities to the Clipboard. CopyHist (6.9.54) - copies all text from the Text window to the Clipboard. CutClip (6.9.59) - cuts (moves) selected entities to the Clipboard. PasteBlock (6.24.9) - pastes entities from the Clipboard as a block at an insertion point. PasteOrig (6.24.41) - pastes entities from the Clipboard at the original coordinates. PasteSpec (6.24.11) - displays the Paste Special dialog box for controlling the format of pasted entities.

Figure 6.1616



6.24.11 PasteSpec

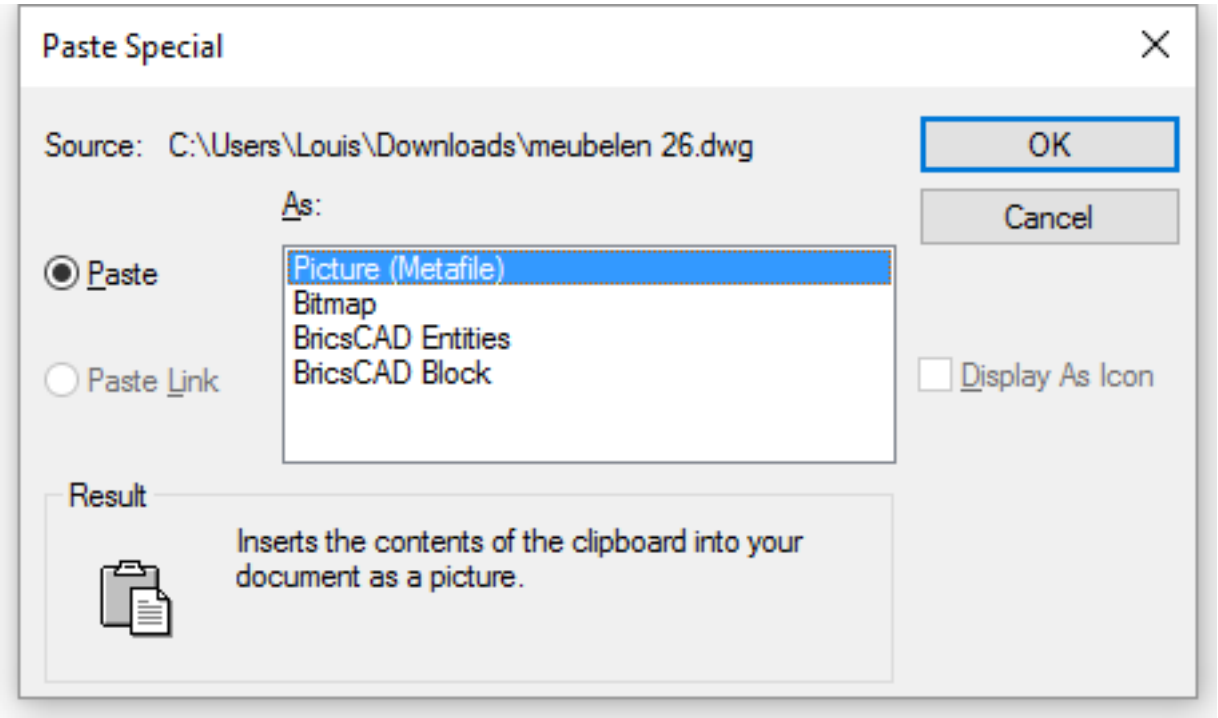
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Gives you the choice of the type of object to paste from the Clipboard into the drawing (short for ”paste special”). **Note:**Before you can use this command, first copy content from a drawing or other document to Clipboard using **Edit | Copy**.

Accessing the Command



command bar: **pastespec** alias: **pa** shortcut:**Ctrl+Alt+V** menu bar: **Edit | Paste Special** ribbon:**Home | Edit | Paste | Paste Special** : **PasteSpec** When Clipboard contains CAD data, displays a dialog box that looks like this: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/PasteSpecialDrawingDialog.png}) Choose a format under As, and then click **OK**.

Figure 6.1617



Command Options

Option	Description
As	Chooses the format in which to paste the entities.This list varies, depending on the content of the Clipboard. When the Clipboard contains data copied from a BricsCAD drawing, then the following formats are available: • **Picture Metafile** - entities are converted to a WMF-format image and then pasted as an OLE image • **Bitmap** - entities are converted to a BMP-format image and then pasted as an OLE image • **BricsCAD Entities** - pasted as individual entities; see below • **BricsCAD Block** - the entities in the Clipboard are pasted as a block; see below Here is the same dialog box, but with entities copied from an Excel spreadsheet: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/

Paste As BricsCAD Entities	Pastes CAD entities into the drawing as BricsCAD entities; prompts you: **Select insertion point or [Rotate/Scale/Mirror]** - **(enter an option)**: 1. **Select insertion point** - **(specify a point in the drawing by picking a point or entering x,y coordinates.)** 2. **Rotate** - prompts you: **Specify rotation angle** - **(enter an angle by which to rotate the entities.)** 3. **Scale** - prompts you: **Specify scale factor for XYZ axes** - **(enter a scale factor.)** 4. **Mirror** - prompts you: **Select mirroring control point** - **(pick a second point for the mirror line.)** The insertion point is the first point for the mirror line. When you enter 0,0 for the insertion point, the entities are pasted to their original coordinates.
Paste As BricsCAD Block	Pastes the CAD entities into the drawing as a block: 1. Starts the Insert command and then displays the Insert Block dialog box. 2. Fill out the properties and then click **OK**. 3. The program gives a block a generated name, such asA$C5C4F59B9, and inserts it into the drawing.
Paste Link	(This option is not available for CAD entities.) Pastes the entities in the drawing as an OLE object, and then creates a link to the originating application.
Display as icon	(This option is not available for CAD entities.) Displays the pasted entities as an icon identifying their source application.

Related Commands

CopyBase (6.9.51)- copies selected entities to the Clipboard with a base point. CopyClip (6.9.52) - copies selected entities to the Clipboard. CopyHist (6.9.54) - copies all text from the Text window to the Clipboard. CutClip (6.9.59) - cuts (moves) selected entities to the Clipboard. PasteBlock (6.24.9) - pasted entities from the Clipboard as a block at an insertion point. PasteClip (6.24.10) - pastes entities from the Clipboard into the current drawing. PasteOrig (6.24.41) - pastes entities from the Clipboard at the original coordinates. Table (6.29.1) - creates a table entity in the drawing through a dialog box.

Figure 6.1618

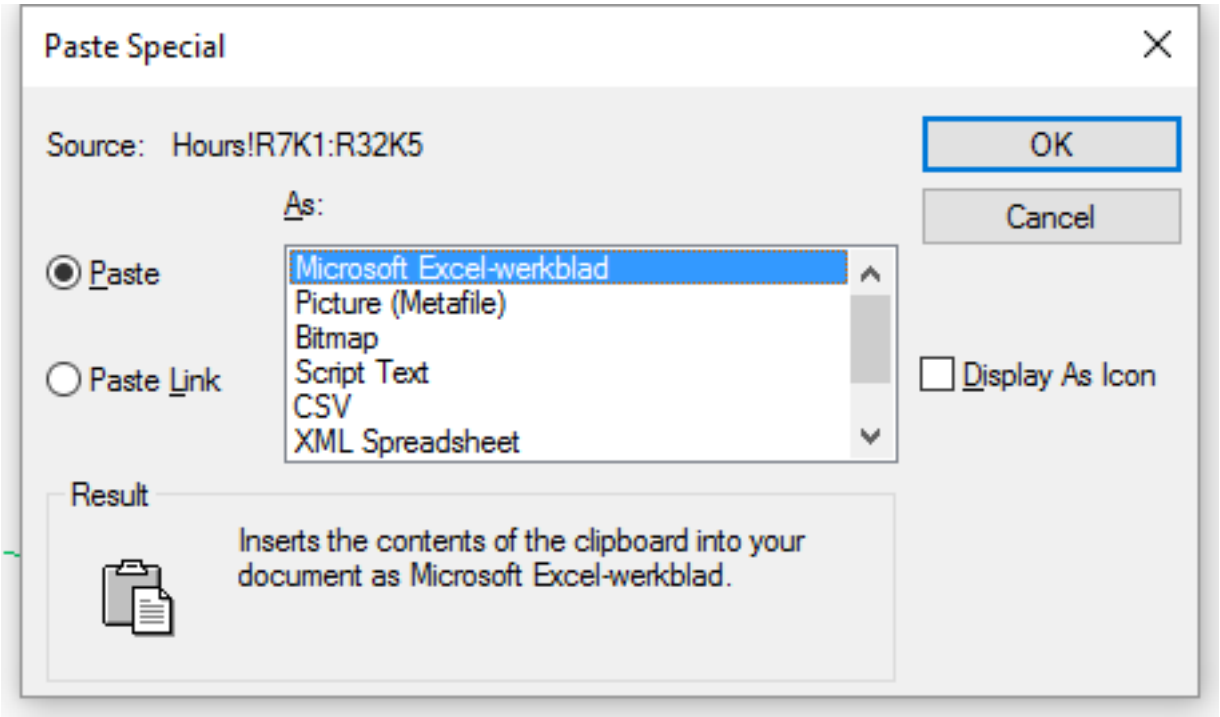


Figure 6.1619



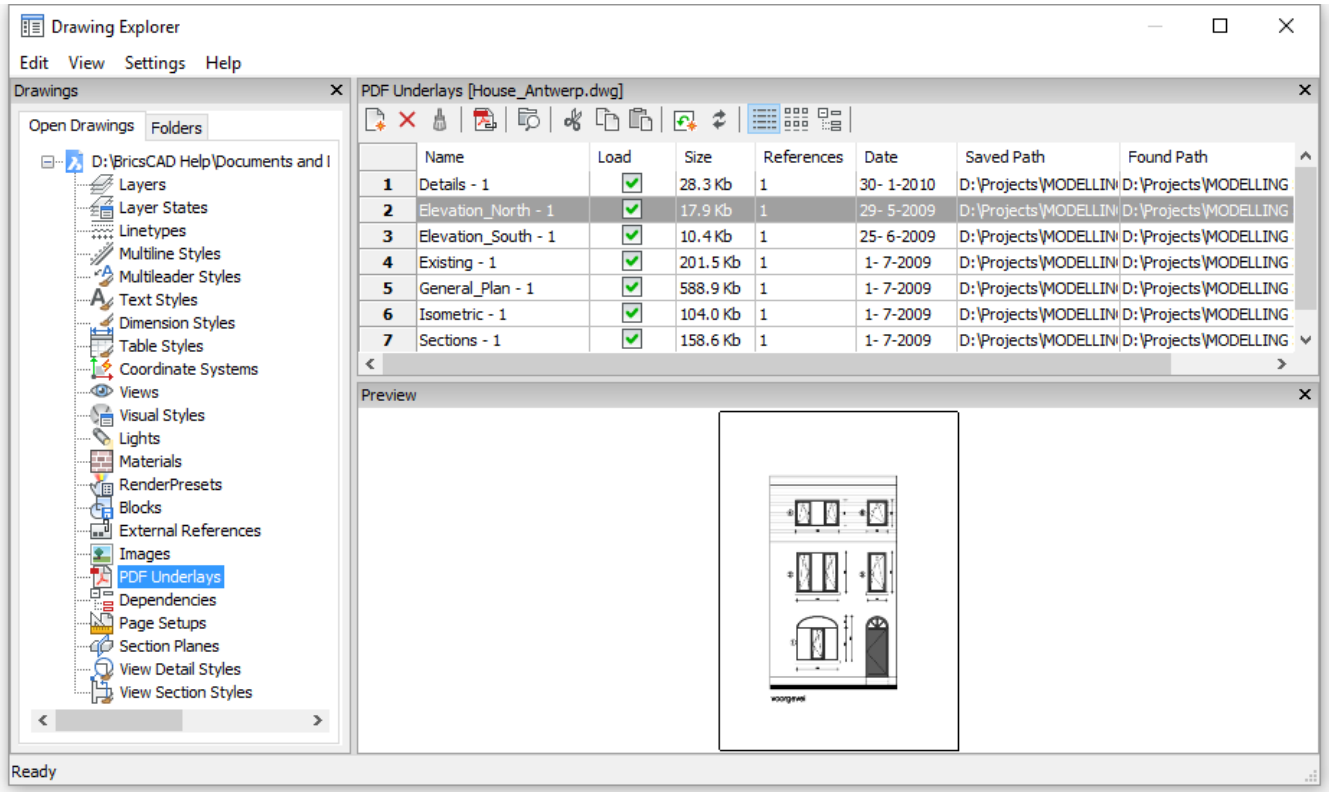
6.24.12 Pdf

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Attaches PDF files as underlays to the current drawing through the Drawing Explorer. **Note:** To import PDF files as drawing entities, use the PdfImport (6.24.17) command.

Accessing the Command

 command bar: **pdf** menu bar: **Tools | Drawing Explorer | PDF Underlays** ribbon:**Home | Settings | Drawing Explorer | PDF Underlays** **Insert | PDF | PDF Underlays** toolbar: **Insert** | : **Pdf** Displays the PDF Underlays section of the Drawing Explorer: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Drawing

Figure 6.1620



Command Options

Toolbar Option	Description
New	Attaches a PDF file to the current drawing: 1. Displays the Select PDF Underlay File dialog box; select a PDF file. 2. Click **Open**. 3. Displays the Attach PDF Underlay dialog box; specify parameters. 4. Click **OK**. 5. Prompts you in the command line: **Insertion point - (pick a point to locate the lower-left corner of the PDF.)** See the PdfAttach (6.24.14)command.
Delete	Detaches the selected PDF attachments without warning.

Insert	Attaches additional pages from a multi-page PDF file already attached to the current drawing: 1. Displays the Attach Pdf Underlay dialog box; choose a page number from the **Page** droplist (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/page-droplist.png}) 2. Fill out the parameters 3. Click **OK**. 4. Prompts you in the command line: **Insertion point** - **(pick a point to locate the lower-left corner of the PDF.)** 5. The added page appears in Drawing Explorer with the page number as a suffix to its name: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pd page-listt.png})
Cut Copy Paste	Cuts or copies information about the PDF attachment to the Clipboard so that it can be pasted into another drawing; you cannot paste the data into the current drawing.
Detail View	Displays the details of attached PDF files, as shown below.
Tree View	Displays the relationships between the current drawing and attached PDF files in a dendritic view: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pd tree-view.png})

PDF Underlays Option	**Description**
Name	Reports the name of the PDF insert. This name is generated by the program, and be edited by you: • Click the name twice slowly • Right-click, and then choose **Rename** from the shortcut menu
Loaded	Toggles loading of the PDF: • **On** - loads the PDF, and makes it visible • **Off** - unloads the PDF, making it invisible
Size	Reports the size of the PDF file.
References	Reports the number of times the PDF is attached to the drawing.
Date	Reports the date of the file. This can be useful in determining whether you ware working with the most recent revision.

Saved Path	Reports the path to the PDF file when first loaded. Whenthe path cannot be found: 1. Click this field 2. Click the **Browse** button. BricsCAD displays the Choose a File dialog box. 3. Locate the missing PDF file. 4. Click **Open**.
Found Path	Reports the path to the PDF file.

Related Commands

PdfAdjust (6.24.13) - Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar. PdfAttach (6.24.14) - inserts a PDF file as an underlay into the current drawing. Pdfclip (6.24.16) - allows clipping of PDF underlays. Pdflayers (6.24.18) - allows to control the display of layers in a PDF underlay. XRef (6.34.8) - attaches and controls externally referenced drawings through the Drawing Explorer. -XRef (6.34.9) - attaches and controls xrefs through the command bar. XClip (6.34.2) - clips externally-referenced drawings. Insert (6.17.13) - inserts drawings as blocks. Explorer (6.13.14) - opens the Drawing Explorer dialog box.

Figure 6.1621

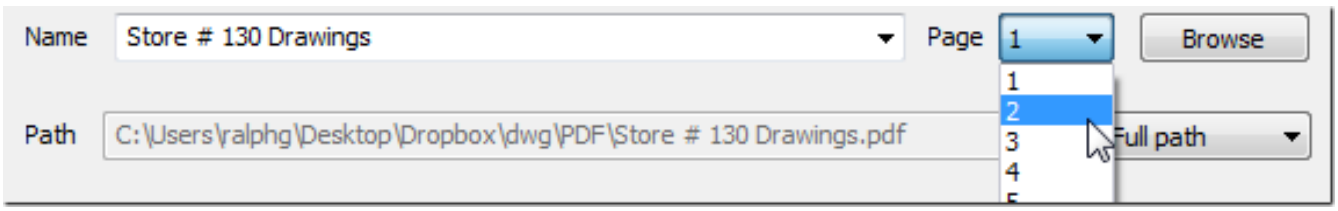


Figure 6.1622

	Name	Load	Size	R...	Date	Saved Path	Found Path
1	Store # 1 Drawings - 1	✔	44.5 Mb	0	5- 4-2013	C:\Users\ralphg\Des	C:\Users\ralphg\
2	Store # 1 Drawings - 3	✔	44.5 Mb	0	5- 4-2013	C:\Users\ralphg\Des	C:\Users\ralphg\

Figure 6.1623

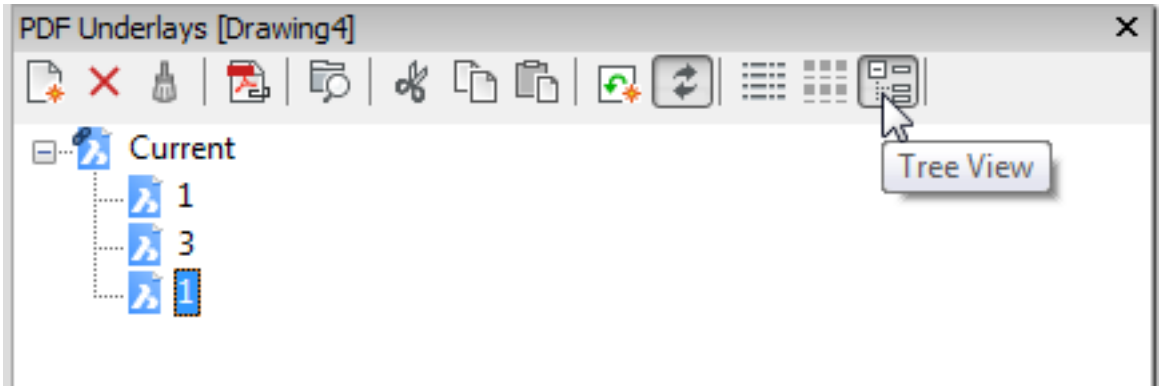
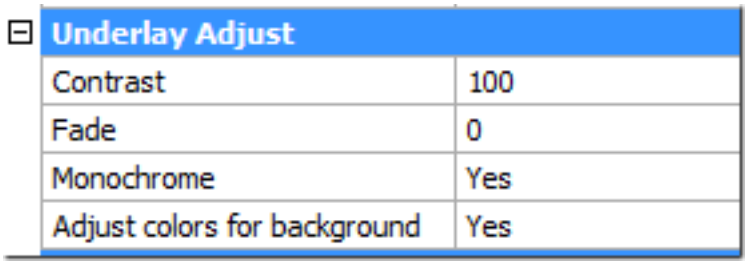


Figure 6.1624



Figure 6.1625



6.24.13 PdfAdjust

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adjusts the fade, contrast and monochrome level of one or more PDF underlays attached to the current drawing. **Note:** You may find it easier to use the Properties (6.24.49) panel's Underlay Adjust section to change these parameters. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/p adjust-properties.png})

Accessing the Command

command bar: **pdfadjust** transparent: **'pdfadjust** menu: **Modify | PDF Underlay | Adjust PDF** : **PdfAdjust Prompts you in the command bar:** Select PDF underlay: **(Select one or more PDF underlays)** Entities in set: 1 Select PDF underlay: **(Select more PDF underlays, or else press Enter to continue)** Enter PDF underlay option [Fade/Contrast/Monochrome] <Fade>: **(Enter an option)**

Command Options

Option	Description
Fade	Sets the fade setting of the PDF underlay(s); prompts you: **Enter fade value [0-100] <0>** - **(type a value)**: • **0** - minimum fade: PDF underlay is fully displayed • **100** - maximum fade: PDF is barely visible
Contrast	Sets the contrast setting of the PDF underlay(s); prompts you: **Enter contrast value [0-100] <100>** - **(type a value.)** • **0** - minimum contrast: both light and dark elements are medium gray • **100** - maximum contrast: dark elements are dark, light elements light
Monochrome	Toggles the monochrome setting of the PDf underlay(s); prompts you: **Monochrome? <Yes>/No** - **(enter an option)**: • **Yes** - turns colors into shades of gray • **No**- keeps PDF as-is

Procedures

- To adjust the display settings of a PDF underlay (1.22.21)

Related Commands

Figure 6.1626



6.24.14 Pdfattach

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Attaches a PDF file as an underlay into the current drawing through dialog boxes.
Note: To import PDF files as drawing entities, use the PdfImport (6.24.17) command.

Accessing the Command

 command bar: **pdfattach** transparent: **'pdfattach** menu bar: **Insert | Attach Pdf** ribbon: **Insert | PDF | Attach PDF** toolbar: **Insert |** **: Pdfattach** Displays adialog box: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/hel
Select a PDF file, then click **Open**.The program displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
Specify parameters, and then click **OK**. The program attaches the PDF file to the drawing as an underlay.

Figure 6.1627

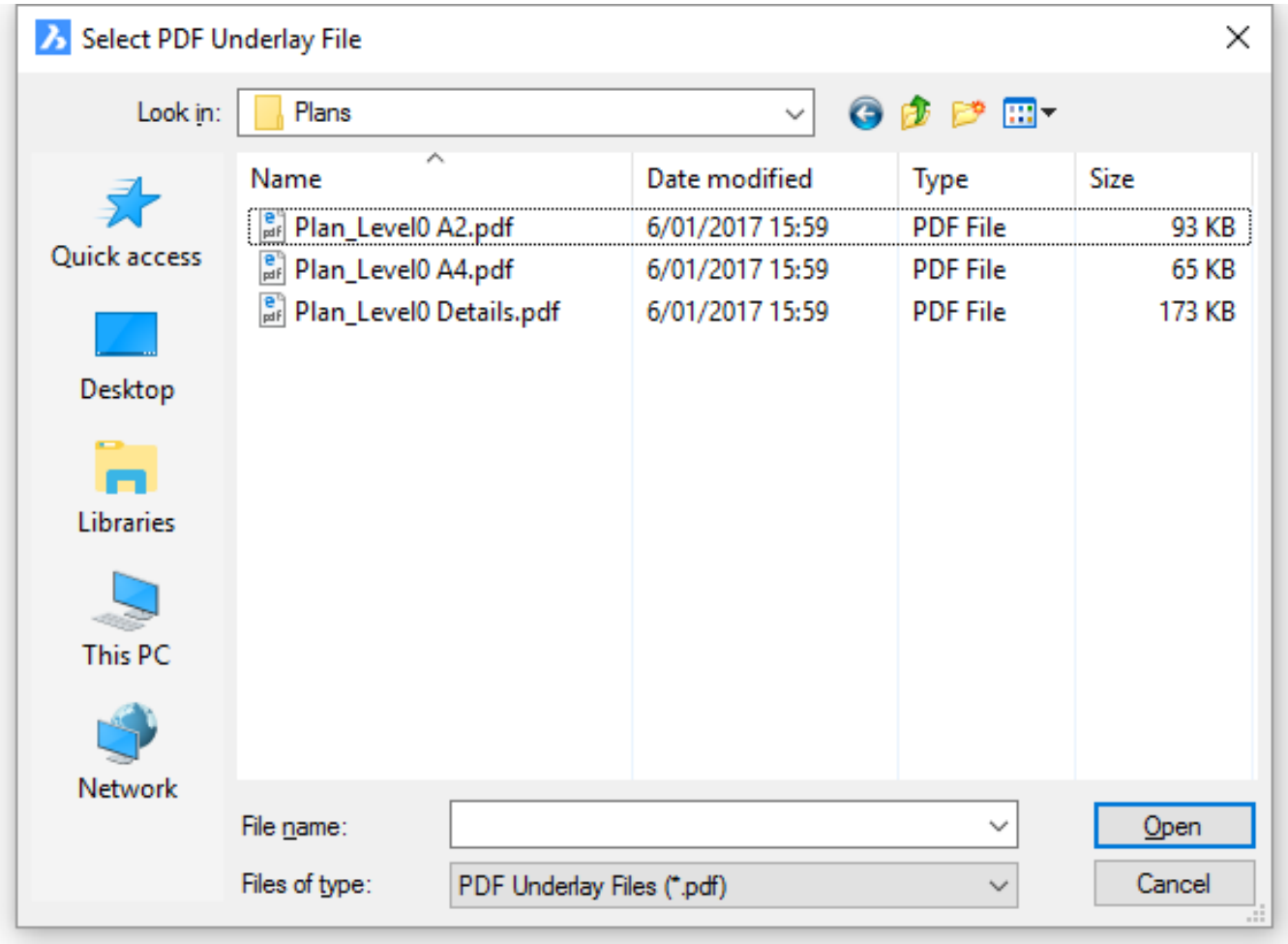
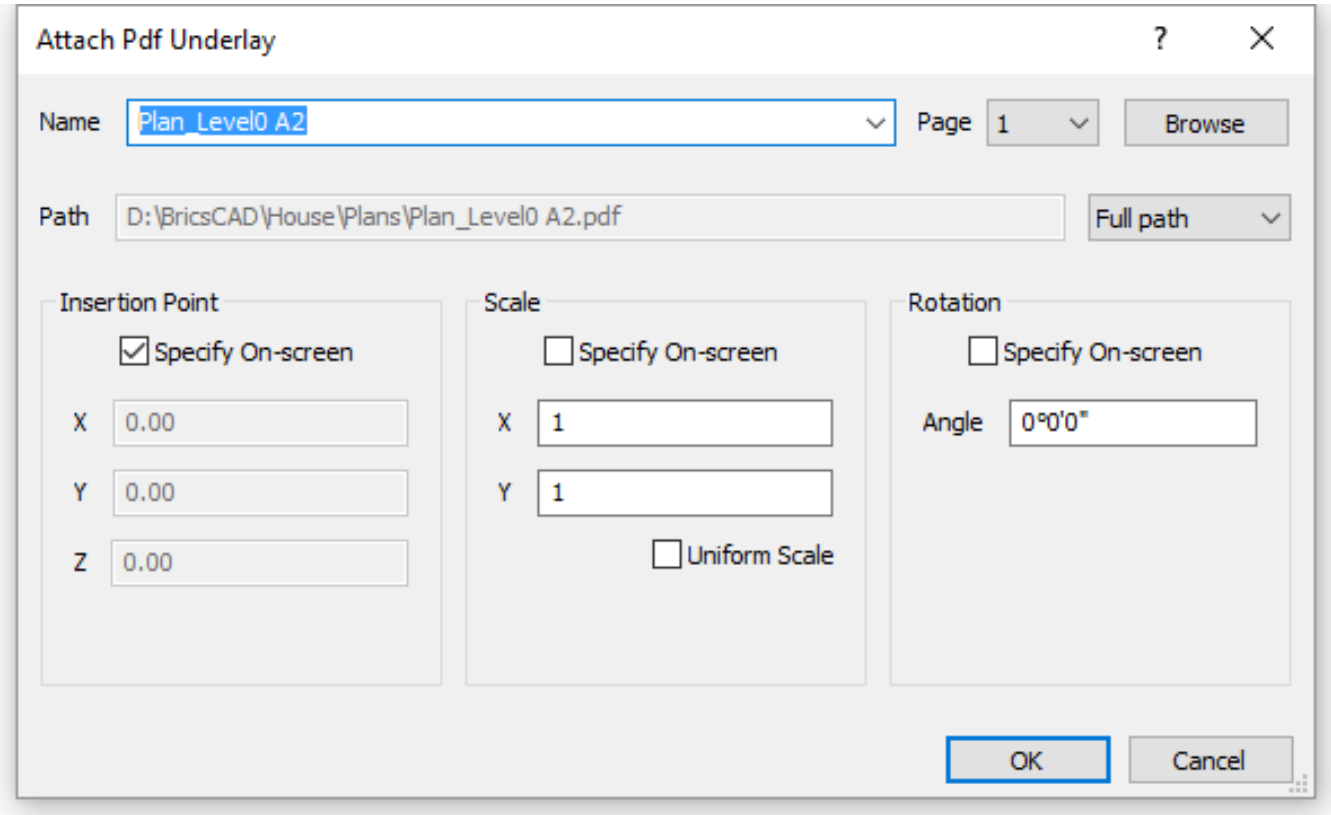


Figure 6.1628



Command Options

Attach PDF Option	Description
Name	Specifies the name of the attached PDF file. Whenmore than one PDF file has been attached to the drawing, their names are listed by the droplist.
Browse	Selects a different PDF file from your computer's disk drive or network: 1. In the Select PDF Underlay dialog box, choose a PDF file. 2. Click **Open**.
Path	Chooses the way the program remembers the path to the PDF file: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ 1. Click the **Path** droplist. 2. Choose an option: • **Full path** - stores the drive and folder(s) name to the PDF file as an absolute reference, such as: c:\cad\PDF\filename.pdf • **Relative path** - stores the part of the path to PDF file's folder, such as ..\filename.pdf .This option does not work until the drawing has been saved at least once.(..\ means one folder higher.) • **No path** - strips the drive and folder names, leaving just the PDF file name, such as filename.pdf.
Insertion Point	Specifies the location of the underlay's lower-left corner: **Specify On-screen** - determines how the insertion point is specified: • **On** - specify the insertion point in the drawing after the dialog box is closed; you will be prompted: **Insertion point for block** - **(pick a point or enter x,y,z coordinates.)** • **Off** - specify the insertion point in the dialog box, using the X, Y, and Z fields: **X**, **Y**, **Z** - specifies the x,y,z coordinates for the PDF underlay's insertion point; use 0,0,0 to insert the underlay at the drawing's origin.

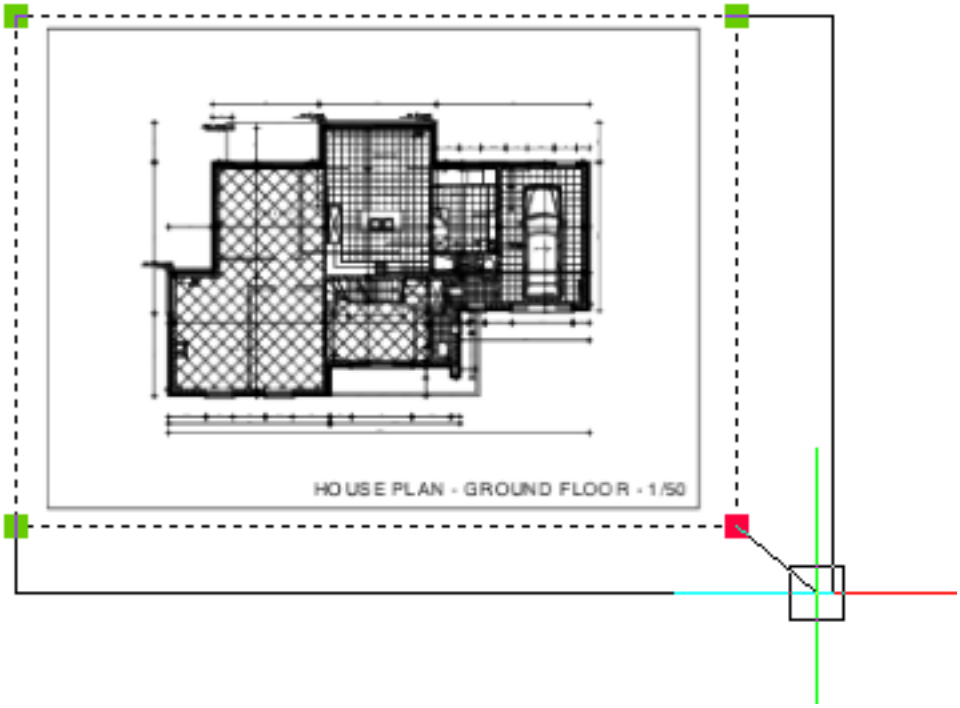
Scale	Specifies the size of the PDF underlay: **Specify On-screen** - determines how the scale factors are specified: • **On** - specify the scale factor in the drawing after the dialog box is closed; you will be prompted: **Scale factor <1.00>**- **(type a value or specify a point.)** • **Off** - specify the scale factor in the dialog box using the X, Y, and Z fields: **X**, **Y**, **Z** - specifies the scale factors of the underlay in the x, y, and z directions; use 1,1,1 to keep the underlay at its original size
Rotation	Specifies the rotation angle of the PDF underlay: **Specify On-screen** - determines how the rotation angle is specified: • **On** - specify the rotation angle in the drawing after the dialog box is closed; you will be prompted: **Rotation angle <0.00>** - **(pick two points or enter an angle.)** • **Off** - specify the rotation angle in the dialog box, using the X, Y, and Z fields: **Angle**- specifies the rotation angle about the insertion point. Positive angles rotate the underlay counterclockwise. Use 0 to keep the underlay at it original orientation.
OK	Attaches the PDF as an underlay.

Grips Editing

The scaling of a PDF underlay can be edited directly through grips.

1. Select the PDF underlay.It has four grips.
2. Drag a grip to adjust the scaling of the PDF underlay. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/hel

Figure 6.1629



Procedures

- To control the PDF underlay caching (1.22.21)
- To attach a PDF underlay (1.22.21)
- To insert a PDF underlay (1.22.21)
- To control the display of a PDF underlay (1.22.21)
- To set the layer display in a PDF underlay (1.22.21)
- To clip a PDF underlay (1.22.21)

Related Commands

PdfAdjust (6.24.13) - Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar. -PdfAttach (6.24.15) - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar. Pdfclip (6.24.16) - allows clipping of PDF underlays. Pdflayers (6.24.18) - allows to control the display of layers in a PDF underlay. XRef (6.34.8) - attaches and controls externally referenced drawings through the Drawing Explorer. -XRef (6.34.9) - attaches and controls xrefs through the command bar. XClip (6.34.2) - clips externally-referenced drawings. Insert (6.17.13) - inserts drawings as blocks. -Insert (6.17.14) - inserts blocks, by prompting in the command bar and prompts for attribute values through the command bar or a dialog box.

Figure 6.1630

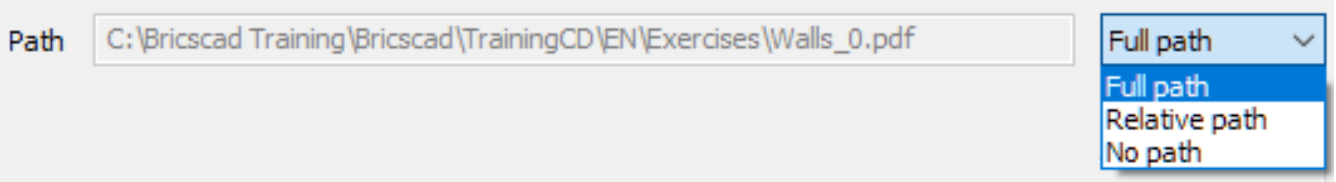


Figure 6.1631



6.24.15 -PdfAttach

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Attaches a PDF file as an underlay into the current drawing through the command bar. **Note:** To import PDF files as drawing entities, use the PdfImport (6.24.17) command.

Accessing the Command

command bar: **-pdfattach** transparent: **'-pdfattach** : **-Pdfattach** PDF Underlay file: **(Type the path and name of the PDF document, or else press ~ to display the file dialog box)** PDF Underlay page number <1>: **(Type the page number, or else press Enter to accept the current page number)** Insertion point: **(Specify the insertion point in the drawing)** Scale [Size/XY scales] <1.00>: **(Type a value in the command bar, pick a point in the drawing, or else enter an option)** Rotation angle <0.00>: **(Type a value, or else pick a point in the drawing)**

Command Options

Option	Description
PDF Underlay file	Specifies the file name of the PDF file to attach: • Type the path and name of the PDF file • Press ~ to display the file dialog box
PDF Underlay page number	(This prompt does not appear when the PDF file has just one page.) Specifies the page number of a multi-page PDF file to insert: • Type the page number • Press **Enter** to accept the current page number
Insertion point	Specifies the insertion point in the drawing: • Pick a point in the drawing • Enter x,y coordinates for the lower left corner of the PDF attachment
Scale	Specifies the size of the PDF underlay: • Enter a scale factor: Larger than 1.0 increases the size of the underlay Smaller than 1.0 makes the underlay smaller • Move the cursor to dynamically size the underlay
Size	Sets the size of the insert dynamically; prompts you: **Specify opposite corner** - **(pick a point.)** The first point is the insertion point. As you move the cursor, the program ghosts the size and position of the underlay relative to the insertion point.

XY Scale factors	Scales the underlay separately in X- and Y direction differently; prompts you: 1. **Specify X scale factor <1.00>** - **(type a value or pick a point.)** 2. **Specify Y scale factor < Equal to X scale factor <1.00>** - **(enter an option)**: • Press **Enter** to set the Y scale factor equal to the X scale factor • Pick a point in the drawing • Type a value
Rotation angle	Specifies the angle of the underlay: • Enter an angle: Positive numbers rotate the underlay counter clockwise from the x-axis Negative numbers rotate it clockwise • Move the cursor to dynamically rotate the underlay

Related Commands

PdfAdjust (6.24.13) - Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar. Pdfclip (6.24.16) - allows clipping of PDF underlays. Pdflayers (6.24.18) - allows to control the display of layers in a PDF underlay. XRef (6.34.8) - attaches and controls externally referenced drawings through the Drawing Explorer. -XRef (6.34.9) - attaches and controls xrefs through the command bar. XClip (6.34.2) - clips externally-referenced drawings. Insert (6.17.13) - inserts drawings as blocks. -Insert (6.17.14) - inserts blocks, by prompting in the command bar and prompts for attribute values through the command bar or a dialog box.

Figure 6.1632



6.24.16 Pdfclip

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Clips a PDF underlay with a rectangular or polygonal boundary, and inverts the boundary.

Accessing the Command

 command bar: **pdfclip** command bar: **'pdfclip** ribbon:**Insert | PDF | Clip PDF : Pdfclip** Select PDF underlay: **(Choose a pdf underlay)** Enter PDF clipping option [ON/OFF/Delete/New] <New>: **(Enter an option, or else press Enter to execute the default option)**

Command Options

Option	Description
Select PDF underlay	Selects the PDF underlay to clip: click the frame of the PDF underlay, not the body of the PDF itself.
ON	Turns the clipping boundary on; the area outside the clipping boundary is hidden from view.
OFF	Turns the clipping boundary off; all of the PDF underlay is again visible. The clipping boundary is saved.
Delete	Deletes an existing clipping boundary.
Invert	Inverts the clipping boundary. Initially, the PDF underlay outside of the clipping boundary is hidden. Use this option to invert it, so that the portion of the PDF inside the boundary is hidden, and that outside is visible. You may find it more convenient to use the Misc section of the Properties (6.24.49) panel to turn the clipping on and off, and to invert it.
New	Draws a new clipping boundary: Whenno boundary exists, you are prompted: **Enter PDF clipping type [Polygonal/Rectangular] <Rectangular>** - **(type P or R)**: • **Polygonal** -draws multi-sided clipping boundary; see below • **Rectangular** -draws rectangular clipping boundary; see below When a boundary already exists you are first prompted: **Delete old boundary? [Yes/No]** - **(type Y or N)**: • **Yes**- replaces the existing clipping boundary, and then proceeds to prompts you for the new one • **No**- the program exits the command

Polygonal	Draws multi-sided clipping boundaries; prompts you: 1. **Start of polyline** - **(pick a point in the drawing, or specify x,y coordinates.)** 2. **N ext point** - **(pick another point in the drawing, or specify x,y coordinates.)** 3. **Undo/next point** - **(pick another point in the drawing, specify x,y coordinates, or press Enter to complete the polygon.)** As you pick points, the program ghosts the polygon. Type **U** to undo the previous point.
Rectangular	Draws rectangular clipping boundary; prompts you: • **Specify first corner** - **(pick a point in the drawing, or specify x,y coordinates.)** • **Specify opposite corner** - **(pick another point in the drawing, or specify x,y coordinates.)** The program draws the rectangle.

Procedures

- To clip a PDF underlay (1.22.21)
- To toggle the display of the clipped part of a PDF underlay (1.22.21)
- To delete a clipping boundary (1.22.21)

Related Commands

ImageClip (6.17.7) - clips images. PdfAdjust (6.24.13) - Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar. pdfattach (6.24.14) - inserts a PDF file as an underlay into the current drawing. -PdfAttach (6.24.15) - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar. Pdflayers (6.24.18) - allows to control the display of layers in a PDF underlay. VpClip (6.31.17) - clips viewports. XClip (6.34.2) - clips externally-referenced drawings.

Figure 6.1633



6.24.17 PdfImport

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Imports a PDF andconverts its content to simple CAD entities; also converts a PDF underlay already attached to the drawing to CAD entities. **Note:** This command performs the following conversions:

PDF Entity	CAD Entity
Straight and curved lines	Polylines or splines
SHX text	Polylines
TrueType text	MText
Solid-filled areas and wide lines	Hatches (solid color) with 50% transparency
Raster images	Raster images
Layers	Layers
Lines with linetypes	Polyline segments

Accessing the Command


Command bar: **PdfImport** Ribbon: **Insert |PDF | PDF Import : PdfImport Prompts you:** Select pdf underlay or [File] <File>: **(Select a PDF underlay in the drawing, or else enter F to access the file dialog box)** When you select an underlay already in the drawing, the program prompts you: Specify first corner of area to import or [Polygonal/All/Settings] <All>:**(Pick a point, or enter an option)** Specify the opposite corner:**(Pick another point to specify the size of the PDF)** Keep, Detach or Unload PDF underlay? [Keep/Detach/Unload] <Unload>: **(Enter an option) When you select a PDF file, the program prompts you:** Specify page number to import or list pages [?/Settings] <1>:**(Enter a page number, or else enter an option)** Specify insertion point <0, 0>: **(Pick a point, or else enter x,y,z coordinates)** Specify scale <1.>:**(Specify a scale factor)** Specify rotation <0.>:**(Specify a rotation angle)**

Command Options

When you want to convert an attached PDF underlay to CAD entities:

Underlay Option	Description
Specify first corner of area to import	Picks the first of two points to convert a rectangular area of an underlay to CAD entities: 1. Pick a point in the drawing or specify x,y coordinates 2. Prompts you: **Specify the opposite corner - (pick the other point to form a rectangular area.)**
Polygonal	Picks three or more points to convert a polygonal area of an underlay to CAD entities: 1. **Specify first point - (pick a point on the PDF or else or specify x,y coordinates.)** 2. **Specify next point or [Undo] - (pick another point, or enter U to undo the last segment.)** 3. **Specify next point or [Undo] - (pick another point, or else press Enter to continue.)** When you pick points that are not on the PDF, the program ignores your pick points, and you have to pick more.
All	Selects the entire PDF underlay for conversion to CAD entities.
Settings	Displays the PDF Import Settings section of

Detach	Detaches the PDF underlay so that it is not longer a part of the drawing
Unload	Unloads the PDF underlay so that it is no longer seen in the drawing, but is still attached to the drawing.

When you want to convert a PDF file to CAD entities:

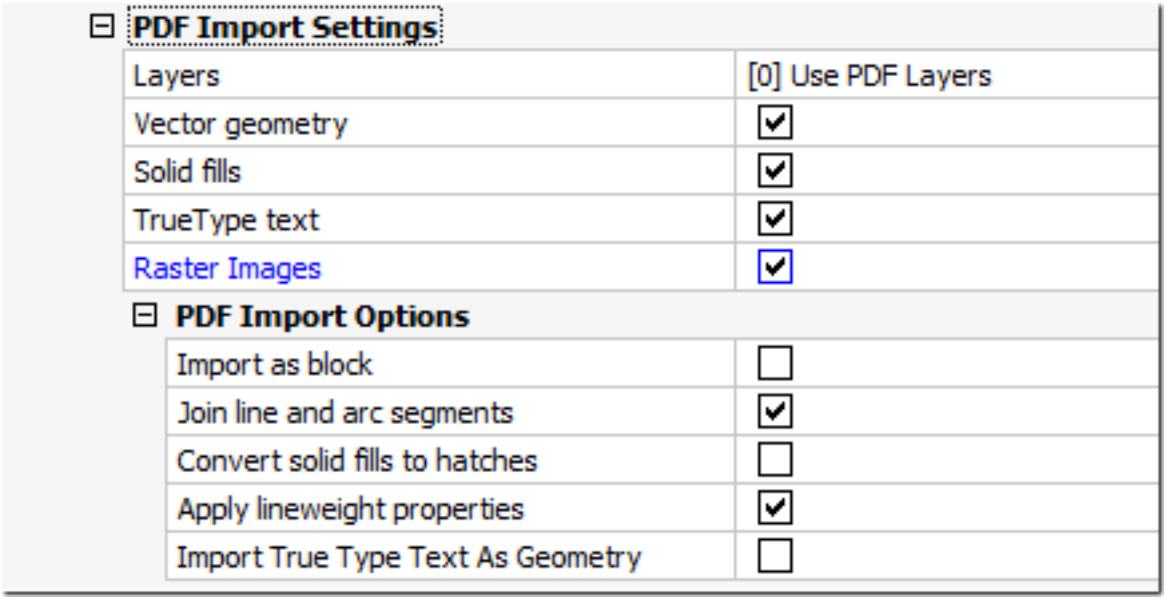
Import Option	**Description**
Specify page number to import	Imports a specific page from the PDF file: • Press **Enter** to import page 1, especially useful when the PDF file contains just one page • Press **?** (question mark) to list the page numbers in the PDF document; see below • Specify the page number, such as 3.
?	Lists the page numbers in the PDF document; prompts you: • **Page(s) to list <* for all> [*] - (press Enter, and thenpressF2 key to see the list in the Prompt History (6.29.13) window, such as this): 1 2 3 4** To view the content of the pages before importing them, use File Manager's preview window or a PDF viewing program like Acrobat.
Settings	Displays the PDF Import Settings section of theSettings (1.13.11)dialog box; see below.
Specify insertion point	Specifies the insertion point in the drawing: • Press **Enter** to accept the default of 0,0 • Pick a point in the drawing • Enter x,y coordinates for the lower left corner of the imported file
Specify scale	Specifies the scaling factor for the inserted geometry: • Press **Enter** to accept the default of 1 • Enter a scale value You cannot pick points in the drawing to show the size
Specify rotation	Specifies the rotation angle: • Press **Enter** to accept the default of 0 degrees • Enter an angle You cannot pick points in the drawing to show the angle

When you select the **Settings** option,the program displays the PDF Import Settings section of theSettings (1.13.11)dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pdfimport-settings-dlg.png})

Settings Option	**Description**

Layers	Determines how PDF layers are handled: • **0** - use the layers in the PDF files; incoming layer names are prefixed with 'PDF-', such as PDF_A-ANNO-DIM PDF_A-STAIR PDF_DOOR • **1** - segregate entities on separate layers, such as PDF_Geometry PDF_Images PDF_Text PDF_Solid Fills • **2** - place entire PDF on the current layer
Vector Geometry	Determines if vector geometry is imported: • **Yes** - imports and converts vector geometry • **No** - does not import vector geometry
Solids Fills	Determines if filled areas are imported: • **Yes** - imports and converts solid fills, such as filled areas and wide lines into hatches • **No** - does not import solid fills Hatched areas are given 50% transparency,
TrueType Text	Determines if TrueType text is imported: • **Yes** - imports TrueType text and assigns them style names based on the font name, such as: PDF Arial PDF Arial Bold PDF Times New Roman Bold • **No** - does not import text SHX text is converted to polylines
Raster Images	Determines if raster images are imported: • **Yes** - attaches raster images • **No** - does not import raster images Raster images are saved as PNG files in a folder.
Import as Block	Imports the PDF and inserts it as a block.
Join Line and Arc Segments	Attempts to join lines and polylines into a single segment, when possible.
Convert Solids Fills to Hatches	Converts solid fills into hatched areas
Apply Lineweight Properties	Gives imported lines with weight (width) a lineweight
Import TrueType Text as Geometry	Imports TrueType text as geometry, as is done with SHX text. The benefit here is if you do not have matching fonts.

Figure 6.1634



Related Commands

Pdfattach (6.24.14) - inserts a PDF file as an underlay into the current drawing; displays dialog boxes. -PdfAttach (6.24.15) - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar.

Figure 6.1635



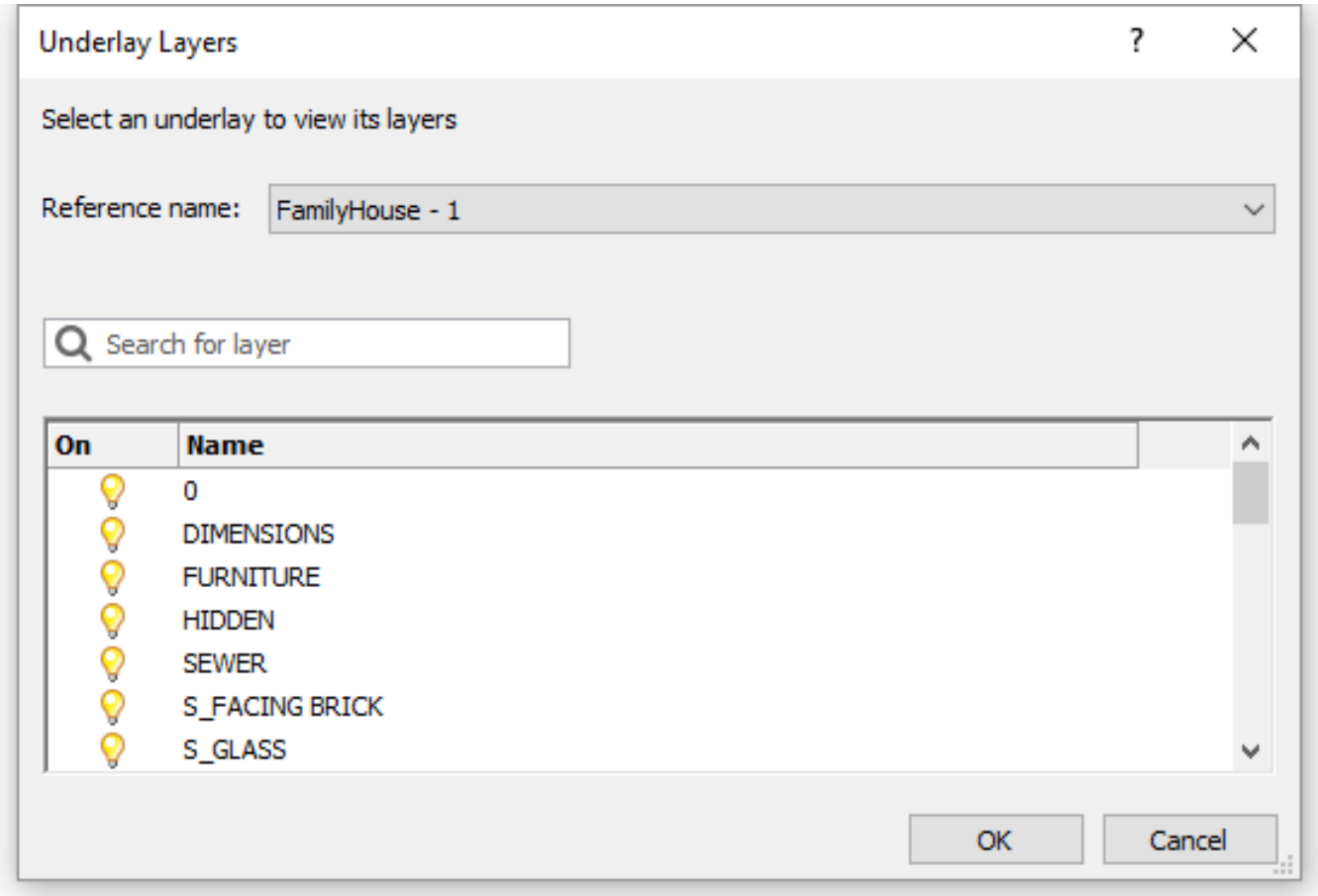
6.24.18 Pdflayers

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the display of layers in PDF underlays.

Accessing the Command

command bar: **pdflayers** transparent: **'pdflayers** : **Pdflayers Prompts you in the command bar:** Select PDF underlay: **(Click the frame of a PDF underlay)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU

Figure 6.1636



Command Options

Option	Description
Reference name	Selects a different PDF underlay that is in the current drawing.
Search for layer	Type a search string in the search field. Only layers of which part of the layer name matches the search string are listed. You can use wildcards (? or *) in the search field.
On	Click the column title to sort the layer list by display.
or	Click the icon to turn the display of a layer on/off.
Name	Click the column title to sort the layer list by name.

Procedures

- To set the layer display in a PDF underlay (1.22.21)

Related Commands

PdfAdjust (6.24.13) - Allows to adjust the fade, contrast and monochrome settings of a PDF-underlay in the command bar. pdfattach (6.24.14) - inserts a PDF file as an underlay into the current drawing. -PdfAttach (6.24.15) - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar. pdfclip (6.24.16) - clips a PDF underlay.

Figure 6.1637



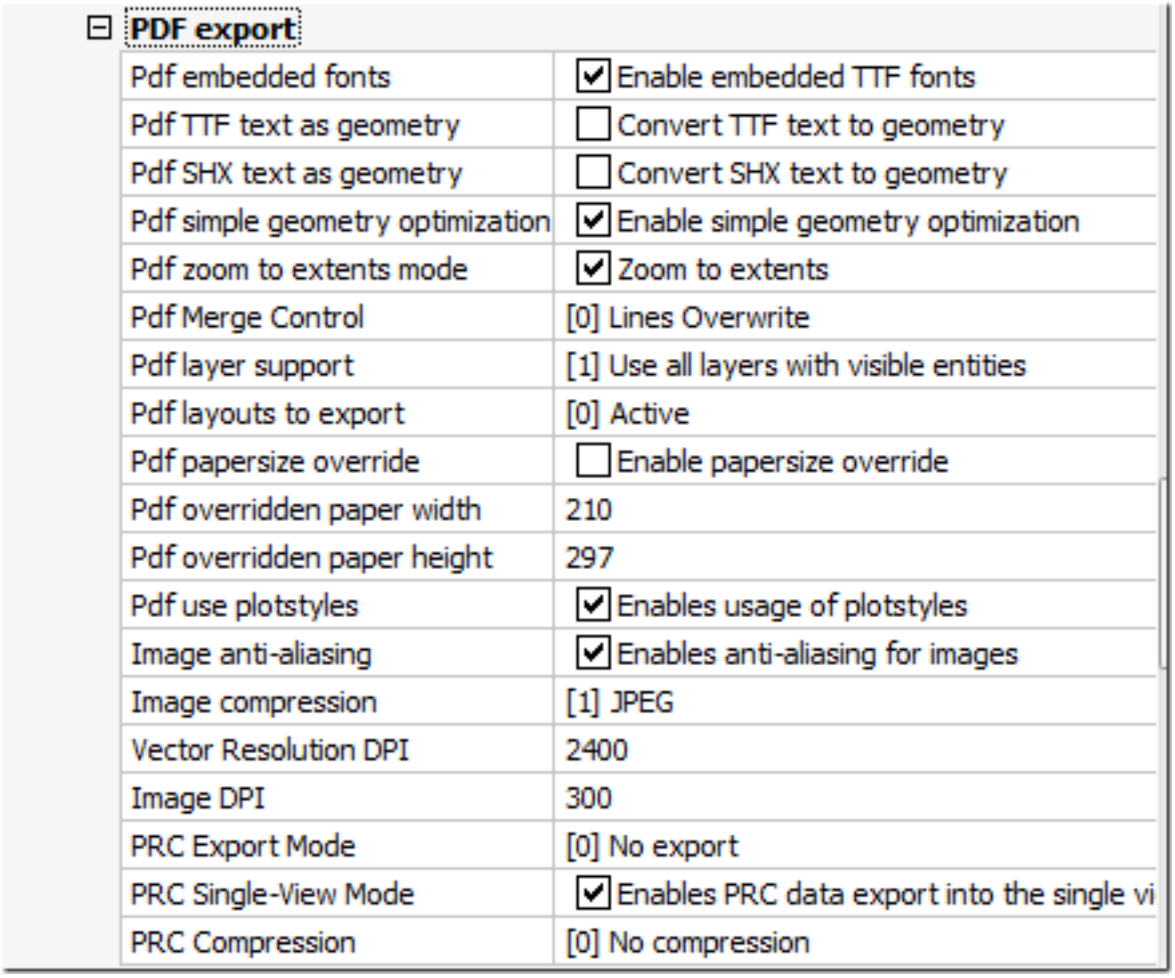
6.24.19 PdfOptions

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Determines how drawings are exported in format through the Settings dialog box.
Note: To export drawings in PDF format, use the ExportPdf (6.13.17) command.

Accessing the Command

command bar: **pdfoptions** transparent:**'pdfoptions** menu bar: **File | PDF Export Options** : **PdfOptions** Displays the PDF Export section of the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comma settings-dlg.png}) Change options, and then click the **X** to exit the dialog box.

Figure 6.1638



Command Options

Option	Description
Pdf embedded fonts	Determines whether TTF font files are embedded in the PDF file: • **Yes** - includes fonts, which may violate the copyright on some fonts. • **No** - does not include fonts, which may result in missing fonts if the receiving computer does not have the required SHX or TTF files.
Pdf TTF text as geometry	Determines whether TTF (true type fonts) are exported as geometry: • **Yes** - explodes fonts into lines and arcs. • **No** - does not explode fonts.
Pdf SHX text as geometry	Determines whether SHX (shapes compiled) are exported as geometry: • **Yes** - explodes font into lines and arcs. • **No** - does not explode fonts.
Pdf simple geometry optimization	Determines if the geometry is simplified to reduce file size: • **Yes** - optimizes geometry, such as turning multiple segments into a single polyline. • **No** - does not optimize geometry.
Pdf zoom to extents mode	Determines if the drawing is zoomed to extents: • **Yes** - zooms drawing to extents, which ensures the drawing matches the paper size. • **No** - does not zoom the drawing, which ensures the drawing is exported as-is.
Pdf Merge Control	Controls the appearance of crossing lines. • **Lines Overwrite** - the lines on top overwrite the lines below. See the Publish (6.24.55) dialog: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • **Lines Merge** - the colors of crossing lines are blended into a new color. See the Publish (6.24.55) dialog: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so
Pdf layer support	Determines how layers are exported: • **Don't use layers** - removes all layer information. • **Use layers/Don't include OFF-layers** - excludes layers that are turned off. • **Use layers/Include OFF-layers** - exports all layers. See Layer (6.20.2) command.

Pdf layouts to export	Determines which layouts to export: • **Active** - exports the current layout. • **All** - exports all layouts. Each layout is plotted to a separate page in the PDF file.
Pdf papersize override	Determines whether page size is specified: • **Yes** - uses the custom page size specified below • **No** - does not use specified page size
Pdf overridden paper width	Specifies the width of the page, in drawing units
Pdf overridden paper height	Specifies the height of the page, in drawing units
Pdf use plotstyles	Toggles the use of plot styles when exporting to PDF: • **Yes** - uses plot styles defined in the drawing for greater control over individual entities • **No** - does not use plot styles
Image Antialiasing	Toggles anti-aliasing for images that require upscaling when exporting: • **Yes** - use anti-aliasing to smooth images • **No** - does not use anti-aliasing
Image Compression	Toggles the use of compression for exported images: • **0** - no compression • **1** - compress the images using JPEG format
Vector Resolution DPI	Sets the resolution for vector graphics when exporting; values between 72 and 40000 are accepted.
Image DPI	Controls the minimum resolution for exported raster images. The value cannot exceed the value set for Vector Resolution DPI.
PRC Export Mode	Determines whether 3D models are exported in PRC format: • **0**- do export 3D models as 3D • **1**- export 3D models in b-rep (boundary representation) format • **2** - export 3D models in mesh format PRC options work only when the optional Communicator add-on is installed.
PRC Single-view Mode	Toggles whether the 3D model is exported in a single viewport: • **Yes** - yes • **No** - no
PRC Compression	Toggles the use of compression for 3D models: • **0** - use no compression • **1** - use medium compression • 2 - use high compression

Figure 6.1639



Figure 6.1640



Related Commands

Export (6.13.15) - exports drawings in PDF format. Plot (6.24.26) and Print (6.24.46) - export drawings to PDF files if the **Print As PDF.pc3** printer configuration is selected. Publish (6.24.55) - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.

Figure 6.1641



6.24.20 PEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Edits polylines, 3D polylines, and 3D meshes (short for "polyline edit"); converts 2D entities into polylines (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image437.png}) **Note:** This prompts of this command change, depending on the entity you are editing

- Single or multiple 2D polylines
- Single 3D polyline
- 3D mesh
- Single or multiple lines, arcs, circles, splines, or helixes

Accessing the Command

 command bar: **pedit** aliases: **editpline** **pe** menu: **Modify | Edit Polyline** toolbar: **Modify |**  mouse: Double-click a polyline : **PEdit Prompts you in the command bar:** Select polyline to edit [Multiple]: **(Choose one polyline or polyface mesh, or else pressM to select multiple polylines)**

- When a 2D polyline is selected, **you are prompted:**

Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/Spline/Taper/Width/Undo] <eXit>: **(Enter an option, or press Enter to exit the command)**

- When a 3D polyline is selected, **you are prompted:**

Edit polyline [Edit vertices/Close/Decurve/Join/Reverse/Spline/Undo]<eXit>: **(Enter an option, or press Enter to exit the command)**

- When a 3D mesh is selected, **you are prompted:**

Edit polyline: [Edit vertices/Desmooth/Mopen/Nclose/Undo]<eXit>: **(Enter an option, or press Enter to exit the command)**

- When a line, arc, circle or spline is selected, **you are prompted:**

The selected entity is not a polyline. Turn it into one? [Yes-turn into polyline/No-leave as is] <Yes-Turn into polyline>: **(Press Enter to turn the selected entity into a 2D polyline)** Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/Spline/Taper/Width/Undo] <eXit>: **(Enter an option, or press Enter to exit the command)**

- When the Multiple option is chosen, **you are prompted:**

Select entities: (**Select lines, polylines, arcs, circles, helixes, and/or splines)**

- When one or more lines, arcs, circles or spline are selected, **you are prompted:**

The selected entity is not a polyline. Turn it into one? [Yes-turn into polyline/No-leave as is] <Yes-Turn into polyline>: (**Press Enter to turn the selected entity into a 2D polyline)** Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/Spline/Taper/Width/Undo] <eXit>: **(Enter an option, or press Enter to exit the command)**

Command Options for 2D Polylines

2D Polyline Option	Description

Edit vertices	Edit vertices; prompts you: **Edit vertices: Next/ Previous/ Angle/ Break/ Insert/ Move/ Regen/ SElect/ Straighten/ Width/eXit/ <Next>** - **(enter an option.)** BricsCAD displays a triangle marker at the start of the polyline, and an X marker at the current vertex. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • Enter **Next** or **Previous** to move the marker from vertex to vertex • Enter the **SElect** option to select the vertex directly **Note:** It is much easier to edit the vertices of a polyline with grips editing.
Next	Moves the X marker to the next vertex.When the marker reaches the last vertex of a closed polyline, it does not move to the ”next” (first) vertex.
Previous	Moves the X marker to the previous vertex.
Angle	Changes the angle of the current segment; prompts you: **Included angle for segment (>0 is ccw, 0 is straight, <0 is cw) <0>** - **(enter an angle)**: • If it is an arc segment, then the angle (curvature) of the arc is changed • If it is a line segment, it is turned into an arc • Press**0**(zero degrees) to convert the arc segment into a line segment (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Break	Removes segments from the polyline between the two vertices marked with X markers. This action turns: • Closed polylines into open polylines • Open polylines into two polylines, with a gap between them This action cannot be undone. Prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Next/Previous/Select/Go/eXit/<Next>** - **(enter an option to move the X marker along the polyline)**: • **Next** - choose the next vertex • **Previous** - choose the previous vertex • **Select** - moves second X marker directly to another vertex; prompts you: **Select point along segment move to** - **(pick a point on the polyline. The programplaces the second X marker on the vertex nearest to the pick point; this option is faster than using Next and Previous.)** • **Go** - break the polyline between the first and second marked vertices (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **eXit** - return to the previous prompt **Note:** It is much easier to use the Break (6.6.18) or Trim (6.29.26) commands to remove part of a polyline.
Insert	Inserts a vertex. A drag line previews the location of the new vertex; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Location of vertex** - pick a point for the new vertex. The programadds a vertex and a segment: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Move	Moves the current vertex. Ahe drag line previews the new location of the vertex. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Prompts you: **New location for vertex** - **(pick point to move the vertex. The programmoves the vertex and associated segments.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Note:** It is much easier to move the vertices of a polyline with grips editing.
Regen	Regenerates the polyline to show changes made by the editing actions of this command.
SElect	Moves the X marker directly to another vertex; prompts you: **Select point along segment to edit** - **(pick a point on the polyline; the programplaces the X marker on the vertex nearest to the pick point.This option is faster than using the Next and Previous options.)**

Straighten	Removes all segments between two vertices. When you select the start and end points of an multi-segment polyline, the entire polyline is straightened, becoming a single segment; this also applies to closed polylines. Prompts you: **Straighten: Next/Previous/Select/Go/eXit** - **(enter an option.)** • **Next** -moves the second X marker to the next vertex • **Previous** -moves the second X marker to the previous vertex • **Select** - moves second X marker directly to another vertex; prompts you: **Select point along segment move to** - pick a point on the polyline • **Go** - replaces the segments between the two marked vertices with a single segment: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou This action cannot be undone. • **eXit** -exits this sub-sub-option; returns to the previous prompt
Width	Changes the width of the current segment; prompts you: 1. **Enter new starting width** - **(specifies the width at the starting end of the polyline.)** 2. **Enter new ending width** - **(specifies the width at the polyline's end.)** The "current" segment lays between the X-marked vertex and the next vertex.
eXit	Exits this sub-option; returns to the previous prompt
Close	Closes the polyline by adding a segment between the start and endpoints. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i When the polyline is closed, this option reads **Open**.
Open	Opens closed polylines by removing the last segment drawn to close the polyline. When the polyline is open, this option reads **Close**.
Decurve	Reverses the effects the Fit and Spline options.
Fit	Fits a curve to the polyline. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i To unfit the polyline, use the **Decurve** option.

Join	Adds open entities to the selected polyline; prompts you: **Select entities to join**- **(choose an entity)**: • Line • Arc • Polyline segment and arc • Helix This option works only with open and with cojoined entities.**Cojoined** means that the endpoints of entities meet, such as when drawn with ENDpoint object snap or with the Last point option. The joined entities take on the source polyline's properties, such as color, width, and layer. When editing multiple polylines,prompts you: **Enter fuzz distance or [Jointype]** - **(do one of the following)**: • Press **Enter** to create one or more polylines with cojoined lines, polylines, and arcs in the selection set. • Enter a so-called 'fuzz distance' then press **Enter** to create one or more polylines from the entities in the selection set. When endpoints are within the fuzz distance, the entities are extended or trimmed. • Type **J**to specify the type of joining; prompts you: **Specify type of joining: [Extend/Add/Both]** - **(enter an option to specify what happens when entities lie within the fuzz distance)**: **Extend** - extends or trims segments to the nearest endpoints **Add** - adds linear segments between the nearest endpoints **Both** - extends or trims if possible, otherwise adds linear segments The program then reports the number of polylines created.
Linetype-mode	Determines how linetype appear on polylines; prompts you: **Linetype continuous along polyline: ON/OFF <OFF>** - **(enter an option)**: • **ON** - linetypes start and stop at the polyline's start and endpoints • **OFF** - linetypes start and stop at each vertex This option is stored in the PLineGen variable.
Reverse	Reverses the direction of the polyline: the starting point is now its end point, and vice versa.The polyline does not change its look with this operation.This option affects operations that depend on the direction of a polyline, such as vertex editing.The polyline does not change its look with this operation, except that the triangle marker moves to the other end of open polylines.
Spline	Converts the polyline into a spline. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i All width information is lost; use the **Width** option to reapply width.The spline is a Bezier-spline whose smoothing is defined by the SplineType system variable.

Taper	Tapers the width of the entire polyline from one end to the other; prompts you: • **Enter new starting polyline width** - **(specify the width at the starting end of the polyline.)** • **Enter new ending polyline width** - **(specify the width at the polyline's end.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Width	Changes the width of all segments; prompts you: **Enter new width for all segments** - **(specify a width.)** This option overrides tapered widths. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Undo	Undoes the last action.
eXit	Exits the command.

Command Options for 3D Polylines

3D Polyline Option	Description
Edit vertices	Edit vertices; prompts you: **Edit vertices: Next/ Previous/ Break/ Insert/ Move/ Regen/ SElect/ Straighten/ eXit/<Next>** - **(see the options for 2D polylines, above.)** The Angle and Width options are missing from the prompt, because 3D polylines cannot contain arc segments and cannot have width.

Command Options for 3D Polyface Meshes

3D Polyface Mesh Option	Description
Edit vertices	Moves vertices; prompts you: **Edit vertex (0,0): Select/Next/Previous/Left/RIght/Up/Down/Move/REgen/eXit** - **(enter an option.BricsCAD displays a triangle marker on the home vertex (0,0) and an X marker on the current vertex.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i It is easier to move vertices using grips.
Select	Moves X marker directly to another vertex; prompts you: **Select point along segment to edit** - **(pick a point on the mesh.)** The program places the X marker on the vertex nearest to the pick point.This option is faster than using the **Next** and **Previous** options.
Next	Moves the X marker to the next vertex.When the marker reaches the last vertex, it does not move to the ”next’ (first) vertex.
Previous	Moves the X marker to the previous vertex.
Left	Moves the X marker to the vertex on the left.
RIght	Moves the X marker to the vertex on the right.
Up	Moves the X marker to the higher-numbered vertex.
Down	Moves the X marker to the lower-numbered vertex.
Move	Moves the current vertex; prompts you: **New location for vertex** - **(pick a point to move the vertex.The drag line previews the new location of the vertex.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i BricsCAD moves the vertex and associated faces.
REgen	Regenerates the polyline to show changes made by the editing actions of this command.

eXit	Exits this sub-option; returns to the previous prompt line.
Desmooth	Removes the Bezier-spline smoothing of 3D meshes.
Mclose	Closes the 3D mesh in the m direction: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i When the mesh is closed, this option reads MOpen.
Mopen	Opens the 3D mesh in the m direction.When the mesh is closed, this option reads MClose.
Nclose	Closes the 3D mesh in the n direction: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i When the mesh is closed, this option reads NOpen.
Nopen	Open the 3D mesh in the n direction.When the mesh is closed, this option reads NClose.
Undo	Undoes the last action.
eXit	Exits the command.

Procedures

- Converting an entity into a polyline (1.23.29)
- To fit a curve to a polyline (1.23.32)
- Convert a straight polyline segment into an arc (1.23.33)
- Break a polyline into two separate polylines (1.23.33)
- Insert a new vertex in a polyline (1.23.33)
- Move a vertex in a polyline (1.23.33)
- Delete vertices in a polyline (1.23.33)
- Change the width of a polyline segment (1.23.33)
- To apply a uniform width to a polyline (1.23.35)
- To taper a polyline uniformly along its length (1.23.35)
- To join an arc, line, or polyline to an existing polyline (1.23.31)
- Opening and closing polylines (1.23.30)
- Setting the Linetype generation mode (1.23.34)

Related Commands

3dPoly (6.3.10) - draws 3D polylines. 3dFace (6.3.5) - draws 3D 4-edged faces with optional invisible edges. 3dMesh (6.3.7) - draws 3D surface meshes of size m x n. ConvertPoly (6.9.45) - converts lightweight polylines to classic polylines (2D polylines) and vice versa. Join (6.18.1) - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints. PeditExt (6.24.21) - Edits vertices and segments of a polyline. PFace (6.24.23) - draws 3D multi-sided meshes. PLine (6.24.27) - draws polylines. Splinedit (6.27.40) - modifies the properties of a spline or converses a spline to a polyline.

Figure 6.1642

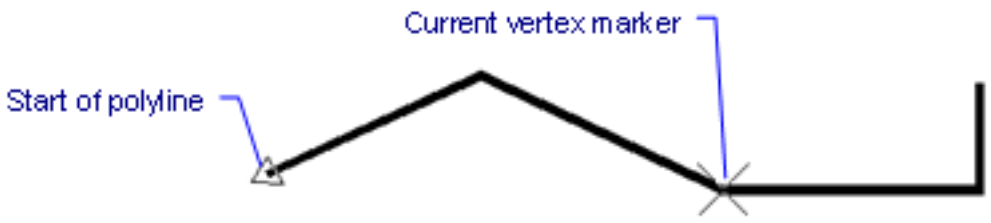


Figure 6.1643

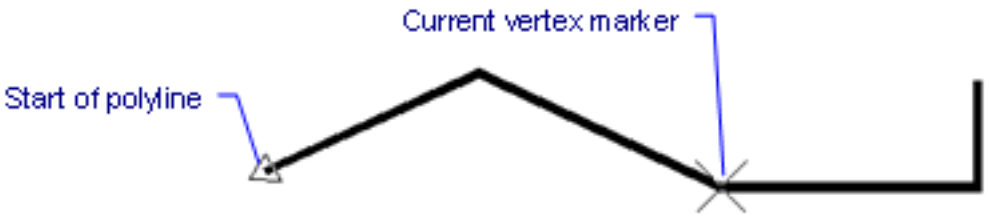


Figure 6.1644

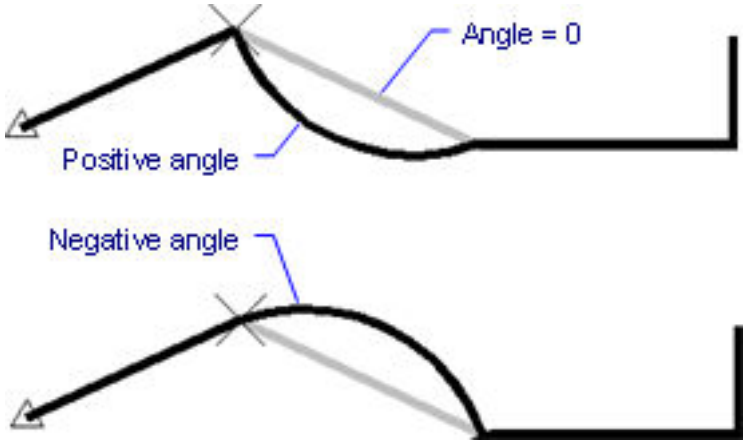


Figure 6.1645

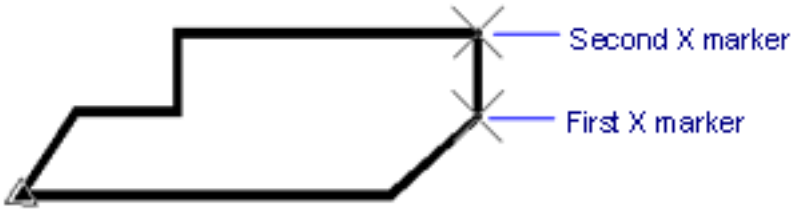


Figure 6.1646

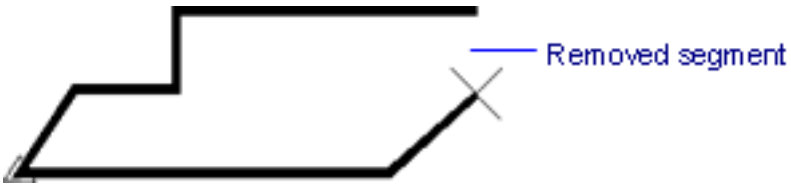


Figure 6.1647

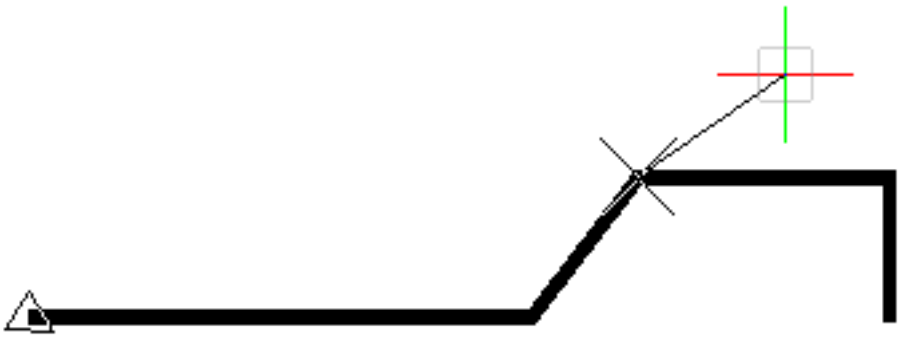


Figure 6.1648

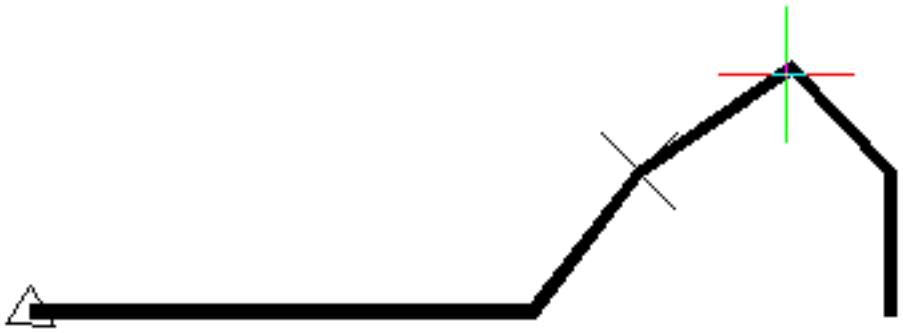


Figure 6.1649

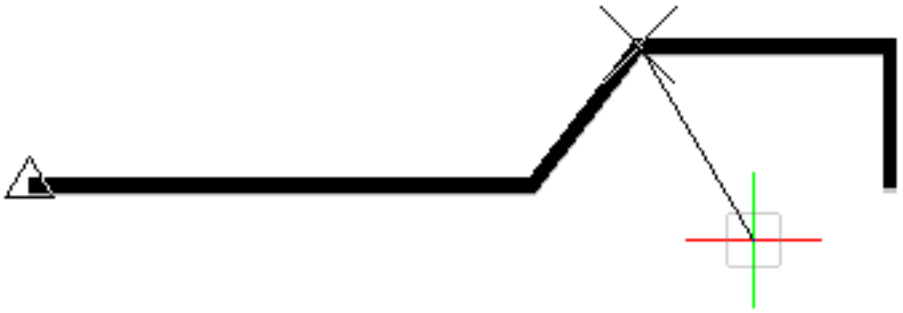


Figure 6.1650

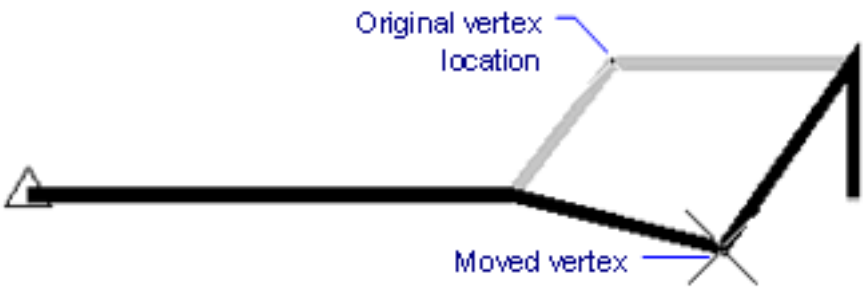


Figure 6.1651

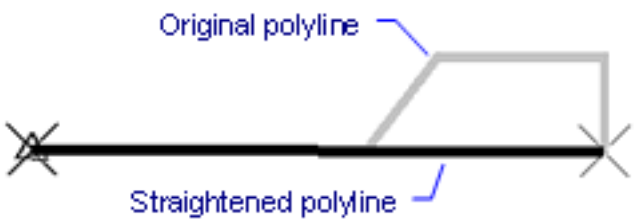


Figure 6.1652

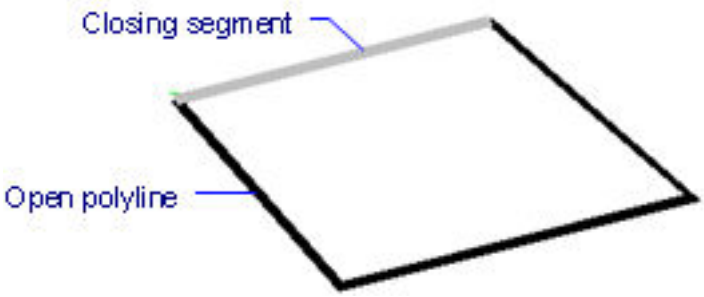


Figure 6.1653



Figure 6.1654

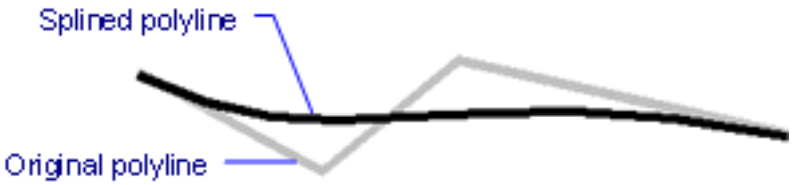


Figure 6.1655

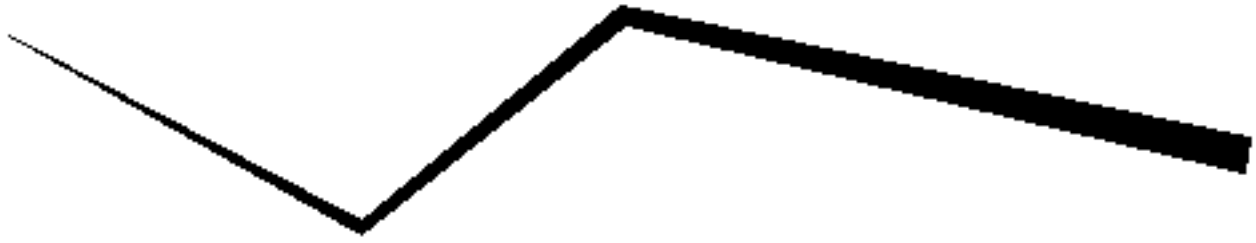


Figure 6.1656



Figure 6.1657

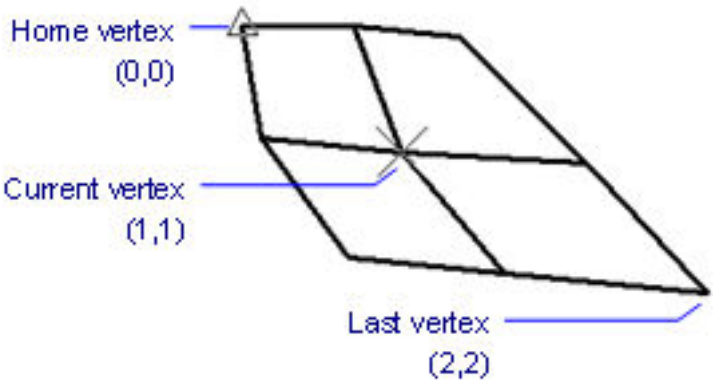


Figure 6.1658

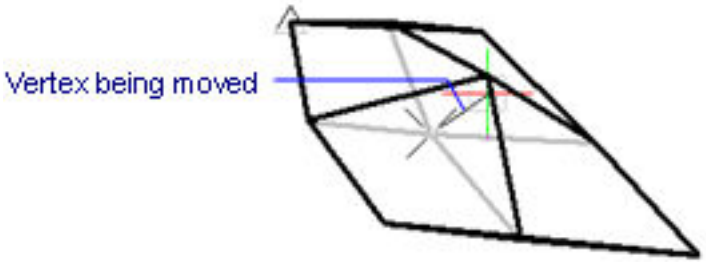


Figure 6.1659

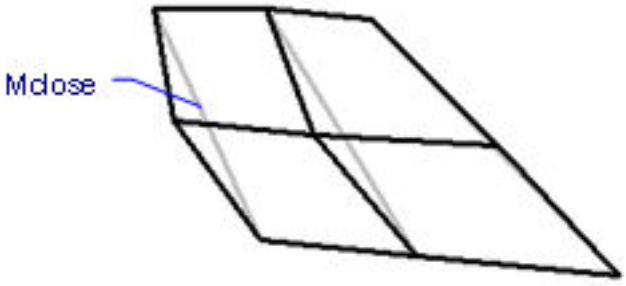


Figure 6.1660

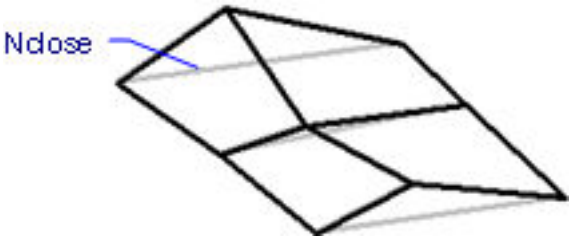


Figure 6.1661



6.24.21 PeditExt

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Edits its vertices and segments of a polyline interactively, without first selecting the polyline. **Note:** This command was meant for use with the Quad cursor.

Accessing the Command

command bar: **peditext** Quad cursor menu: **2D Editing** |  : **PeditExt Prompts you in the command bar:** Select option [Add vertex/add vertex at End/adjust Bulge/Remove segment/Stretch segment/stretch Vertex]: (S**elect an option)**

Command Options

Option	**Description - command line**
Add vertex	Adds a vertex to the selected segment; prompts you: 1. **Select the polyline segment to add a vertex to** - (**select a polyline segment. The new vertex displays dynamically)**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/add-eg.png}) 2. **Select a point** - **(specify the location of the new vertex.)**  Quad icon ✚ adds a vertex to the segment under the cursor.
add vertex at End	Adds a new vertex at the end of the polyline; prompts you: • **Select the polyline near the end you wish to add a vertex** - **(click a polyline near the end point.The new vertex displays dynamically)**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/end-eg.png}) • **Select a point** - **(specify the location of the new vertex.)**  Quad icon ✚ adds a new vertex at the end of the polyline under the cursor.

adjust Bulge	Modifies the bulge factor of the selected segment; prompts you: 1. **Select the polyline segment to adjust bulge** - **(select a polyline segment.)** The bulge of the selected segment changes dynamically. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/buldge-eg.png}) 2. **Select a point** - **(specify a point to set the new bulge.)** Quad icon  modifies the bulge factor of the segment under the cursor. Bulge factor (1.23.38)defines the shape of a polyline segment: • Straight bulge = 0 • Counter-clockwise arc bulge > 0 • Clockwise arc bulge bulge < 0
Remove segment	Deletes the selected segment; prompts you: **Select the polyline segment to remove** - **(select a polyline segment.)** Quad icon  deletes the segment under the cursor.
Stretch segment	Moves the selected segment; prompts you: 1. **Select the polyline segment to stretch** - **(select a polyline segment.The selected segment moves dynamically; the adjacent segments are stretched)**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/stertch-eg.png}) 2. **Select a point** - **(specify the location of the segment.)** Quad icon  moves the segment under the cursor.
stretch Vertex	Moves the selected vertex; prompts you: 1. **Select the polyline vertex to stretch** - **(select a polyline vertex.The selected vertex moves dynamically)**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/vertex-eg.png}) 2. **Select a point** - **(specify the location of the vertex.)** Quad icon  moves the vertex under the cursor.

Procedures

- Editing a polyline using the Quad (1.23.33)

Related Commands

PEdit (6.24.20) - edits polylines, 3D polylines, and 3D meshes.

Figure 6.1662



Figure 6.1663

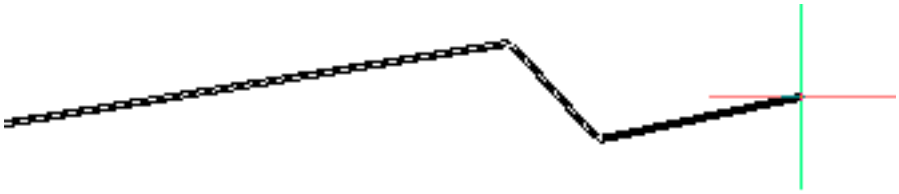


Figure 6.1664

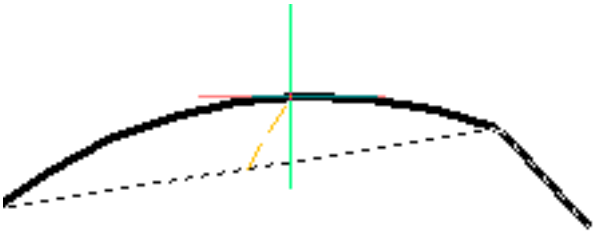


Figure 6.1665

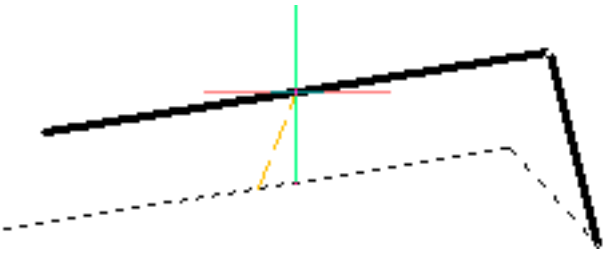


Figure 6.1666

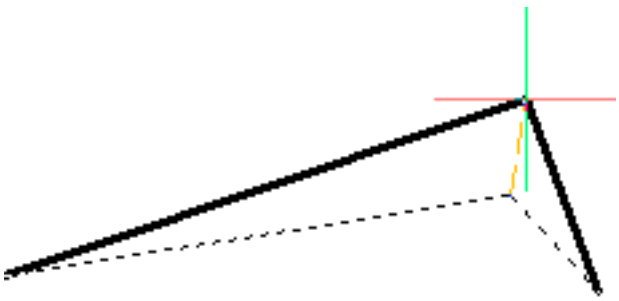
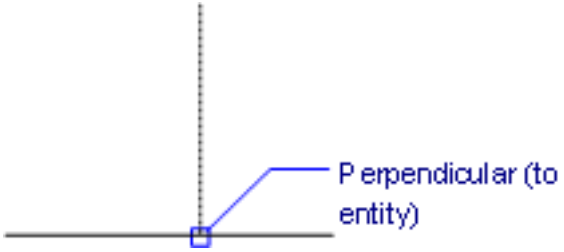


Figure 6.1667



Figure 6.1668



6.24.22 Perpendicular

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles perpendicular entity snap mode. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco

Accessing the Command

 command bar: **perpendicular** transparent: **per** ribbon:**Home | Entity Snaps | Snap to Perpendicular** toolbar: **Entity Snaps** | : **Perpendicular Displays no prompt in the command bar.** When perpendicular entity snap is on, the cursor snaps to points of perpendicularity to entities.

Command Options

Option	Description
Perpendicular	Entering this command toggles Perpendicular entity snap. It snaps to points of perpendicularity. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

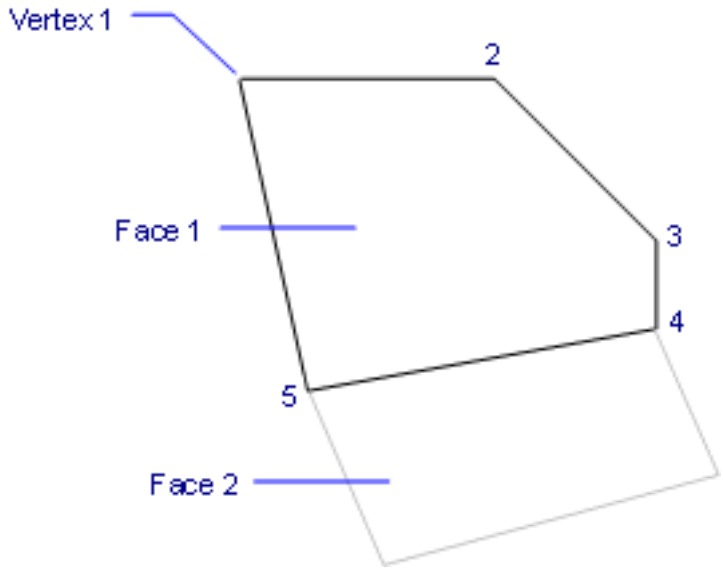
Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Quadrant (6.25.7) - toggles quadrant snap mode. Tangent (6.29.9) - toggles tangent snap mode. Zperpendicular (6.35.8) - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.

Figure 6.1669



Figure 6.1670



6.24.23 PFace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws 3D multi-sided meshes (short for "polyface mesh"). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image353.png})

Accessing the Command



command bar: **pface** menu: **Model | Meshes | Polyface Mesh** ribbon:**Model | Meshes | Polyface Mesh** toolbar: **Meshes | 3D Faces |**  : **PFace Prompts you in the command bar:** Position of mesh vertex 1: **(Pick a point, or enter x,y,z coordinates.)** Position of mesh vertex 2/<Enter to define faces>: **(Pick a point, or enter x,y,z coordinates.)** Position of mesh vertex 3/<Enter to define faces>: **(Pick a point, enter x,y,z coordinates, or press Enter to stop placing vertices.)** Face 1, Vertex 1: Color/Layer/<Exit>: **(Enter a number or an option)** Face 1, Vertex 2: Color/Layer/<Exit>: **(Enter a number or an option.)** The prompt continue until all faces and vertices are dealt with.

Command Options

Option	Description
Position of mesh vertex 1:	Specifies the first vertex of the face.A face can have up to 256 sides.
Position of mesh vertex n	Specifies the next vertex of the face.
Color	Specifies the color of the edge; see Color (6.9.33) command.
Layer	Specifies the layer of the edge; see Layer (6.20.2) command.
Face n , vertex n	Specifies the face and vertex number.Enter a negative number to make the edge invisible.

Related Commands

3dFace (6.3.5) - draws 3D 4-edged faces with optional invisible edges. 3dMesh (6.3.7) - draws 3D surface meshes of size m x n. PEdit (6.24.20) - edits 3D meshes.

Figure 6.1671



6.24.24 PlaceView

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places named views from a source drawing into a paper space layout of the current drawing. **Note:** This command works only in paper space, and is meant for use with sheet sets.

Accessing the Command

command bar: **placeview** mouse: drag a model view from the Content Browser (6.9.41) panel onto a paper space layout **: PlaceView**

- When you start the command in model place, **the program reports:**

Command only valid in Paper Space.

- When you start the command in paper space, **the command prompts you:**

Enter file name: **(Type the drawing name, including the full path)** Enter view name to place or [?]: **(Type the name of the model view, or enter ? for a list of view names)** Enter view origin point: **(Specify a point)** The drawing is attached as an external reference and a paper space viewport is placed in the layout showing the named view.

Command Options

Option	Description
Enter file name	Names the file from which the views will be place.
Enter view name to place	Names the view to place.
?	Prints a list of the named views in the Prompt History (6.29.13) window, such as: Start of Road Hairpin Curve End of Road
Enter view origin point	Specify the location of the lower left corner: • Pick a point • Enter x,y coordinates

Related Commands

ContentBrowserOpen (6.9.41) - opens the Content Browser panel. XAttach (6.34.1) - attaches externally-referenced drawings through a dialog box. XRef (6.34.8) - attaches DWG files to the current drawing through the Drawing Explorer .

6.24.25 Plan

Displays the plan viewpoint of drawings, which looks straight down on the x,y-plane.

Accessing the Command

command bar: **plan** menu bar: **View | Plan View** ribbon: **View | View | Viewpoint | Plan View** toolbar: **View |** : **Plan Prompts you in the command bar:** Plan view of [Current/Ucs/World] <Current UCS>: **(Enter an option, or else press Enter)** Left: 3D model in 3D viewpoint; right: 3D model in plan view. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image356.png}) (\vref {fig:C:/Dev/Task-2448/images/static

Figure 6.1672

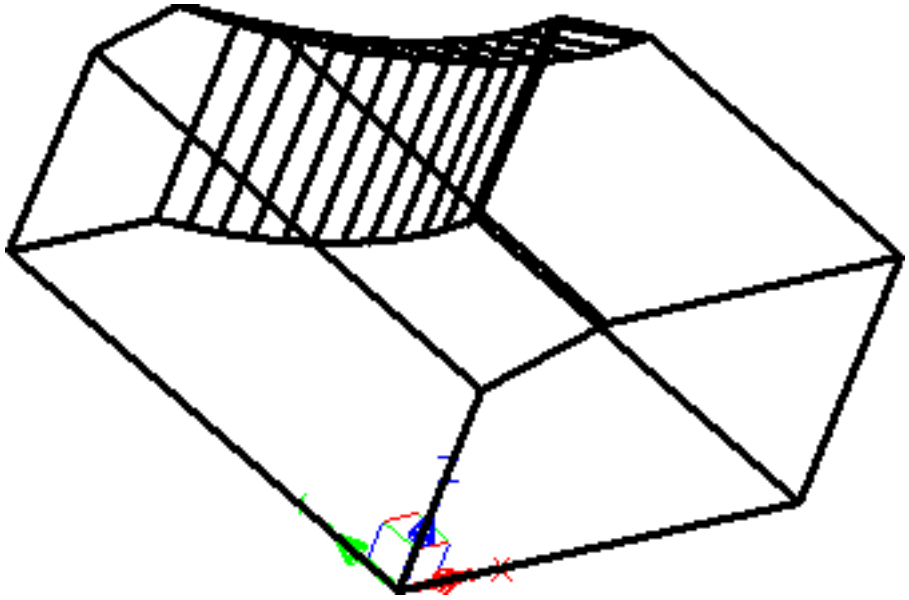
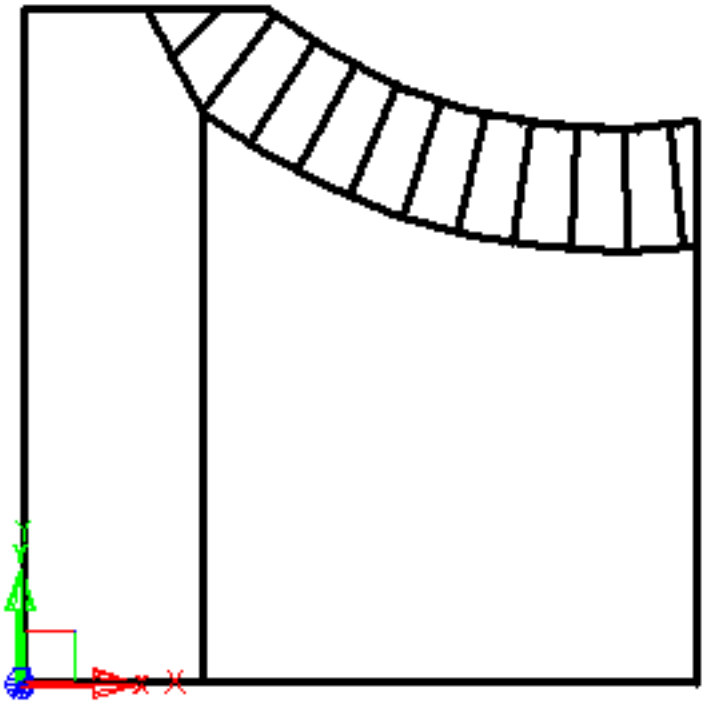


Figure 6.1673



Command Options

Option	Description
UCS	Displays the plan view of a named UCS; prompts you: **? to list/<Name of UCS>** - **(specify the name of the UCS or enter ?.)**
?	Lists the names of UCSs in the current drawing
World	Displays the plan view of the World Coordinate System.
Current	Displays the plan view of the current UCS.

Related Commands

SetUcs (6.27.25) - sets the UCS to a specified viewpoint. UcsIcon (6.30.3) - controls the appearance of the UCS icon. Ucs (6.30.2) - creates and modifies user-defined coordinate systems. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Settings (6.27.23) - sets parameters of UCSs

Figure 6.1674



6.24.26 Plot

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Plots drawings to printers and to files, through prompts at the command bar; this command is meant for scripts and routines. **Note:** To print drawings through a dialog box, use the Print (6.24.46)command.

Accessing the Command

command bar: **plot** alias: **-plot** : **Plot Prompts you in the command bar:** Detailed plot configuration [Yes/No] <No>: **(Type Y or N)**

- When you don't want detailed plot configuration, **you are prompted:**

Enter a layout name or ? <Model>: **(Enter a name, type ? for a list of layout names, or press Enter to accept the default)** Enter a page setup name <>: **(Enter a name, press Enter for none, or type _previous plot or <previous plot>)** Enter an output device name or ? <default.pc3>: **(Enter a name, type ? for a list of device names, or press Enter to accept the default)** Write the plot to a file [Yes/No] <No>: **(Type Y or N)** Save changes to model tab [Yes/No] <No>: **(Type Y or N)** Proceed with Plot [Yes/No] <No>: **(Type Y or N)**

- When you want detailed plot configuration, **you are prompted:**

Enter a layout name or ? <Model>: **(Enter a name, type ? for a list of layout names, or press Enter to accept the default)** Enter an output device name or ? <dfault.pc3>: (**Enter a name, type ? for a list of device names, or press Enter to accept the default)** Enter paper size or ? <Letter>: **(Enter a name, type ? for a list of size names, or press Enter to accept the default)** Enter paper units [Inches/Millimeters] <Millimeters>: **(Type I or M)** Enter drawing orientation [Portrait/Landscape] <Portrait>: **(Type P or L)** Plot upside down [Yes/No] <No>: **(Type Y or N)** Enter plot area [Display/Extents/Limits/View/Window] <Display>: **(Enter an option)** Enter plot scale (Plotted millimeters = Drawing Units) or [Fit] <Fit>: **(Enter a scale factor or type F)** Enter plot offset (x,y) or <0.0000,0.0000>: **(Enter a x,y offset distance, or press Enter to accept the default)** Plot with plot styles [Yes/No] <Yes>: **(Type Y or N)** Enter plot style table name (enter . for none) or ? <bricscad.stb>: **(Enter a name, type ? for a list of style names, or press Enter to accept the default).** Plot with lineweights [Yes/No] <Yes>: **(Type Y or N)** Remove hidden lines [Yes/No] <Yes>: **(Type Y or N)** Write the plot to a file [Yes/No] <Yes>: **(Type Y or N)** Save changes to model tab [Yes/No] <Yes>: **(Type Y or N)** Proceed with Plot [Yes/No] <Yes>: **(Type Y or N)**

Command Options

Option	Description
Detailed plot configuration	Toggles detailed plot configuration: • **Yes** - prompts you with 16 options; see below • **No** - prompts you with six options
Enter a layout name	Specifies the name of the layout to plot. • Enter the name of a layout, such as layout1 • **?** - lists the names of layouts in the drawing. • **Model** - plots the model tab only.
Enter a page setup name	Specifies the name of a page setup: • Enter a name • Press **Enter** for none See thePageSetup (6.24.1)command.

Enter an output device name	Specifies the name of the output device (plotter or file): • Enter a name of a PC3 file • **?** - lists the names of devices available on this computer, such as: ”Adobe PDF” ”Default Windows System Printer.pc3” ”Print As PDF.pc3” When the default is None, you must specify a name.
Write the plot to a file	Determines whether the plot is sent to a file: • **Yes** - plots the drawing to a PLT file; the program displays the Create Plot File dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so a file name, and then click **Save**. • **No** - plots the drawing to a printer or other output device.
Save changes to model tab or Save changes to layout-name	Toggles saving the plot parameters to the model or layout tab: • **Yes** - saves the plot parameters, which can be used for the next plot • **No** - does not save plot parameters
Proceed with Plot	Toggles plotting of the drawing: • **Yes** - plots the drawing • **No** - does not plot the drawing; exits the command

Detailed Configuration Option	**Description**
Enter a layout name	Specifies the name of the layout to plot. • Enter the name of a layout, such as layout1 • **?** - lists the names of layouts in the drawing. • **Model** - plots the model tab only. model tab only.
Enter an output device name	Specifies the name of the output device (plotter or file): • Enter a name of a PC3 file • **?** - lists the names of devices available on this computer, such as: ”Adobe PDF” ”Default Windows System Printer.pc3” ”Print As PDF.pc3” When the default is None, you must specify a name.
Enter paper size	Specifies the name of the paper size: • Enter the name of a paper size • **?** - lists the names of paper sizes available for the plotter, such as: Available media sizes. ”Letter” ”Legal” ”Executive” ”A3” The available paper sizes vary according to the printer selected.

Enter paper units

Specifies the units used for paper sizes:

- **Inches** - shows paper sizes in inches
- **Millimeters** - shows paper sizes in millimeters

Enter drawing orientation

Determines how the drawing is plotted on the paper:

- **Portrait** - prints the drawing so that the tall edge is upright
- **Landscape** - prints the drawing so that the short edge is upright

Plot upside down

Toggles upside-down plotting:

- **Yes** - prints drawing upside-down
- **No** - prints drawing normally

Enter plot area

Specifies the area of the drawing to plot:

- **Display** - plots the area shown in the current viewport.
- **Extents** - plots the extents of the drawing, ensuring all non-frozen entities are plotted.
- **Limits** - plots the limits of the drawing, as defined by the Limits command.
- **View** - plots the current view, or a named view. See View command; prompts you: **Enter view name** - **(specify the name of a view.)**
- **Window** - plots a rectangular area defined by the user; prompts you:

1. **Select first corner of window** - **(specify one corner of the rectangle.)**
2. **Select second corner of window** - **(specify the other corner.)**

Enter plot scale

Specifies the plot scale factor:

- **Plotted Inches (or Millimeters)** - specifies the inches (or millimeters) on the paper.
- **Drawing Units** - specifies the number of drawing units (inches, meters, etc).
- **F** - fits the drawing to the paper; BricsCAD calculates the scale factor automatically.

Enter plot offset

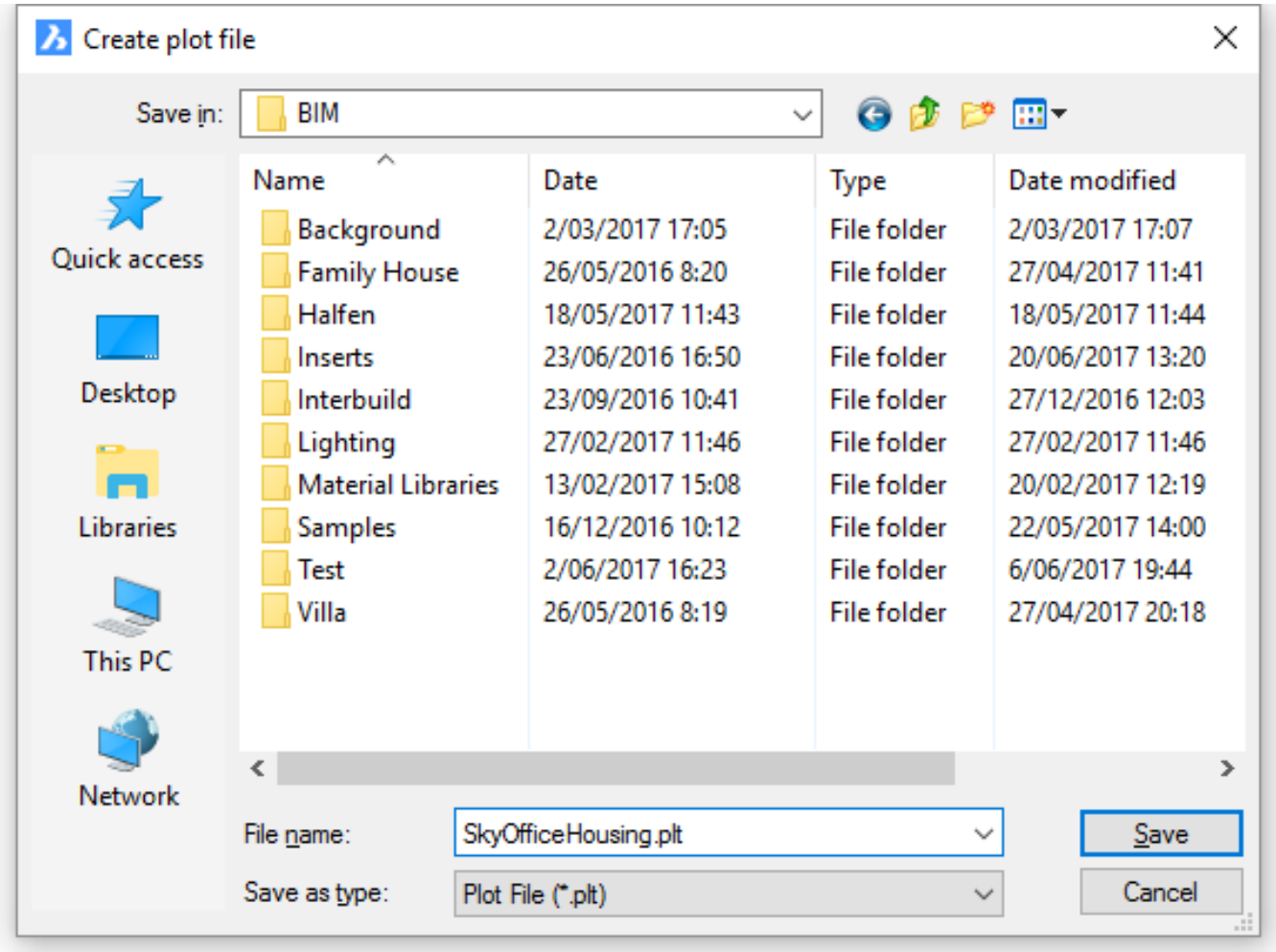
Specifies the offset distance for the print; enter positive or negative distances to move the drawing in the x and/or y distances:

- **Positive values** - moves the drawing up and to the right
- **Negative values -** moves the drawing down and to the left

The lower left corner of the drawing is moved by the specified distance.This is useful when the paper has a title block area that might interfere with the drawing.

Plot with plot styles	Toggles the use of plot styles: • **On** - uses plot styles to determine the look of the printed drawing. • **Off** - does not use plot styles. Plot styles override lineweight settings.
Enter plot style table name	Specifies the plot style table to use, which assigns properties to ”pens,” colors, and entities. • Enter a STB file name • **?** - lists the names of plot styles available in the drawing. • **.** - specifies no plot style.
Plot with lineweights	(This option is unavailable when Plot with Plot Styles is turned on.) Toggles the use the lineweights: • **On** - mimics lineweights assigned to the drawing in the print • **Off** - ignores lineweights
Remove hidden lines	Toggles removal of hidden lines for 3D drawings: • **Yes** - removes hidden lines • **No** - does not remove hidden lines
Write the plot to a file	Determines whether the plot is sent to a file: • **Yes** - plots the drawing to a PLT file; the program displays the Create Plot File dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so a file name, and then click **Save**. • **No** - plots the drawing to a printer or other output device.
Save changes to model tab or Save changes to layout-name	Toggles saving the plot parameters to the model or layout tab: • **Yes** - saves the plot parameters, which can be used for the next plot • **No** - does not save plot parameters
Proceed with Plot	Toggles plotting of the drawing: • **Yes** - plots the drawing • **No** - does not plot the drawing; exits the command

Figure 6.1675



Related Commands

Print (6.24.46) - plots drawings through a dialog box. QPrint (6.25.3) - plots drawings using the current settings. Preview (6.24.45) - previews drawings before they are plotted. ConvertCtb (6.9.42) - converts CTB plot style files to STB files. ConvertPStyles (6.9.46) - converts drawings from CTB to STB plot styles. Layer (6.20.2) - assigns plot styles to layers. PageSetup (6.24.1) - assigns plot style to drawings. ExportPdf (6.13.17) - exports layouts in a drawing to a pdf docucment. PlotStyle (6.24.29) - sets the current plot style. StylesManager (6.27.55) - creates and attaches plot styles to drawings. Publish (6.24.55) - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file. -Publish (6.24.56) - prints a saved sheet list (= a list of model space or paper space layouts).

Figure 6.1676

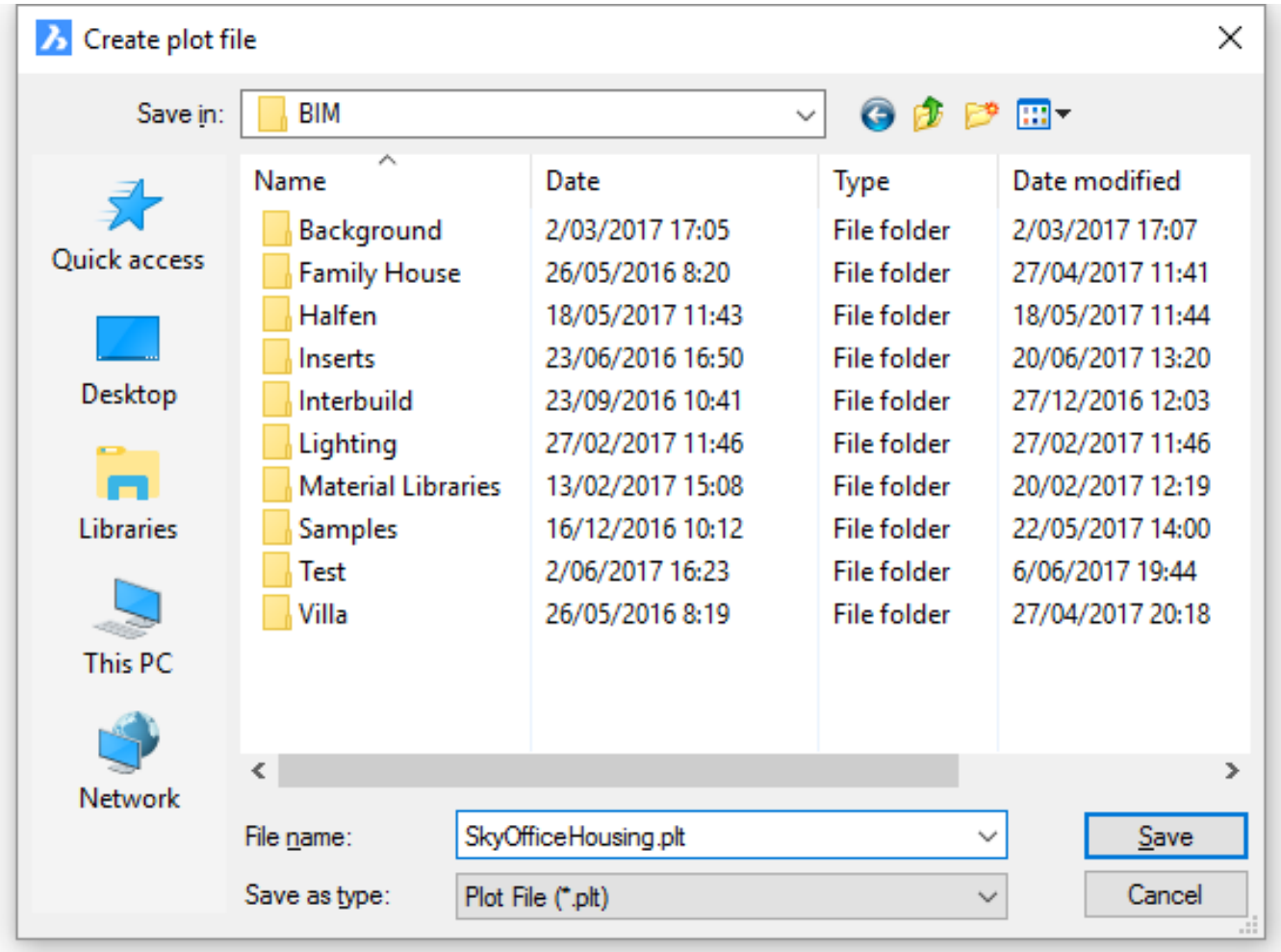
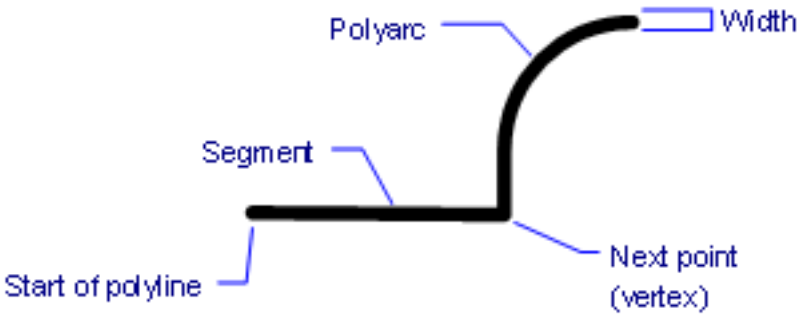


Figure 6.1677



Figure 6.1678



6.24.27 PLine

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws polylines (short for "polyline"). **Note:** A polyline is a multi-segment entity that can contain lines, arcs, and have segments of variable width, as well as be edited to have curves and splines. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

Accessing the Command


command bar: **pline** aliases: **pl** **polyline** menu bar: **Draw | Polyline** ribbon:**Draw | Draw | Polylines** toolbar: **Draw |**  : **PLine**

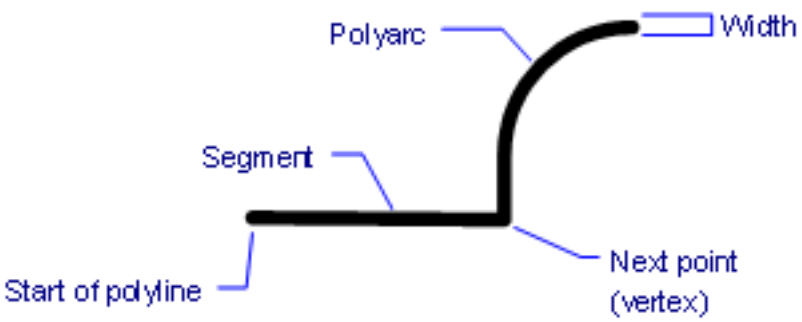
- The first time the command is used in a drawing, **prompts you in the command bar:**

Start of polyline: **(Pick a point)**

- **Subsequent times the the first prompt changes to**

Select start of polyline or [Follow] <Last point>:**(Pick a point or enter an option)** Set next point or [draw Arcs/Distance/Follow/Ha **(Pick a point, or enter an option)** Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width/Undo]: **(Pick a point, enter an option, or press Enter to exi.)** Set next point or [draw Arcs/Close/Distance/Follow/Halfwidth/Width/Undo]: **(Pick a point, enter an option, or press Enter to exit)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/C

Figure 6.1679



Command Options

Option	Description
Start of polyline	Specifies the start point of the polyline's first segment: • Enter x,y,z coordinates • Pick a point in the drawing
Last point	Starts the polyline at the last point picked in the drawing, which is useful for continuing from an existing polyline.
Next point	Specifies the next point (endpoint for the current segment); each next-point becomes a vertex in the polyline.
Arc	Switches to arc-drawing mode; draws arc segments; prompts you: **Set end of arc or [draw Lines/ Angle/ CEnter/ CLose/ Direction/ Halfwidth/ Radius/Second point/Width/Undo]** - **(pick a point or choose an option.)** • **draw Lines** - switches to line-drawing mode; draws line segments. • **Angle** - sets the included angle of the arc segment; prompts you: **Included angle for segment (+ for ccw, - for cw)** - **(enter an angle.) Set end of arc or [Center/Radius]** - **(pick point or choose an option.)** • **CEnter** - sets the center of the arc segment; prompts you: **Center point** - **(pick a point.) Set end of arc or [Angle/Length of chord]** - **(pick a point or choose an option.)** • **CLose** - closes the arc and then returns to the previous prompt • **Direction** - defines the direction of the tangent line in the start point of the arc; prompts you: **Direction from start** - **(pick a point to define the direction of the tangent line.) End of arc** - **(pick a point.)** • **Halfwidth** - specifies the halfwidth of the arc, the distance from the centerline to the edge of the wide arc; prompts you: **Starting half-width <0>** - **(enter a width for the start of the arc segment.) Ending half-width <175.96>** - **(enter a width for the end of the arc segment.)** • **Radius** - specifies the radius of the arc; prompts you: **Radius of arc** - **(specify a radius.) Set end of arc or [included Angle]** - **(pick a point to end the arc.)** • **Second point** - prompts you to define the second point and endpoint of a 3-point arc; prompts you: **Second point** - **(pick a point along the arc.) End of arc** - **(pick a point to determine the end of the arc.)** See the Arc (6.4.12) command.
Close	Closes the polyline; the programautomatically draws a segment from the current next-point to the starting point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Distance	Specifies the length and angle of the next segment; prompts you: 1. **Distance for segment** - **(specify the length of the segment.)** 2. **Angle of segment** - **(specify the angle of the segment.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Distances are measured from the last vertex.Angles are measured counterclockwise from the positive x-axis.
Follow	Draws the next segment at the same angle; move the mouse to indicate the length. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Halfwidth	Specifies the halfwidth for the current segment, as well as all segments that follow -- until you change the width or halfwidth. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i You can specify different starting and ending halfwidths, which creates tapered segments; prompts you: 1. **Starting half-width** - **(specify the width at the start end of the segment; enter a value, or pick two points in the drawing.)** 2. **Ending half-width** - **(specify the width at the start end of the segment; enter a value, or pick two points in the drawing.)**
Width	Specifies the width for the current segment, as well as all segments that follow -- until you change the width or halfwidth. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i You can specify different starting and ending widths, which creates tapered segments; prompts you: 1. **Starting width** - **(specify the width at the start end of the segment; enter a value, or pick two points in the drawing.)** 2. **Ending width** - **(specify the width at the start end of the segment; enter a value, or pick two points in the drawing.)** When you specify 0 as the width, the polyline is drawn to look like a line.
Undo	Undoes (undraws) the last segment.

Grips Editing

Depending on the value of the GRIPS (6.10.17) system variable polylines have grips on its endpoints and vertices and/or at the midpoint of each segment. The value of GRIPS can be 0, 1 or 2

Value	**Grips display**
0	No grips
1	Endpoint grips only
2	Endpoint and midpoints grips

Figure 6.1680

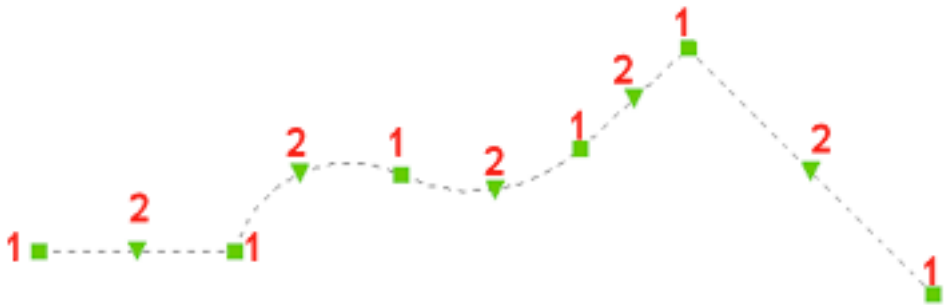
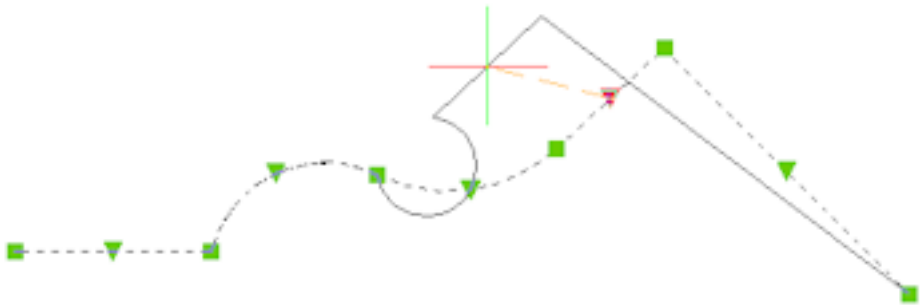


Figure 6.1681



Polylines can be edited directly through grips: 1. Select the polyline. Notice that it has a square grip at each vertex (1) and a triangular grip at the midpoint of each segment (2). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
2. Do one of the following:

- Drag the midpoint grip of a line segment to move the segment. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
- Drag the midpoint grip of an arc segment to modify its radius. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel
- Drag a vertex grip to move the selected vertex. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/C
- Splined polylines have a grip at each control point, which can be stretched:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/PlineGripStretchSplineCubic.png})
Cubic or **Quadratic** splined polyline. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/
Curve fit splined polyline.

Figure 6.1682

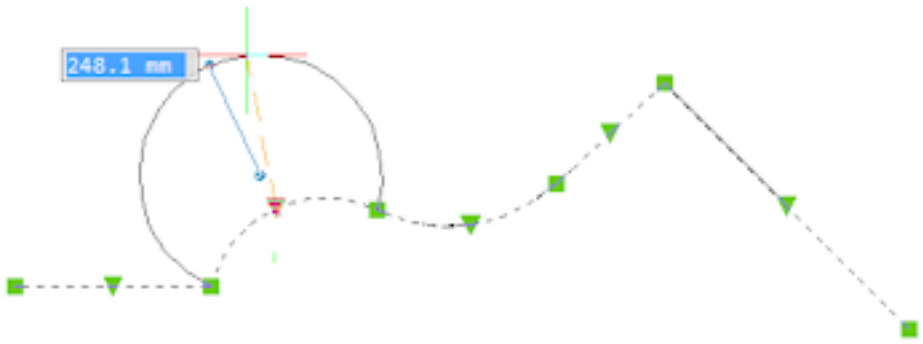


Figure 6.1683

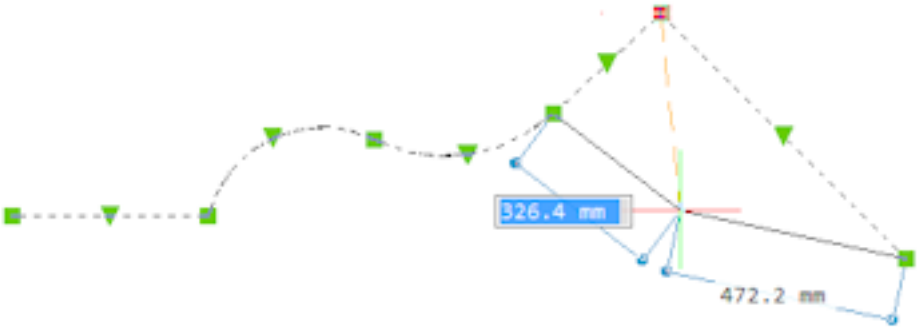


Figure 6.1684

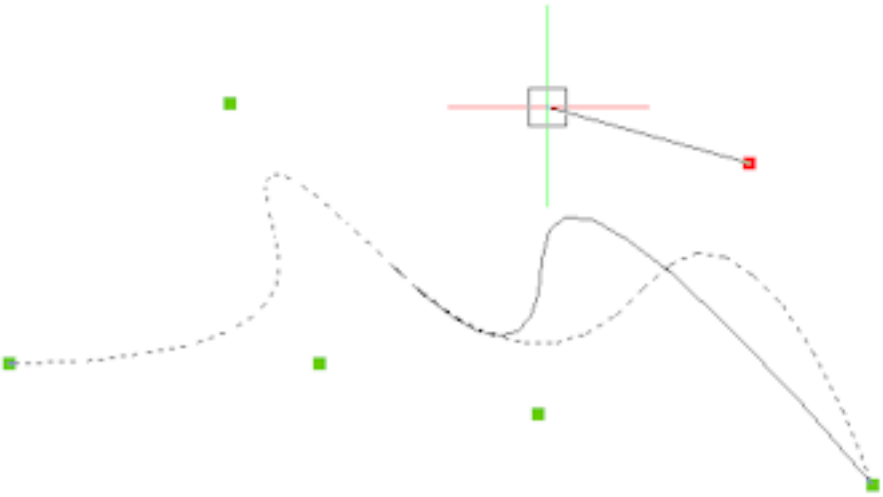
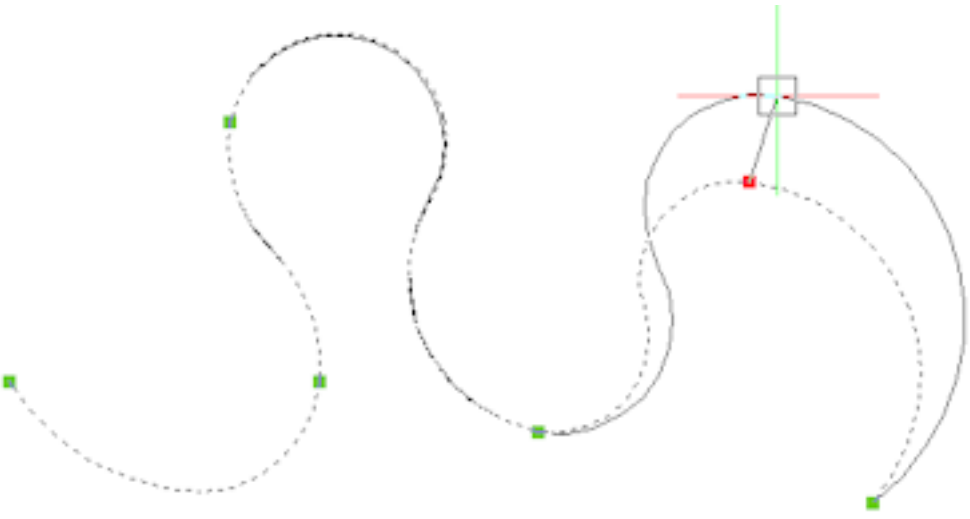


Figure 6.1685



Procedures

- Drawing polylines (1.17.12)
- Creating boundary polylines (1.17.12)

Related Commands

3dPoly (6.3.10) - draws polylines with x,y,z coordinates. Boundary (6.6.15) - creates a polyline that forms a boundary inside a closed area through a dialog box -Boundary (6.6.16) - creates a polyline boundary inside closed areas through the command bar. ConvertPoly (6.9.45) - converts lightweight polylines to classic polylines (2D polylines) and vice versa. Helix (6.16.7) - creates a 2D spiral or a 3D helix. MLine (6.21.26) - draws multilines. PEdit (6.24.20) - edits polylines. Donut (6.10.35) - draws fat circles out of polylines. Polygon (6.24.43) - draws regular polygons out of polylines. Rectang (6.26.6) - draws rectangles out of polylines. Solid (6.27.37) - draws solid-filled entities with 3 and 4 sides. Trace (6.29.24) - draws wide lines.

Figure 6.1686

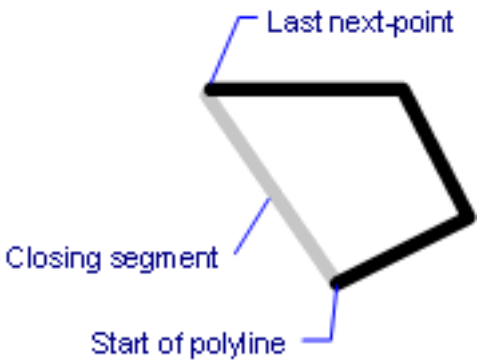


Figure 6.1687

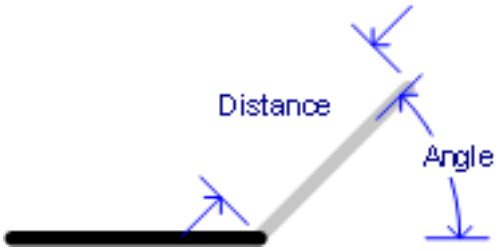


Figure 6.1688



Figure 6.1689



Figure 6.1690



Figure 6.1691



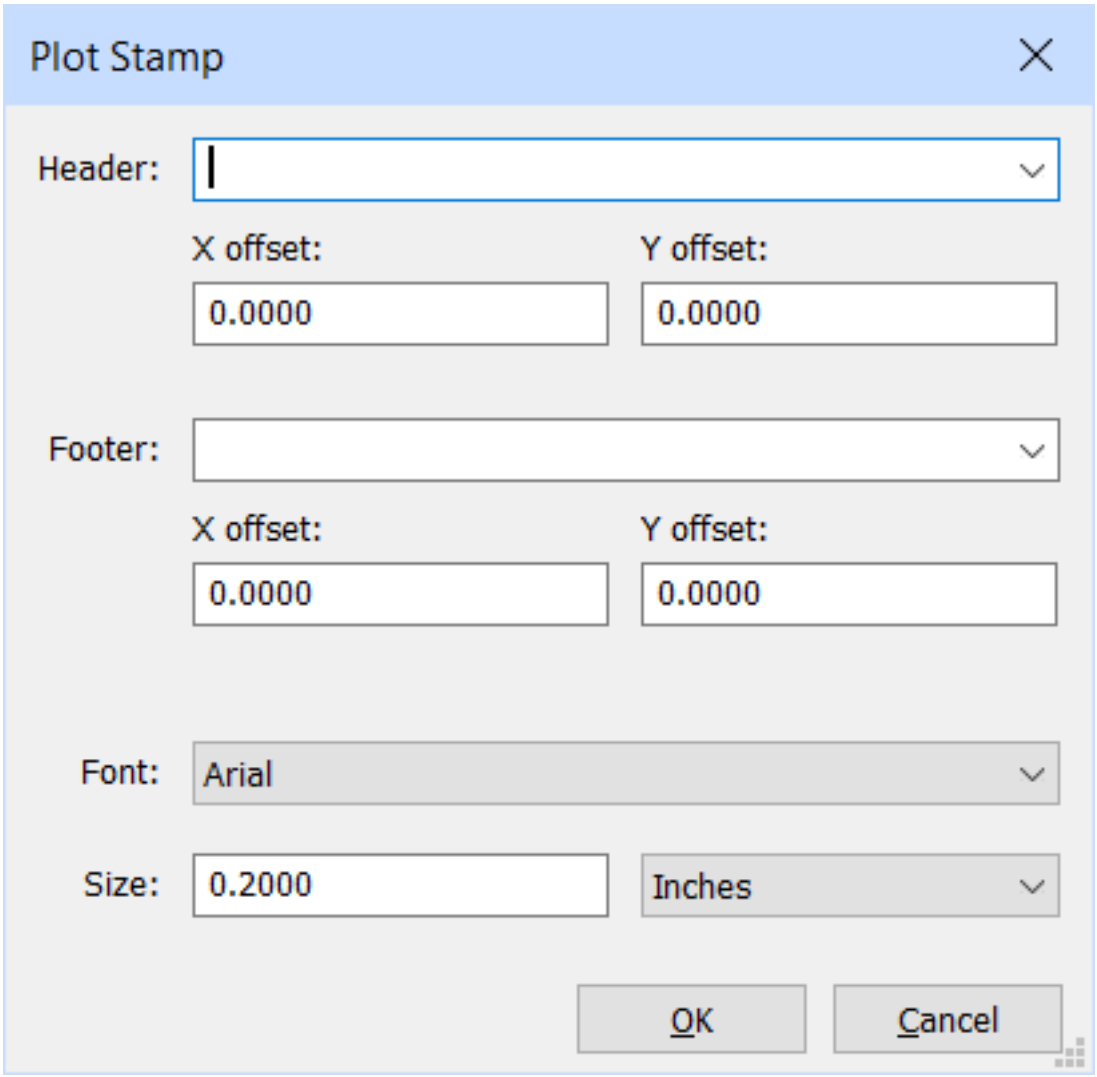
6.24.28 PlotStamp

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adds information about drawings and prints to the edge of the plotted drawing.

Accessing the Command

command bar: **plotstamp** :**PlotStamp** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS dlg.png}) Enter options, and then click **OK**. Use the **Preview (6.24.45)** command to check that the plot stamp appears correctly.

Figure 6.1692



Command Options

Option	Description
Header Options	The header appears at the top of the plot. Enter the following: 1. **Header** - click the droplist to choose a meta-phrase for the header, or else enter your own text (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/header-droplist.png}) After you select a meta-phrase (one that starts with the $), it is not filled in until you run the Preview or Print commands. Data such as $Name and $Company are extracted from properties defined by the **DwgProps (6.10.42)** command. If you leave properties blank, the program substitutes generic information, such as "YourName" and "YourCompany." The $Date and $DateTime fields report the date and time the drawing is plotted, not when the drawing was created or last edited. 2. **X offset** - specify the distance from the left margin; some data, like the drawing name and the date, are centered 3. **Yoffset** - specify the distance from the top margin of the drawing; enter a positive number to move the plotstamp downward
Footer Options	The footer appears at the bottom of the plot. Enter the following: 1. **Header** - click the droplist to choose preset wording for the header, or else enter your own text. 2. **X offset** - specify the distance from the left margin; some data, like the drawing name and the date, are centered 3. **Yoffset** - specify the distance from the bottom margin of the plot; enter a positive number to move the plotstamp upwards
Font Options	Specify the font used for the plotstamp: 1. **Font** - click the droplist, and then choose one of the TTF and SHX font names from among those provided by your computer 2. **Size** - specify the height of the text; the initial unitsare the current drawing units, such as 0.2 being 0.2 inches tall 3. **Inches** - click the droplist to choose a different set of units, either inches or millimeters
OK	Applies the changes to the plot stamp, which will appear the next time you use the Preview or Print commands.

Procedures

- General procedure for printing (1.25.1)
  - To define the plot stamp (1.25.1)

Related Commands

Print (6.24.46) - prints drawings through a dialog box Plot (6.24.26)- plots drawings through the command bar. QPrint (6.25.3)- plots drawings using the current settings. Preview (6.24.45)- previews drawings before they are plotted. DwgProps (6.10.42) - provides data used by the PlotStamp command Publish (6.24.55)- allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file.

Figure 6.1693

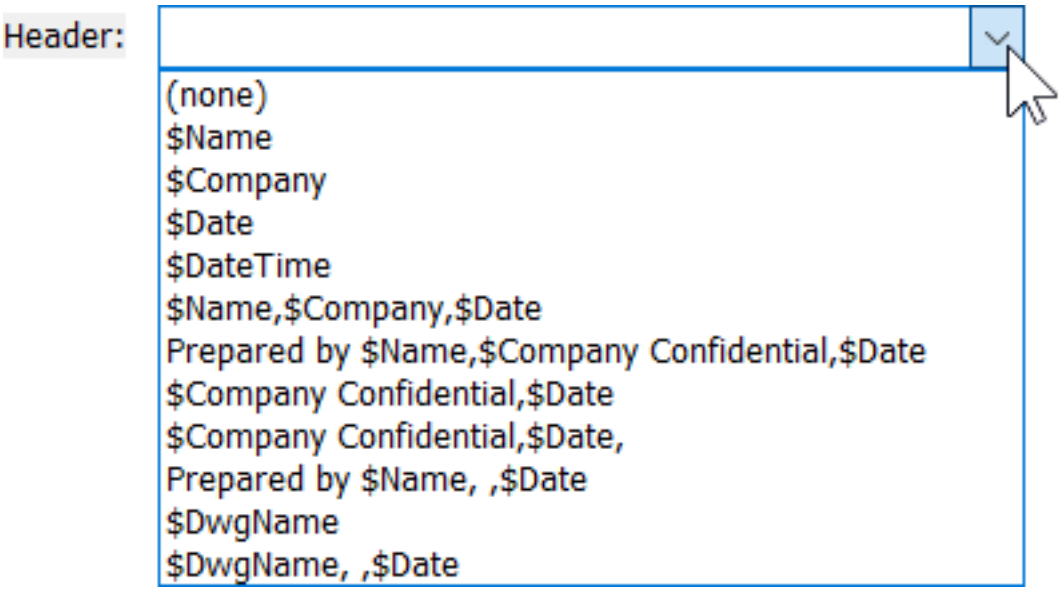


Figure 6.1694



6.24.29 PlotStyle

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the current plot style. **Note:** This command operates only when plot styles are enabled in drawings.

Accessing the Command

command bar: **plotstyle** : **PlotStyle** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
Choose a plot style name, and then click **OK**. When drawings are not set up for plot styles, this command displays the dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/PlotStylesWarning.png})
Click **OK**.

Figure 6.1695

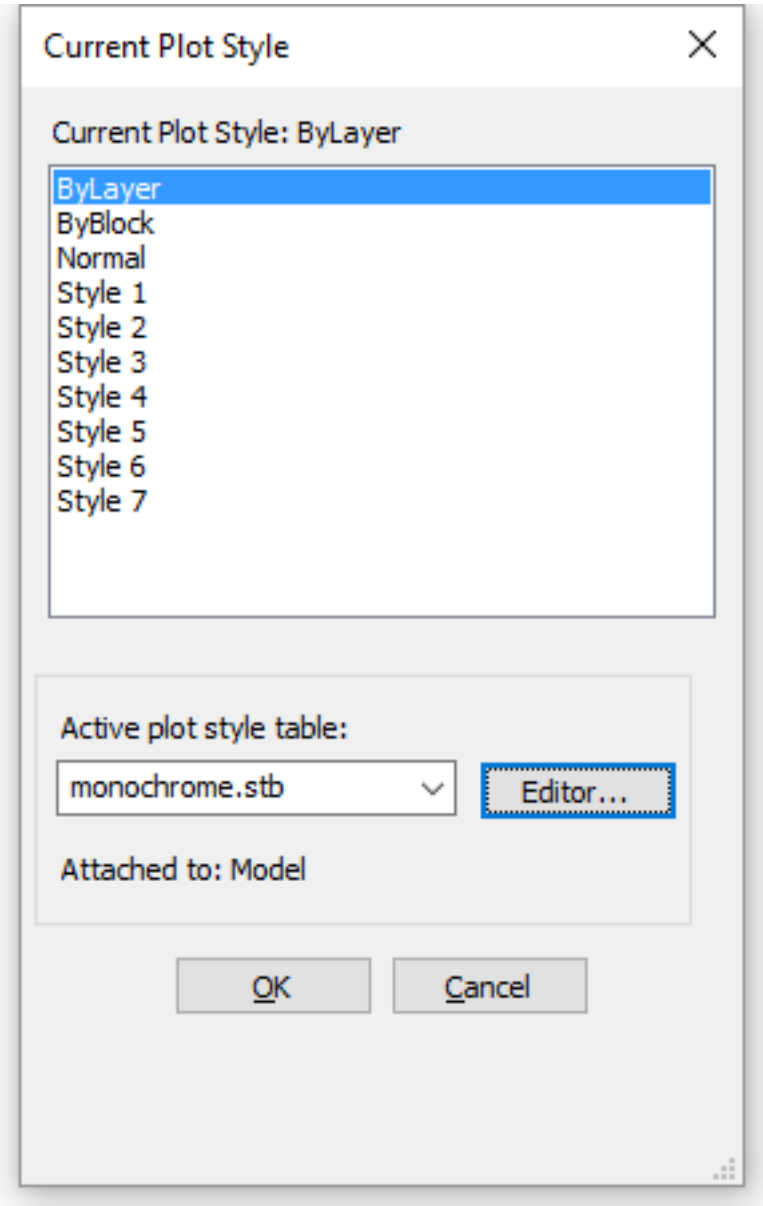
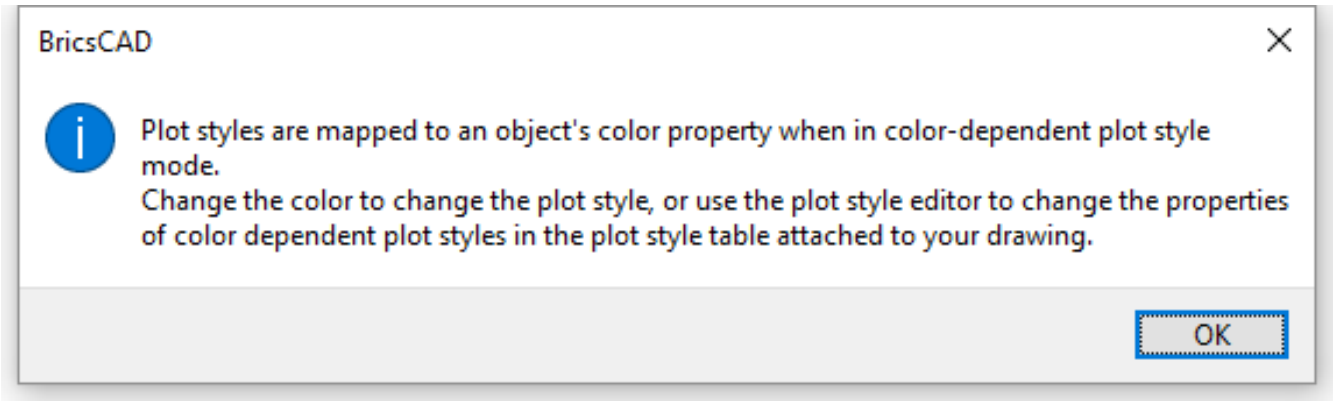


Figure 6.1696



Command Options

Option	Description
Active plot style table	Lists the names of STB files (short for ”style table”).
Editor	Displays the PlotStyle Table Editor dialog box; see the StylesManager (6.27.55) command.

Related Commands

ConvertCtb (6.9.42) - converts CTB plot style files to STB files. ConvertPStyles (6.9.46) - converts drawings from CTB to STB plot styles. Layer (6.20.2) - assigns plot styles to layers. PageSetup (6.24.1) - assigns plot style to drawings. Print (6.24.46) - plots drawings with plot styles. Properties (6.24.49) - changes plot styles assigned to entities. StylesManager (6.27.55) - creates and attaches plot styles to drawings.

Figure 6.1697



6.24.30 PlotterManager

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and edits customized parameter PC3 files for printers and other output devices.

Accessing the Command

command bar: **plottermanager** transparent: **'plottermanager** menu bar: **File | Plotter Manager** : **PlotterManager**
Displays a window: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SelectPlotterCon

- To edit a plotter configuration:

1. Choose a PC3 file.(PC3 is short for "plotter configuration version 3".)
2. Double-click it to edit it. The program displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/
3. Edit the settings.
4. Click **OK**.

- To create a new plotter configuration file:

1. Double-click the **Create a Plotter Configuration** shortcut.The Plotter Configuration Editor dialog displays.
2. Select a printer from the **Printer Name** droplist. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/C
3. Edit the settings.
4. Click **OK**.
5. Enter a name to save the new plotter configuration file
6. Click **OK**.

The new PC3 file is added to the **C:\Users\<loginname>\AppData\Roaming\Bricsys\BricsCAD\V...\en_US\plotconfig** folder.

Figure 6.1698

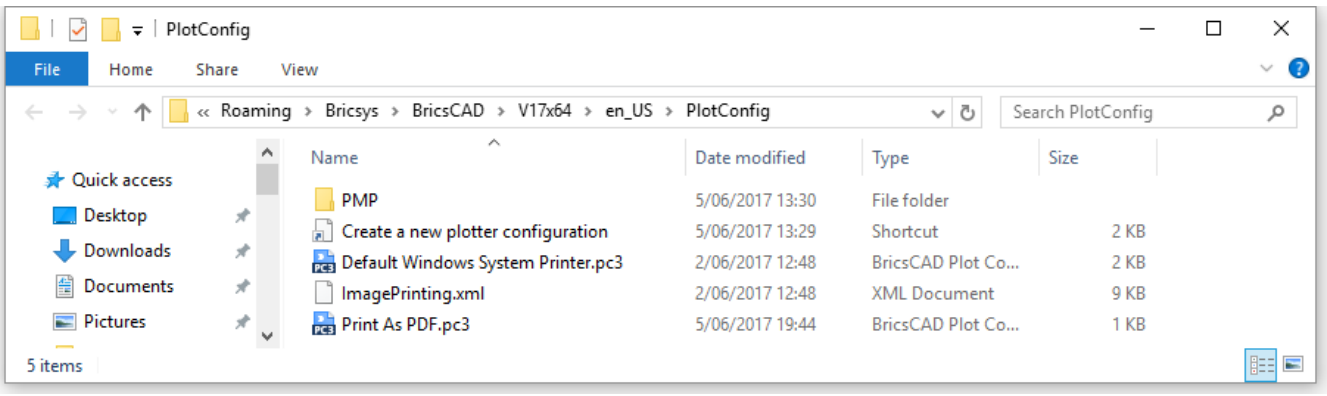


Figure 6.1699

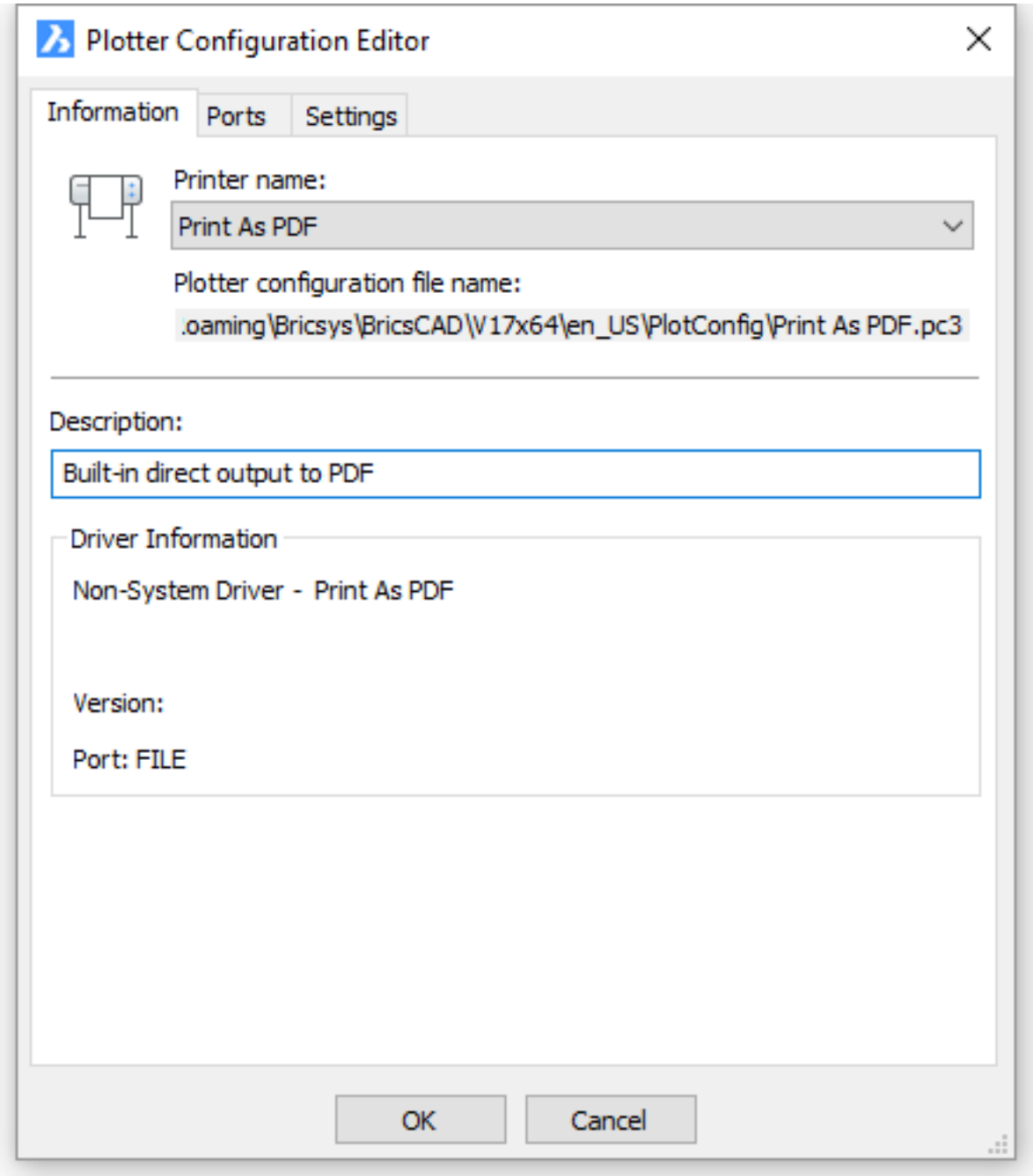
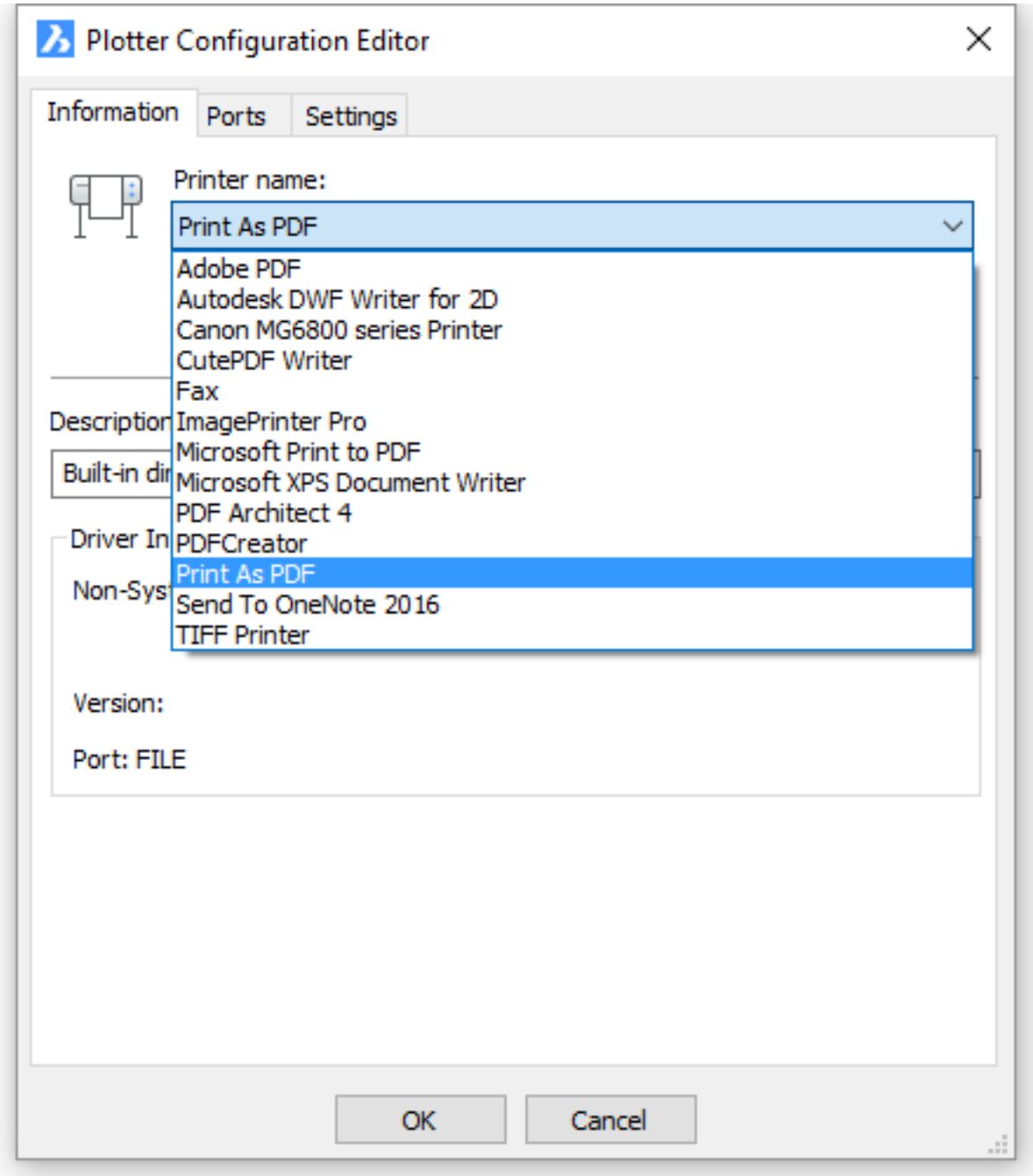


Figure 6.1700

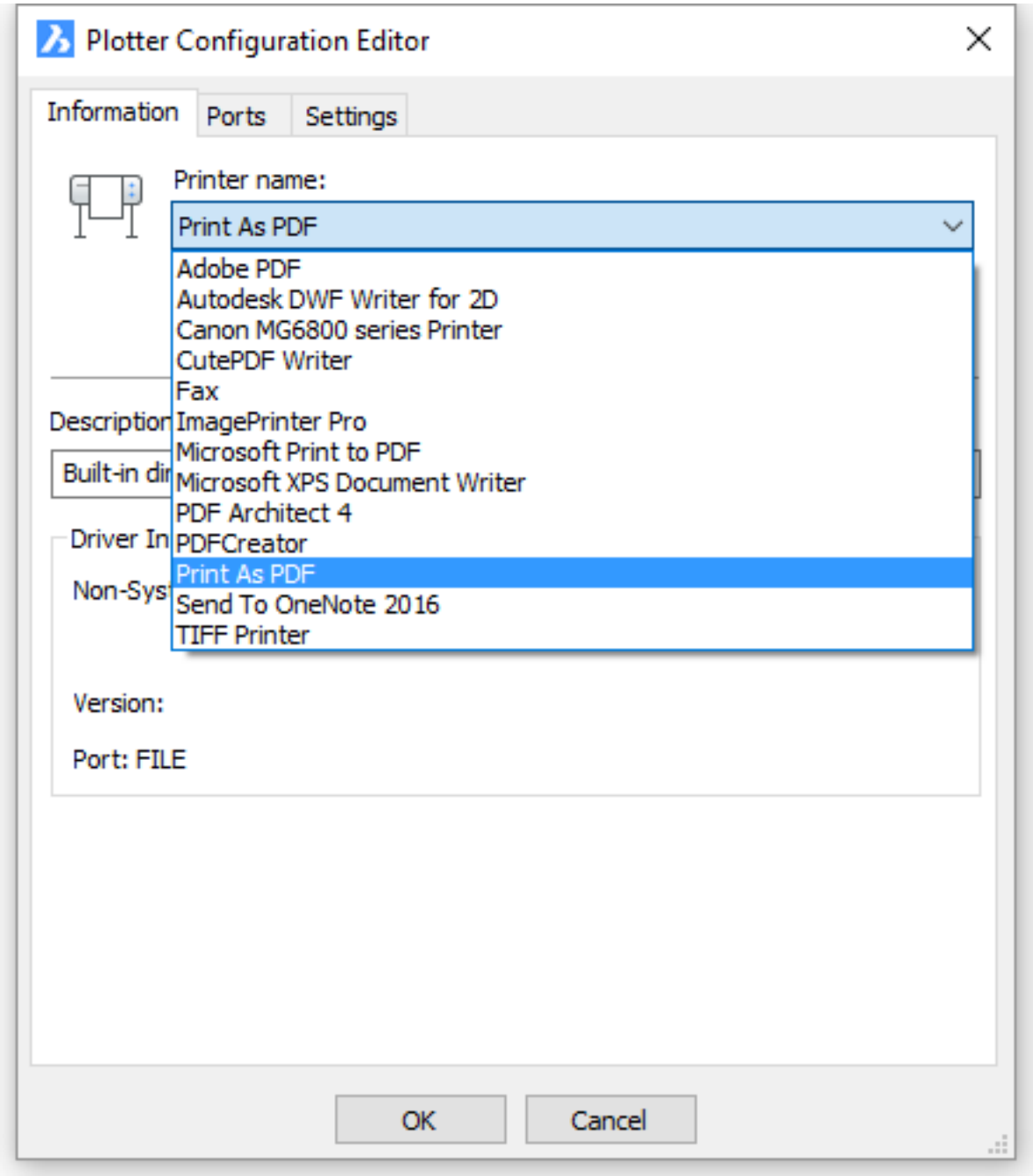


Information Tab Options

Click the Information tab: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/PlotterCon

Information Tab Option	Description
Printer Name	Lists the names of printers supported by the the operating system.
Description	(Ooptional)Describes the plotter configuration, and is displayed by tooltips.
OK	Closes the dialog box and saves the changes.
Cancel	Ignores the changes and closes the dialog box,

Figure 6.1701

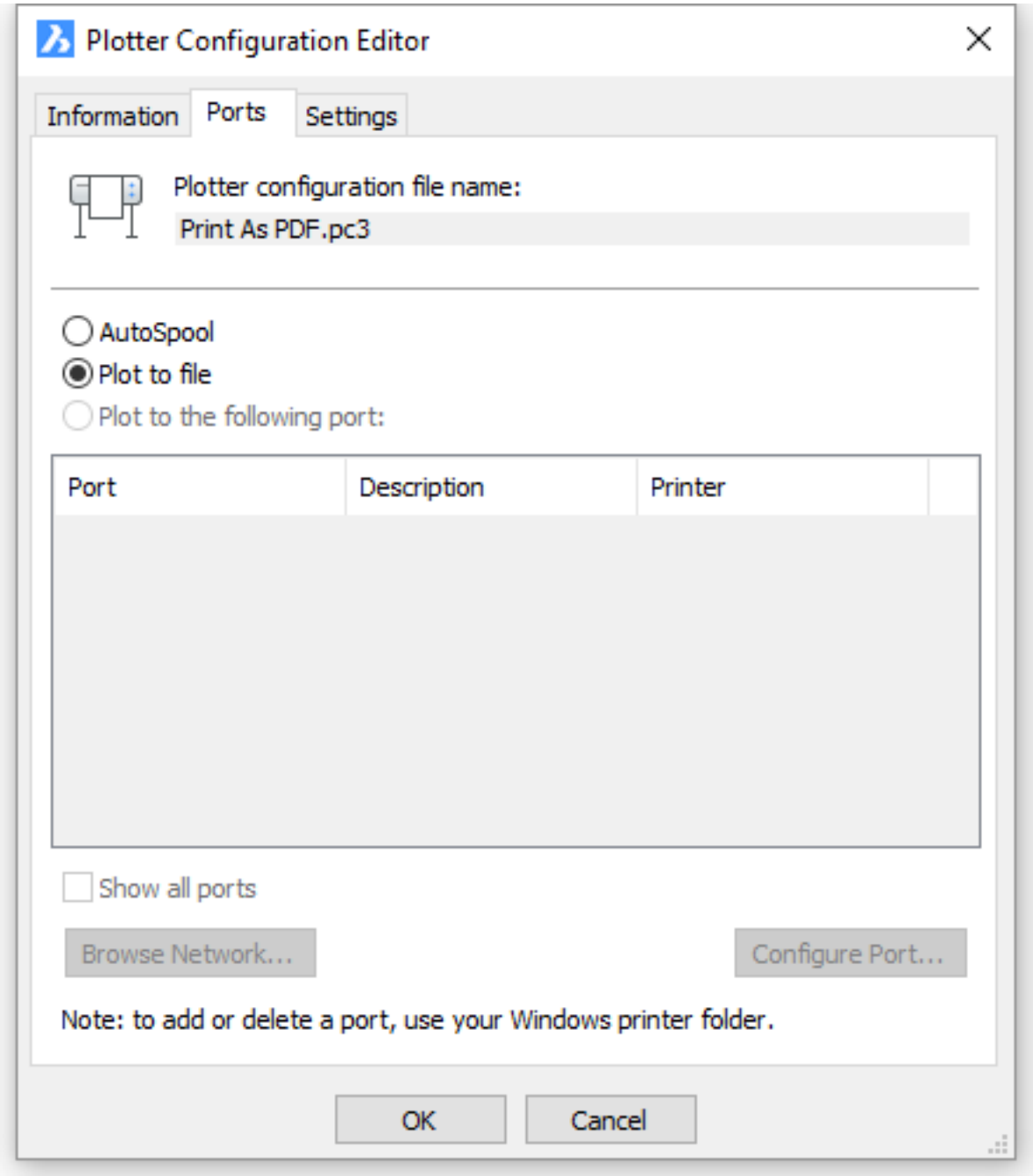


Ports Tab Options

Click the Ports tab: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/PlotterConfigurat

Ports Tab Options	Description
AutoSpool	Sends drawings to the spooler software, which records the plot file and then sends it to the printer. The spooler software must be set up separately, although generally no longer necessary as the operating system usually includes spooling.
Plot to file	Plots drawings to PLT files. These plot files can be read by other software programs.
Plot to the following port	Plots drawings to the printer attached to the specified port.
Show all ports	Displays the names of all local and network ports on the current computer.
Browse Network	Opens the Browse Network window so that you to locate printers attached to the network.
Configure Port	Opens the Configure Port dialog box to configure the selected port.

Figure 6.1702



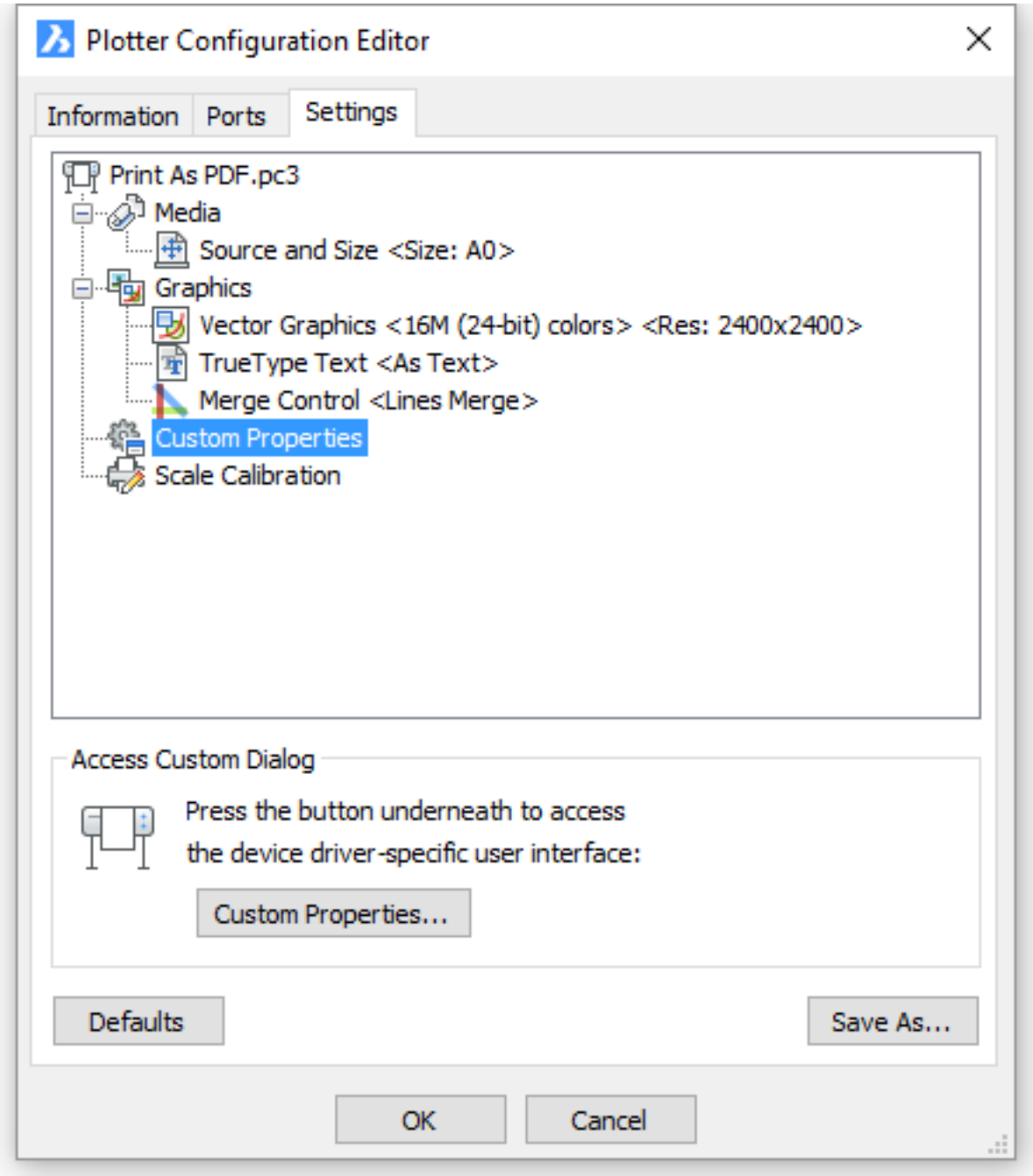
Settings Tab Options

Click the Settings tab: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/PlotterConfigu
The settings available depend on the capabilities of the printer selected. The ones shown below are common to all printers and output devices.

Settings Tab Options	Description
Media	Specifies settings for the media (paper).
Collating	Specifies how to collate pages: • **None** - does not collate; drawings are printed normally • **Regular** - prints layouts of each drawing together This option is useful when printing multiple copies of drawings with multiple layouts.
Graphics	Specifies how the three types of graphics are handled by the printer: Vector Graphics options: • **Color depth** - specifies the number of colors to use • **Color** - outputs the drawing in color. • **Monochrome** - outputs the drawing in monochrome (black or shades or gray) • **Resolution** - adjusts the resolution; a lower resolution is coarser, but prints more quickly • **Dithering** - specifies the type of dithering, which simulates more colors at the cost of lower resolution Raster Graphics options: • **Raster** - adjusts how raster images are handled • **OLE** - adjust how OLE objects are printed; ranges from**None** (does not print OLE entities, faster) to**Best** (prints OLE entities at highest quality) • **Trade-off** - specifies how the program should handle drawings with very large raster images that take up much memory: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou **Lower Resolution** - prints images with lower resolution but full colors; suitable for photographs **Balance** - prints images with as many colors and as much resolution as possible **Fewer Colors** - prints images with fewer colors but full resolution; suitable for grayscale images TrueType text options: • **TrueType as text** - prints text as text; preferred when the printer holds the needed fonts. • **TrueType as graphics** - prints text as graphics; preferred when drawings are printed to files.

Merge Control	Specifies how overlapping lines are merged: • **Lines Overwrite** - specifies that entities with higher draw order overwrite other entities. • **Lines Merge** - specifies that entities are blended to merge their colors; allows entities with lower draw order to be seen.
Custom Properties	Accesses the Custom Properties dialog box, which lists properties unique to the selected output device. The content varies according the capabilities of the device. Custom properties are not supported on the Linux platform.
Scale Calibration	Calibrates the plotter;displays the Calibrate Plotter wizard. This option was necessary in the days of pen plotters, which could go out of alignment; today's inkjet and laser printers not to have the problem.
Save As	Displays the Save As dialog box to save settings as PC3 files. 1. Enter a file name. 2. Click **Save**. PC3 is short for "plotter configuration version 3"; these files store parameters for specific plotters.You can have multiple PC3 files for each printer and output device.
Defaults	Resets all options to their default values; prompts you: **Do you want to set everything to defaults? - (type Y or N.)**

Figure 6.1703



Procedures

- To create a plotter configuration file (1.25.2)
- To edit a plotter configuration (1.25.2)
- To assign a plotter configuration (1.25.2)
- Plotter Configurations (1.8.6)

Related Commands

ConvertCtb (6.9.42) - converts CTB plot style files to STB files. ConvertPStyles (6.9.46) - converts drawings from CTB to STB plot styles. PageSetup (6.24.1) - assigns plot style to drawings. PlotStyle (6.24.29) - sets the current plot style. Print (6.24.46) and Plot (6.24.26) - plot drawings. StylesManager (6.27.55) - creates and attaches plot styles to drawings.

Figure 6.1704

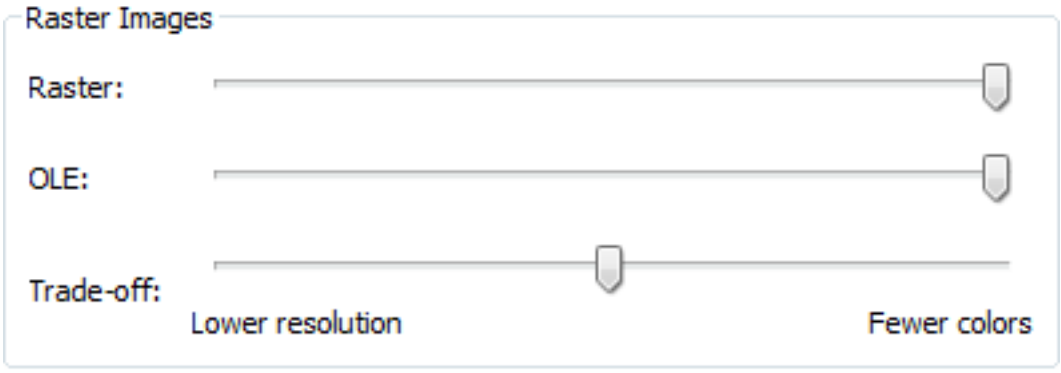


Figure 6.1705



6.24.31 Point

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws point entities.

Accessing the Command

command bar: **point** alias: **po** menu bar: **Draw | Point** ribbon:**Draw | Draw | Points | Point** toolbar: **Draw |** : **Point Prompts you in the command bar:** Location of point or [point Settings.../Multiple points]: **(Pick a point, enter x,y,z coordinates, or enter an option)**

Command Options

Option	Description
Location of point	Specifies the location of the point in the drawing. • Pick a point • Enter x,y,z coordinates
Point settings	Displays the Points section of the **Settings** dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Point display mode** - specifies the look of points. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/F • **Point display size** - specifies the size of points.
Multiple points	Repeats the command to draw multiple points; keeps prompting you: **Location of point or [point Settings...]** - **(specify a point or press Enter to stop.)**

Grips Editing

Points can be edited directly through grips:

1. Select the point. Notice that it has one grip.
2. Drag the grip to move the point.



Procedures

- Defining the point display settings (1.17.25)
- Drawing points (1.17.25)

Related Commands

DdPType (6.10.18) - defines the size and shape of points. Divide (6.10.33) - places evenly-spaced points along

Figure 6.1706

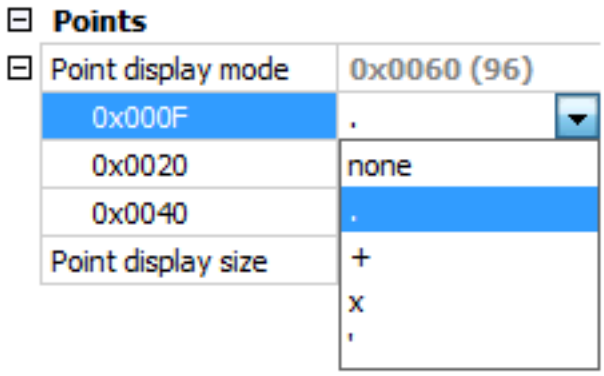


Figure 6.1707

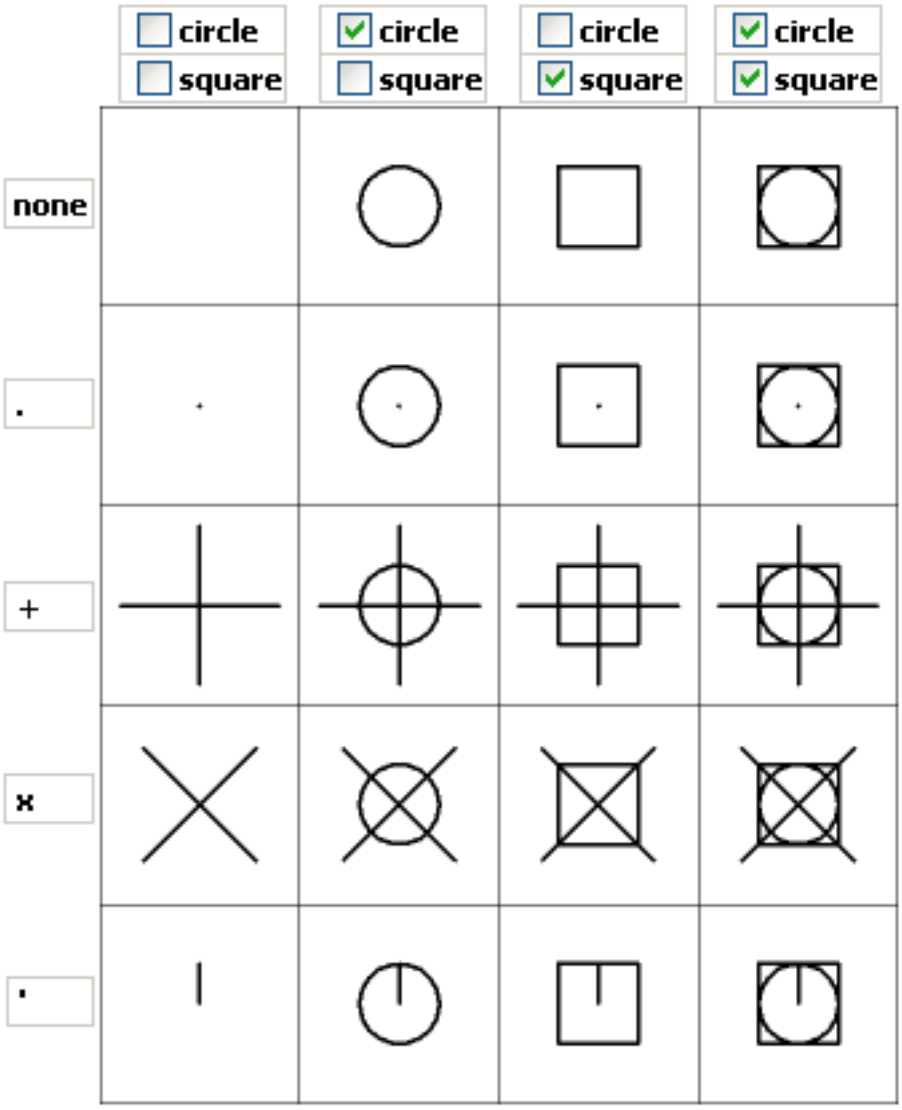


Figure 6.1708



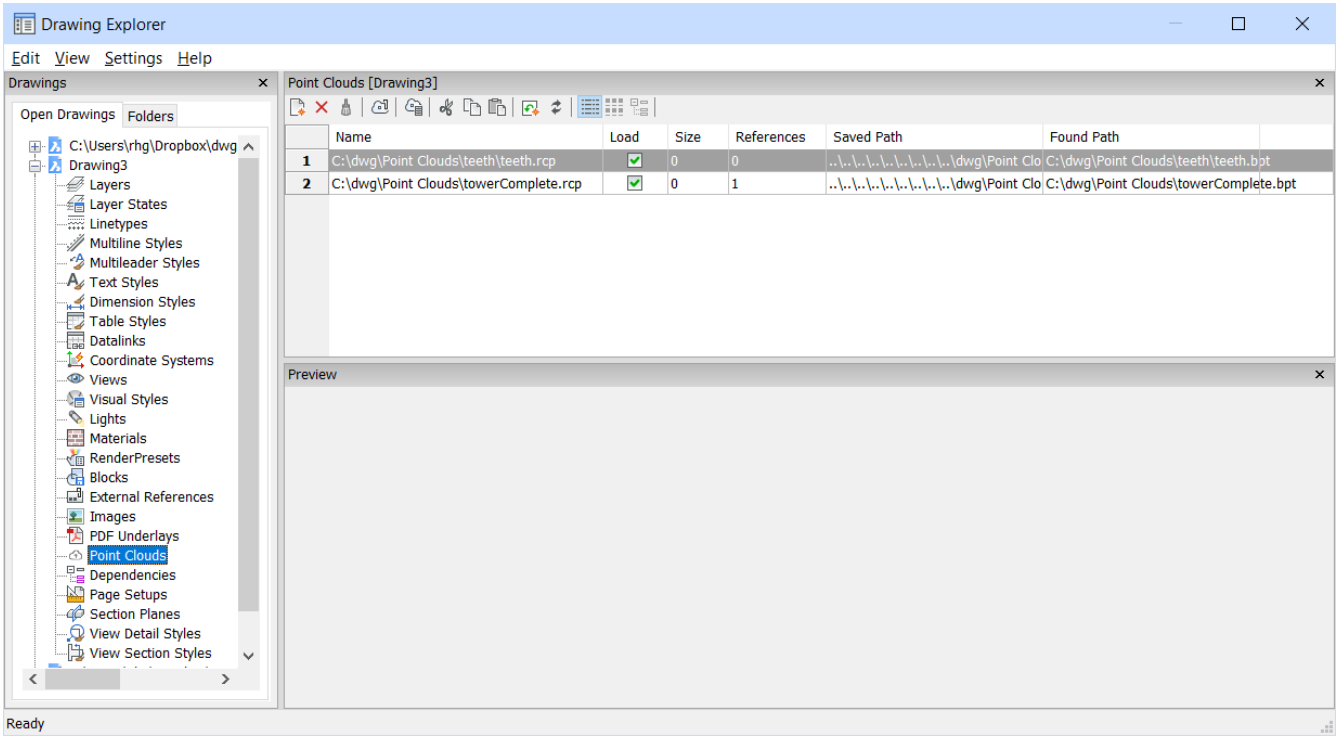
6.24.32 PointCloud

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the Point Cloud section of the Drawing Explorer.

Accessing the Command

Keyboard: **pointcloud :PointCloud** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V dlg.png})

Figure 6.1709



Command Options

Option	Description
New	Attaches a BPT point cloud file to the current drawing, like the PointCloudAttach (6.24.34) command. Preprocess option, shown below.
Delete	Removes the selected point cloud from the current drawing.
Insert	Inserts additional copies of the existing point cloud files into the drawing; displays the Attach Point Cloud dialog box, like the PointCloudAttach (6.24.34) command.
Preprocess	Converts LAS, PTS, and PTX point cloud files into the BricsCAD BPT point cloud format; displays the Point CLoud Data File dialog box, like the PointCloudPreprocess (6.24.60) command
Load	Toggles the display of the point cloud

Related Commands

AttachmentsPanelOpen (6.4.24) - displays the Attachments panel from which point clouds and other attachments can be loaded and controlled. PointCloudAttach (6.24.34) - opens a BPT point cloud file, and then attaches it to the current drawing. PointCloudBoundary (6.24.35) -sets the display of the bounding box of point clouds. PointCloudColorMap (6.24.36) - colorizes the points based on Intensity, Elevation, and Classification of the point cloud. PointCloudCrop (6.24.38) - crops a point cloud entity, displaying only the part of the point cloud inside or outside a prismatic or cylindrical volume. PointCloudPointMax (6.24.37) - defines the maximum amount of points shown per point cloud. PointCloudPointSize (6.24.39) - specifies the size of points. PointCloudPointSize_Minus (6.24.40) - decreases the size of points. PointCloudPointSize_Plus (6.24.59) - increases the size of points. PointCloudPreprocess (6.24.60) -converts point cloud files to BricsCAD BPT format. PointCloudUncrop (6.24.61) - removes all croppings from a point cloud.

Figure 6.1710



6.24.33 PointCloud2dVsDisplay

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Controls whether a bounding box and a warning message display when trying to view a point cloud in the 2D Wireframe visual style.

Accessing the Command

Keyboard:**pointcloud2dvsdisplay : pointcloud2dvsdisplay Prompts you:** 2DDisplay [Display/Hide] <Display>: **(Choose an option.)**

Command Options

Option	Description
Display	A bounding box and a warning message display when the 2D Wireframe visual style is chosen. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/Po
Hide	Point clouds do not display. No bounding box nor warning message display.

Settings

Name	Title	Description
PointClouid2dVsDisplay	Point cloud toggle show/hide bounding box in 2d wireframe mode	Point clounds display in a 3D visual style only. Optionally a bounding box and a warning message display In 2D Wireframe visual style. • 0: Display bounding box and warning message. • 1: Hide bounding box and waring message.

Related Commands

PointCloud (6.24.32)- opens the Drawing Explorer to the Point Cloud section; converts, loads, and unloads point cloud file attachments. PointCloudAttach (6.24.34) - opens a BPT point cloud file, and then attaches it to the current drawing. PointCloudBoundary (6.24.35) -sets the display of the bounding box of point clouds.

Figure 6.1711

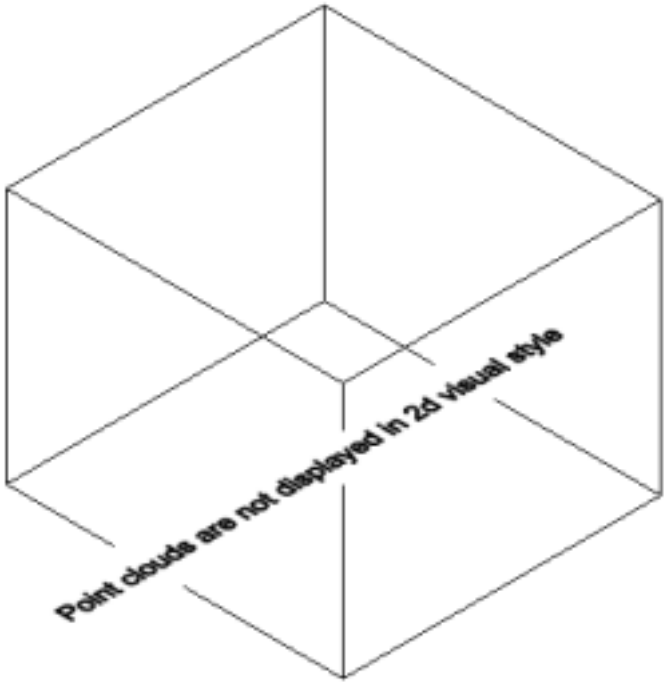


Figure 6.1712



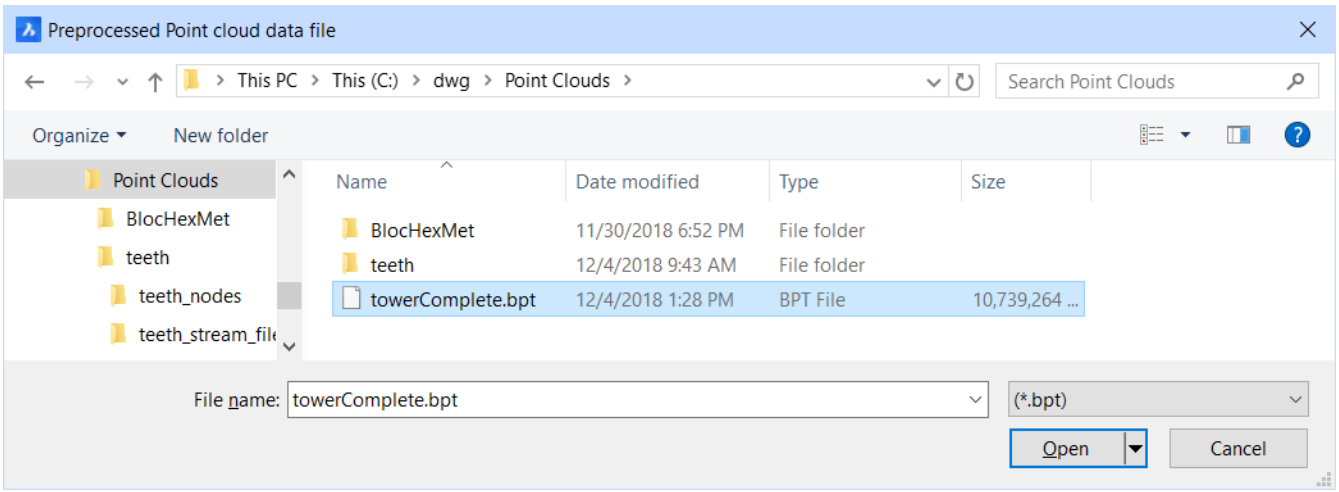
6.24.34 PointCloudAttach

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Attaches point cloud files (that were previously converted to BricsCAD's BPT format) to the current drawing.

Accessing the Command


Keyboard: **pointcloudattach** Toolbar: **Point Cloud :PointCloudAttach** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pointcloudattach-open.png})

Figure 6.1713

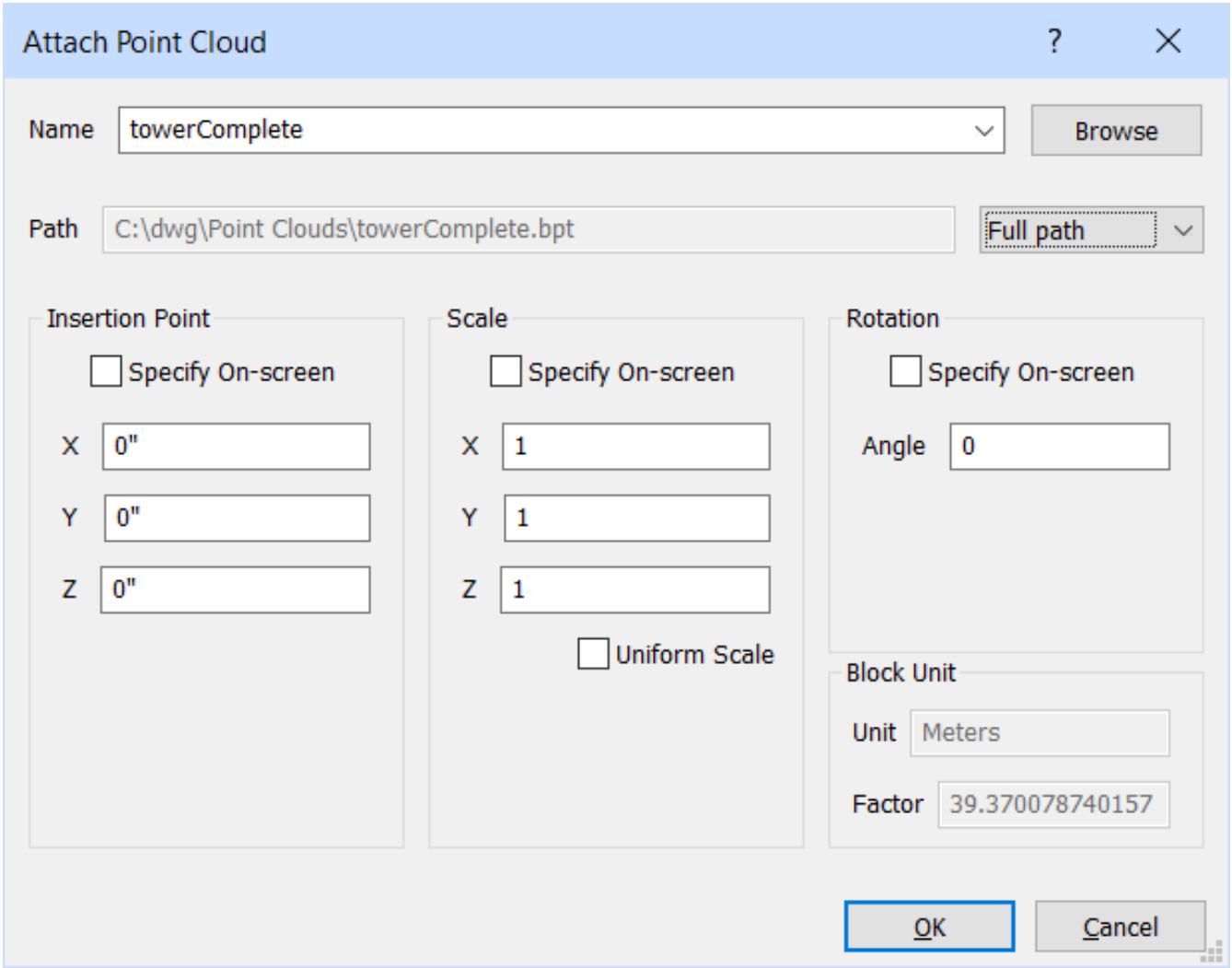


Command Options

Option	Description
File name	Specifies the name of the BPT point cloud file to attach. Point cloud files in other formats must first be converted to BPT with the PointCloudPreprocess (6.24.60) command.
Open	Opens the point cloud file, then displays the Attach Point Cloud dialog box; see below.
Cancel	Cancels the attachment process.

After you click **Open**, the program displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
options.png})

Figure 6.1714



Command Options

Option	Description
Name	Specifies a name for the point cloud to be attached.You can use the name of the file, or give it another name.
Path	Specifies the path to the folder holding the point cloud.The droplist lets you choose whether to • **Full path** - keep the full path • **Relative path** - relative to the folder of the current drawing • **No path** - at all
Insertion Point	Specifies where the lower left corner of the point cloud should be located in the drawing • **Specify on-screen** - prompts you for the insertion point after you close the dialog box • **X, Y, Z** - specifies the x, y, and z coordinates for the insertion point
Scale	Specifies the size of the point cloud: • **Specify on-screen** - prompts you for the scale factor after you close the dialog box • **X, Y, Z** - specifies the scale factor for each of the three directions • **Uniform Scale** - forces all three scale factors to be the same The point cloud is scaled relative to the insertion point.
Rotation	Specifies the angle at which to position the cloud: • **Specify on-screen** - prompts you for the rotation angle after you close the dialog box • **Angle** - specifies the angle The point cloud is rotated relative to the insertion point.
OK	Attaches the point cloud into the current drawing.
Cancel	Cancels the attachment process.

Related Commands

PointCloud (6.24.32)- opens the Drawing Explorer to the Point Cloud section; converts, loads, and unloads point cloud file attachments. AttachmentsPanelOpen (6.4.24) - displays the Attachments panel from which point clouds and other attachments can be loaded and controlled. PointCloudBoundary (6.24.35) -sets the display of the bounding box of point clouds. PointCloudColorMap (6.24.36) - colorizes the points based on Intensity, Elevation, and Classification of the point cloud. PointCloudCrop (6.24.38) - crops a point cloud entity, displaying only the part of the point cloud inside or outside a prismatic or cylindrical volume. PointCloudPointMax (6.24.37) - defines the maximum amount of points shown per point cloud. PointCloudPointSize (6.24.39) - specifies the size of points. PointCloudPointSize_Minus (6.24.40) - decreases the size of points. PointCloudPointSize_Plus (6.24.59) - increases the size of points. PointCloudPreprocess (6.24.60) -converts point cloud files to BricsCAD BPT format. PointCloudUncrop (6.24.61) - removes all croppings from a point cloud.

Figure 6.1715



6.24.35 PointCloudBoundary

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the display of the bounding box of point clouds.

Accessing the Command

Keyboard:**pointcloudboundary : pointcloudboundary Prompts you:** Boundary [Hide/OnlyWhenSelected/Show] <Hide>: **(Choose an option or press Enter to accept the current state.)**

Command Options

Option	Description
Hide	Point cloud boundaries do not display. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/Po
OnlyWhenSelected	The point cloud boundary displays when the point cloud is selected or when hovering over a point cloud when Selection Preview is on or when the Quad is active. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/Po
Show	Always display the point cloud boundary. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/Po

Setting

Name	Title	Description
PointCloudBoundary	Point Cloud Boundary	Sets thedisplay of the bounding box of point clouds. The value is saved in the registry.

Related Commands

PointCloudAttach (6.24.34) - opens a BPT point cloud file, and then attaches it to the current drawing.

Figure 6.1716

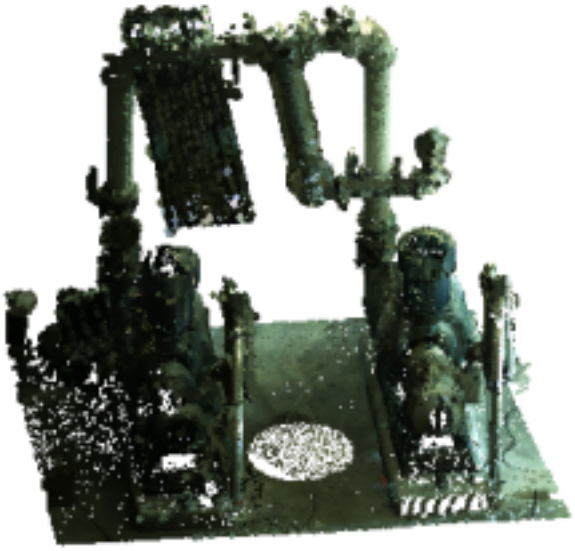


Figure 6.1717

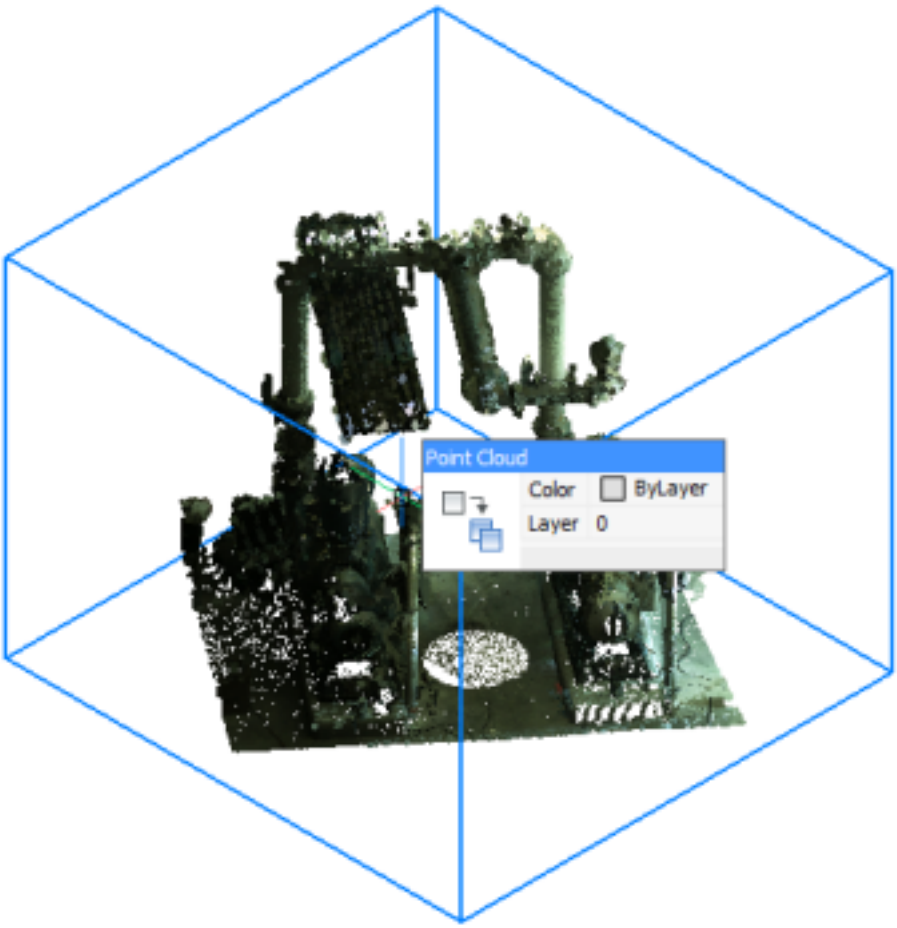


Figure 6.1718

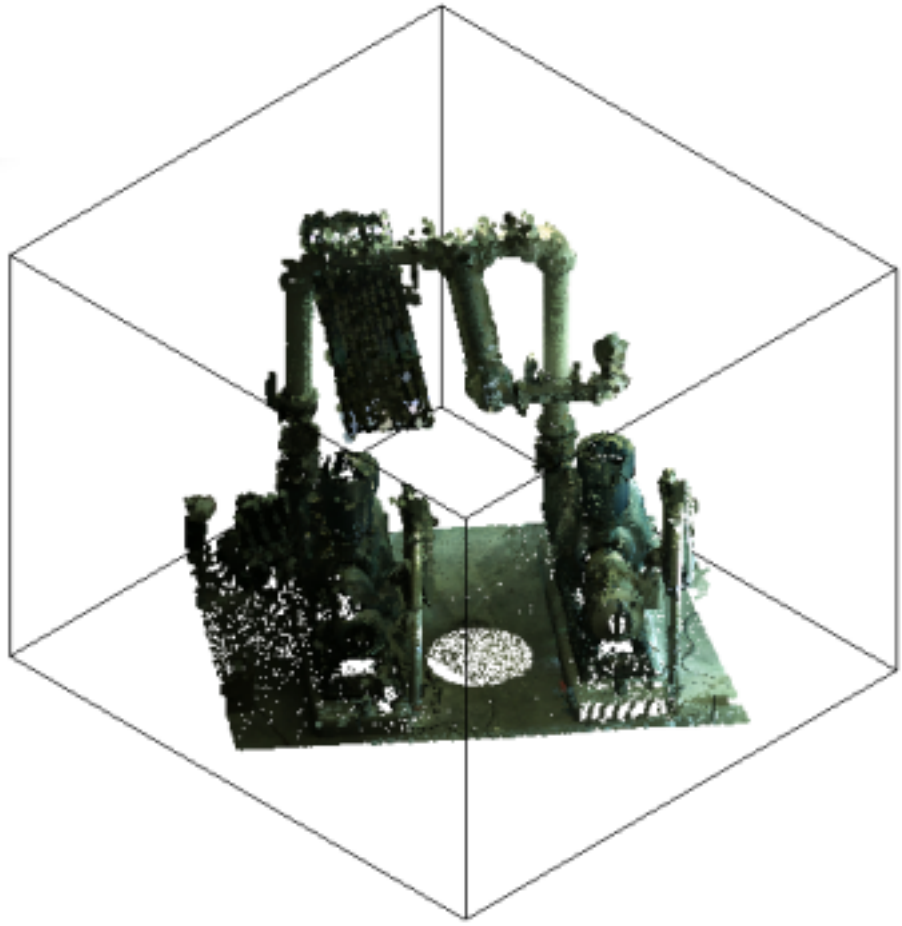


Figure 6.1719



6.24.36 PointCloudColorMap

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Colorizes the point cloud.

Accessing the Command

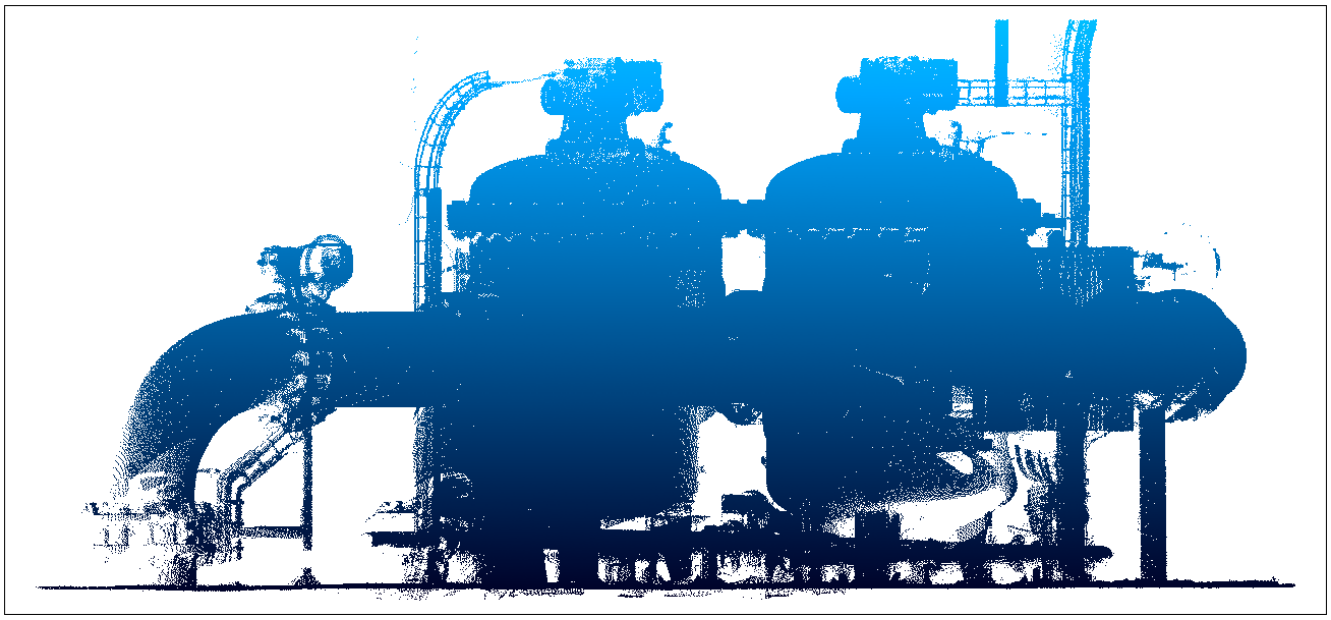
 Keyboard: **pointcloudcolormap** Toolbar: **Point Cloud :PointCloudColorMap Prompts you:** Role [Height/Intensity] <Height>: **(Enter h or i)** Color map [Scan/Jet/Earth/Hydro/grAy/Reds/Greens/Blues] <Scan>: **(Enter an option)**

Command Options

Option	Description
Role	Determines if colors are assigned: • **Height** - colors assigned by the height of the points • **Intensity** -colors assigned according to their intensity value When the point cloud file contains no intensity data, then the colors are assigned by height, irregardless of which option you select.
Color map	Specifies the range of colors to assign points: • **Scan** - assigns colors according to color data collected by the scan • **Jet** - assigns a range of colors from dark blue to dark red • **Earth** - assigns a range of colors from light brown to dark brown • **Hydro** -assigns a range of colors from light to dark blue • **grAy**- assigns a range of colors from light gray to dark gray • **Reds**- assigns a range of colors from light to dark red • **Greens**- assigns a range of colors from light to dark green • **Blues** - assigns a range of colors from light to dark blue

When the command ends, the program colorizes the point cloud, such as the ”Hydro” one illustrated below. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pointcloudcolormap-example.png})

Figure 6.1720



Related Commands

PointCloudAttach (6.24.34) - opens a BPT point cloud file, and then attaches it to the current drawing. PointCloudPointSize (6.24.39) - specifies the size of points. PointCloudPointSize_Minus (6.24.40) - decreases the size of points. PointCloudPointSize_Plus (6.24.59) - increases the size of points. PointCloudPreprocess (6.24.60) - converts point cloud files to BricsCAD BPT format.

Figure 6.1721



6.24.37 PointCloudPointMax

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) This command sets the maximum amount of points shown on screen per point cloud. Since a screen only contains a few million pixels, it makes no sense to display more points at a given moment. Lowering this value results in displaying less detail but faster response. This does not mean that the actual point cloud cannot contain more points!! That latest number can be typically several hundred millions and even larger, and is not affected by this setting.

Accessing the Command

Keyboard:**pointcloudpointmax : pointcloudpointmax Prompts you:** Enter new value for POINTCLOUDPOINTMAX <1000000>: **(Type a value or press Enter to accept the current value.)**

Command Options

No command options.

Settings

Name	Title	Description
PointCloudPointMax	Point cloud max points	Maximum number of points displayed per point cloud. Values between100 000 and 20 000 000 are accepted.

Related Commands

PointCloud (6.24.32) - displays the Point Cloud section of the Drawing Explorer. PointCloudAttach (6.24.34) - attaches point cloud files (that were previously converted to BricsCAD's BPT format) to the current drawing. PointCloudPointSize (6.24.39) - specifies the size of points. PointCloudPointSize_Minus (6.24.40) - decreases the size of points. PointCloudPointSize_Plus (6.24.59) - increases the size of points.

Figure 6.1722



6.24.38 PointCloudCrop

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Crops a point cloud entity, displaying only the part of the point cloud inside or outside a prismatic or cylindrical volume.

Accessing the Command

Keyboard:**pointcloudcrop : pointcloudcrop Prompts you:** Select point cloud entity:**(Select a point cloud.)** Crop option [Rectangular/Circular/Polygonal/Invert/showToggle/removeLast/removeAll] <Rectangular>: **(Choose an option or press Enter to create a rectangular cropping.)**

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/ uncropped point cloud	(\vref {fig:C:/Dev/Task- rectangular cropping inside	(\vref {fig:C:/Dev/Task- rectangular cropping outside

Note: Multiple cropping volumes can be combined.

Command Options

Option	Description
Rectangular	Creates a rectangular prismatic cropping. Prompts you: **First corner point** - **(specify a point.) Specify opposite corner**- **(specify a point.) Inside? [Inside/Outside] <Inside>** - **(choose an option.)** • **Inside**: displays the part of the point cloud inside the rectangular prismatic volume only. • **Outside**:displays the part of the point cloud outside the rectangular prismatic volume only.
Circular	Creates a cylindrical cropping. Prompts you: **Circle center point** - **(specify a point.) Point on circle circumference** - **(specify a point.) Inside? [Inside/Outside] <Inside>**- **(choose an option.)** • **Inside**: displays the part of the point cloud inside the cylindrical volume only. • **Outside**:displays the part of the point cloud outside the cylindrical volume only.
Polygonal	Creates a polygonal prismatic cropping. Prompts you: **Specify first point**- **(specify a point.) Specify next point** - **(specify a point.) Specify next point** - **(specify a point.) Specify next point** - **(specify more points or press Enter to stop.) Inside? [Inside/Outside] <Inside>**- **(choose an option.)** • **Inside**: displays the part of the point cloud inside the polygonal prismatic volume only. • **Outside**:displays the part of the point cloud outside the polygonal prismatic volume only.
Invert	Inverts the current cropping; toggles the cropping inside/outside and vice versa.
Show Toggle	Toggles the current cropping on/off.
Remove Last	Removes the most recently added cropping volume.
Remove All	Removes all cropping

Figure 6.1723

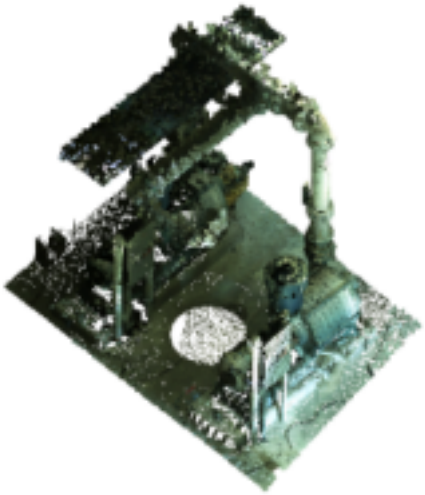


Figure 6.1724



Related Commands

PointCloud (6.24.32)- opens the Drawing Explorer to the Point Cloud section; converts, loads, and unloads point cloud file attachments. PointCloudBoundary (6.24.35) -sets the display of the bounding box of point clouds.

Figure 6.1725



Figure 6.1726



6.24.39 PointCloudPointSize

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the size of points in point clouds.

Accessing the Command

Keyboard: **pointcloudpointsize :PointCloudPointSize Prompts you:** Set point size: **(Enter a number)**

Command Options

Set point size	Specify a number from 0 and larger. The larger the number, the larger each point. Very large numbers can overwhelm the display so that you can no longer see details in the point cloud.

When the command ends, the program displays all points in the new size. You may need to zoom in to see the change.

Related Commands

PointCloudAttach (6.24.34) - opens a BPT point cloud file, and then attaches it to the current drawing. PointCloudPointMax (6.24.37) - defines the maximum amount of points shown per point cloud. PointCloudPointSize_Minus (6.24.40) - decreases the size of points. PointCloudPointSize_Plus (6.24.59) - increases the size of points. PointCloudPreprocess (6.24.60) -converts point cloud files to BricsCAD BPT format.

Figure 6.1727



6.24.40 PointCloudPointSize_Minus

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Decreases the size of points in point clouds.

Accessing the Command

 Keyboard: **pointcloudpointsize_minus** Toolbar: **Point Cloud :PointCloudpointsize_minus Prompts you:** Decreased point size to 1.000000

Command Options

None. This command reduces the size of each point in the cloud by one size.

Related Commands

PointCloudAttach (6.24.34) - opens a BPT point cloud file, and then attaches it to the current drawing. PointCloud-PointMax (6.24.37) - defines the maximum amount of points shown per point cloud. PointCloudPointSize (6.24.39) - specifies the size of points. PointCloudPointSize_Plus (6.24.59) - increases the size of points. PointCloudPrepro-cess (6.24.60) -converts point cloud files to BricsCAD BPT format.

Figure 6.1728



6.24.41 PasteOrig

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Pastes CAD entities from the Clipboard into a different drawing using the coordinates of the entities in the source drawing (short for "paste at original coordinates"). **Note:**Before you can use this command, first use the the CopyClip (6.9.52) orCopyBase (6.9.51) commands to copy entities from the current drawing.

Accessing the Command

 command bar: **pasteorig** menu bar: **Edit | Paste to original coordinates** |ribbon:**Home | Edit | Paste | Paste to Original Coordinates** Switch to another drawing before using the command: : **PasteOrig** There is no prompt. The program pastes the CAD content from the Clipboard into the target drawing using the entities' coordinates. For instance, a circle with center at 2,3 is pasted into the target drawing with its center at 2,3. This command does not paste entities back into the source drawing.

Command Options

None.

Procedures

- Using Copy and Pasteorig to copy entities between drawings (1.23.17)

Related Commands

CopyBase (6.9.51)- copies selected entities to the Clipboard with a base point. CopyClip (6.9.52) - copies selected entities to the Clipboard. CopyHist (6.9.54) - copies all text from the Text window to the Clipboard. CutClip (6.9.59) - cuts (moves) selected entities to the Clipboard. PasteBlock (6.24.9) - pasted entities from the Clipboard as a block at an insertion point. PasteSpec (6.24.11) - displays the Paste Special dialog box for controlling the format of pasted entities.

Figure 6.1729



6.24.42 Pointlight

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates a point light for use in renderings, placing this glyph in the drawing:  **Note:** A point light spreads illumination in all directions, like a bare lightbulb, and so has no target.

Accessing the Command

 command bar: **pointlight** menu bar: **View | Rendering | Lights | New point light** ribbon:**View | Lights | Create | New Point Light** toolbar: **Lights |**  : **Pointlight Prompts you in the command bar:** Specify source position <0,0,0>: **(Specify the position of the light)**

- When the LIGHTINGUNITS (1.26.4) system variable is 0 (no lighting units), the following prompt displays:

Enter an option to change [Name/Intensity/Status/shadoW/Attenuation/Color/eXit] <eXit>:**(Enter an option,or press Enter to end the command)**

- If the LIGHTINGUNITS (1.26.4) system variable is 1 (American lighting units) or 2 (international lighting units), the following prompt displays:

Enter an option to change [Name/Intensity factor/ Status/ Photometry/ shadoW/ Attenuation /filterColor/ eXit] <eXit>:**(Enter an option,or press Enter to end the command)**

Command Options

Option	Description
Specify source position	Specifies the position of the light: • Pick a point • Enter x,y,z coordinates • Press **Enter** to accept the default location
Name	Specifies a name for the light so that it can be identified by the LightList command; prompts you: **Enter light name <Pointlight1>** - **(enter an option)**: • Enter a name • Press **Enter** to accept the default name
Intensity FactorIntensity	Specifies the intensity of the light; prompts you: **Enter intensity (0.00 - max float) <1.0>** - **(enter an option)**: • Enter an intensity values between 0.00 and the maximum value supported by your system • Press **Enter** to accept the default value

Status	Toggles the use of this light; prompts you: **Enter status [ON/OFf] <ON>** - **(enter an option)**: • **ON** - light is included when calculating renderings • **OFf** - light is excluded
Photometry	Specifies the photometric properties of the light; prompts you: **Enter a photometric option to change [Intensity/Color/eXit] <Intensity>** - **(enter an option)**: • **Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0>**: - **(enter the intensity expressed in candela or an option)**: **Enter Flux (Lm)**- **(enter the luminous flux expressed in lumens.) Enter Illuminance (Lx or Fc) or enter an option [Distance] <9424.8>**- **(enter the illuminance expressed in lux or foot-candles.) Enter Distance** - **(Enter a distance expressed in drawing units.)** • **Enter color name or enter an option [?/Kelvin] <D65>**- **(enter a color name.) ?** - lists the names of available colors, such as: D65 Fluorescent Coolwhite Whitefluorescent daylightfluorescenT Incandescent Xenon Halogen Quartz Metalhalide mErcury Phosphormercury highpressureSodium Lowpressuresodium **Enter Kelvin temperature <3600.0>** - **(enter a color temperature expressed in degrees Kelvin.)**

Shadow	Specifies the look of shadows cast by this light; prompts you: **Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled] <Sharp>** - **(enter an option)**: • **Off** - disables the calculation of shadows for the light. • **Sharp** - displays shadows with sharp edges. Use this option to increase performance. • **Soft mapped** - displays realistic shadows with soft edges. Prompts you: Enter map size [64/128/256/512/1024/2048/4096] <256>: • **Soft sampled** : Displays realistic shadows with softer shadows based on extended light sources; prompts you: **Enter an option to change Shape/sAmples/Visible/<eXit>** - **(enter an option)**: **Enter shape [Disk/Rect] <Disk>** - **(type D or R)** **Enter radius <0.0>** - enter the radius of the disk **Enter Width <0.0>** - enter width of the rectangle **Enter Length <0.0>** - enter length of the rectangle • **Enter Shadow Sample <16>** - **(specify the sampling size of the shadow; larger numbers are more accurate but take longer to render.)** • **Enter Shape Visibility [Yes/No] <No>** - **(type Y or N to cast a shadow that represents shapes (more accurate) or is rectangular (faster to render).)**

Attenuation	Specifies how the illumination falls off with distance from the light: **Enter an option to change [attenuation Type/Use limits/attenuation start Limit/attenuation End limit/eXit] <eXit>** - **(enter an option)**: • **Enter attenuation type [None/Inverse linear/inverse Squared] <None>** - specifies the type of attenuation: **None** - No attenuation so that the distance to the light source has no influence **Inverse linear** - attenuation is the inverse of the linear distance from the light: at a distance of 2 units from the light source, light is half as strong; at a distance of 4 units, light is one quarter as strong **Inverse squared** - attenuation is the inverse of the square of the distance from the light: at a distance of 2 units, light is one quarter as strong; at a distance of 4 units, light is one sixteenth as strong • **Use limits** - toggles whether the extent of the illumination is limited; prompts you: **Limits [OFf/ON]** - **(enter an option)**: **ON**- limit the extent **OFf** - do not limit it • **attenuation start Limit** - defines the point where the light starts to shine, measured from the center of the light; prompts you: **Specify start limit offset <1.0>** - **(enter a distance.)** • **attenuation End limit** - defines the point where the light stops to shine, measured from the center of the light; prompts you: **Specify end limit offset <10.0>** - **(enter a distance.)**
colorFilter	Specifies the color of the light; prompts you: **Enter true color (R,G,B) or enter an option [Index color/Hsl]:<255,255,255>** - **(enter an option)**: • **Enter true color** - the RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The name of the model comes from the initials of the three additive primary colors, red, green, and blue. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values). • **Index color** - specifies an AutoCAD Color Index color. • **Hsl** - specifies the color using three parameters Hue, Saturation and Lightness
eXit	Concludes the command and creates the light.

Definitions

Candela The candela (symbol: cd) is the SI base unit of luminous intensity; that is, power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function). A common candle emits light with a luminous intensity of roughly one candela. **Lumen** The lumen (symbol: Lm) is the SI unit of luminous flux, a measure of the power of light perceived by the human eye. Luminous flux differs from radiant flux in that luminous flux measurements (such as lumens) are intended to reflect the varying sensitivity of the human eye to different wavelengths of light, while radiant flux measurements (such as watts) indicate the total power of light emitted. The lumen is defined in relation to the candela as: 1 lm = 1 cd·sr As a full sphere has a solid angle of 4·pi steradians, a light source that uniformly radiates one candela in all directions has a total luminous flux of 1 cd·4pi sr = 4pi ≈ 12.57 lumens. For example, if a candle emits light with a luminous intensity of about 1 candela, the total light emitted is about 12.6 lumens. **Illuminance** In photometry, illuminance is the total luminous flux incident on a surface, per unit area. It is a measure of the intensity of the incident light, wavelength-weighted by the luminosity function to correlate with human brightness perception. **LIGHTINGUNITS**=2 International lighting units usesSI units derived units illuminance is measured in lux (lx) or lumens per square metre. **LIGHTINGUNITS**=1 American lighting units use the foot-candle (Fc) is a non-metric unit of illuminance that is used in photography. The unit is defined as the amount of illumination the inside surface of a 1-foot radius sphere would be receiving if there were a uniform point source of one candela in the exact center of the sphere. Alternatively, it can be defined as the illuminance on a 1-square foot surface of which there is a uniformly distributed flux of one lumen. This can be thought of as the amount of light that actually falls on a given surface. The foot-candle is equal to one lumen per square foot. One foot-candle is equal to approximately 10.764 lux, although in the lighting industry, typically this is approximated as 1 foot-candle being equal to 10 lux. **Kelvin Temperature** Color temperature is a characteristic of visible light that has important applications in lighting, photography, videography, publishing, manufacturing, astrophysics, and other fields. The color temperature of a light source is the temperature of an ideal black-body radiator that radiates light of comparable hue to that light source. The temperature is conventionally stated in units of absolute temperature: degrees kelvin (K). Higher color temperatures (5,000 K or more) are called cool colors (blueish white); lower color temperatures (2,700 – 3,000 K) are called warm colors (yellowish white through red).

Related Commands

Distantlight (6.10.32) - creates a distant light. Light (6.20.28) - creates a new light in the drawing. Lightlist (6.20.29) - displays the Drawing Explorer - Lights dialog window. Render (6.26.22) - calculates a photorealistic image of the 3D model. Spotlight (6.27.42) - creates a spot light. Sunproperties (6.27.57) - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog. Weblight (6.33.6) - creates a web light.

Figure 6.1730



6.24.43 Polygon

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws regular polygons, from 3 to 1,024 sides. **Note:**This command draws regular polygons, where each side is the same length, and are made of polylines.To draw irregular polygons, use the Rectang (6.26.6) or PLine (6.24.27) commands. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image369.png})

Accessing the Command

 command bar: **polygon** alias: **pol** menu bar: **Draw | Polygon** ribbon:**Draw | Draw | Polygons | Polygon** toolbar: **Draw | Polygon** |  : **Polygon Prompts you in the command bar:** Polygon: Set number of sides or [Width of line/Multiple polygons] <4>: **(Specify the number of sides, press Enter for the default number of sides, or elseenter an option)** This command draws polygons by several different methods:

- Location of center point and the midpoint of a side; **prompts you**:

Specify by: Set center of polygon or [specify by Edge]:**(Pick a point for the center)** Specify by: Select midpoint of side or [specify by Vertex]:**(Pick a point at the midpoint of one side)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel

- Length of one edge by entering the **E** option:

First endpoint of edge: **(Pick a point)** Second endpoint of edge: **(Pick anotherpoint)** (\vref {fig:C:/Dev/Task-2448/images/static

- Location of center point and one vertex:

Specify by: Select midpoint of side or [specify by Vertex]: **(Type V)** Specify by: Select vertex point or [specify by Side]: **(Pick a point at what should be one of the vertices)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cr

Figure 6.1731

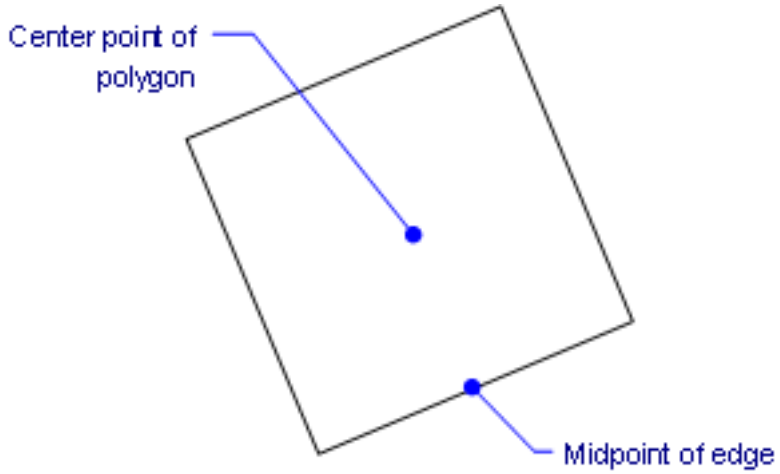


Figure 6.1732

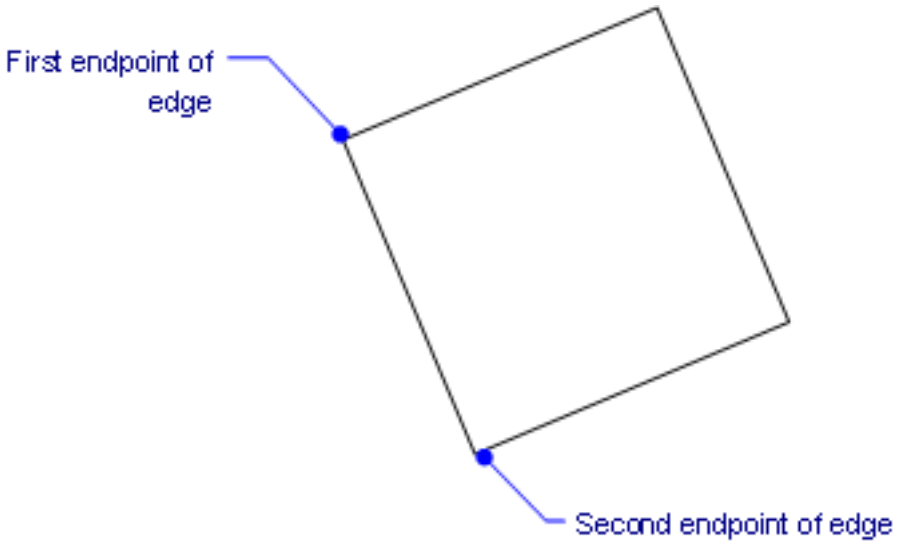
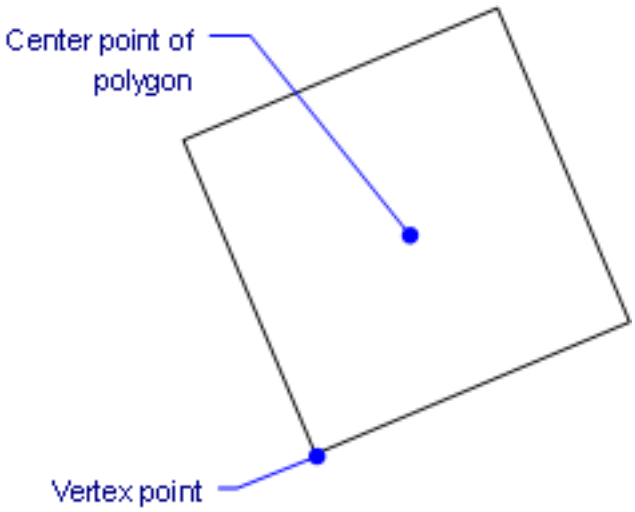


Figure 6.1733



Command Options

Option	Description
Number of sides	Specifies the number of sides; enter a number between 3 and 1024.
Multiple polygons	Repeats the command to draw additional polygons.
Width of line	Specifies the width of the edge lines.
Center of polygon	Specifies the center point of the polygon.
Midpoint of side	Specifies the midpoint of one of the sides: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i The midpoint of the side determines the size and angle of the polygon.
Edge	Specifies the length of an edge; prompts you: 1. **First endpoint of edge** - **(specify the edge's start point.)** 2. **Second endpoint of edge** - **(specify the edge's endpoint.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i The length of the side determines the size and angle of the polygon.
Vertex	Specifies a vertex (corner of the polygon); prompts you: **Select vertex point** - **(specify the vertex.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i The location of the vertex determines the size and angle of the polygon.

Procedures

Drawing polygons

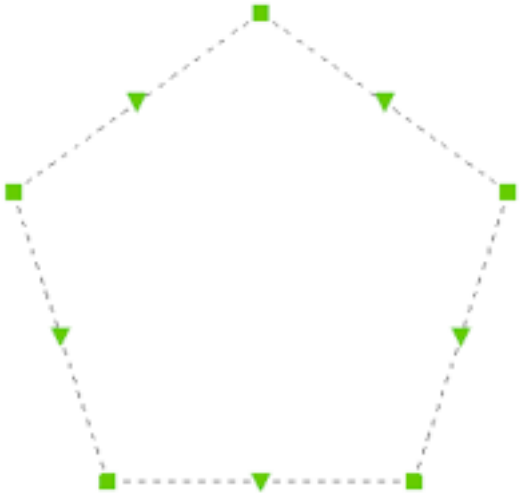
Grips Editing

Polygons can be edited directly through grips:

1. Select the polygon. Notice that it has a square grip at each vertex and a triangular grip at the midpoint of each segment. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/PolygonGri
2. Drag a square grip to move a vertex.
3. Drag a triangular grip to move a segment.

Notice that the polygon is no longer regular.

Figure 6.1734



Related Commands

Rectang (6.26.6) - draws rectangles and squares. PEdit (6.24.20) - edits polygons.

Figure 6.1735

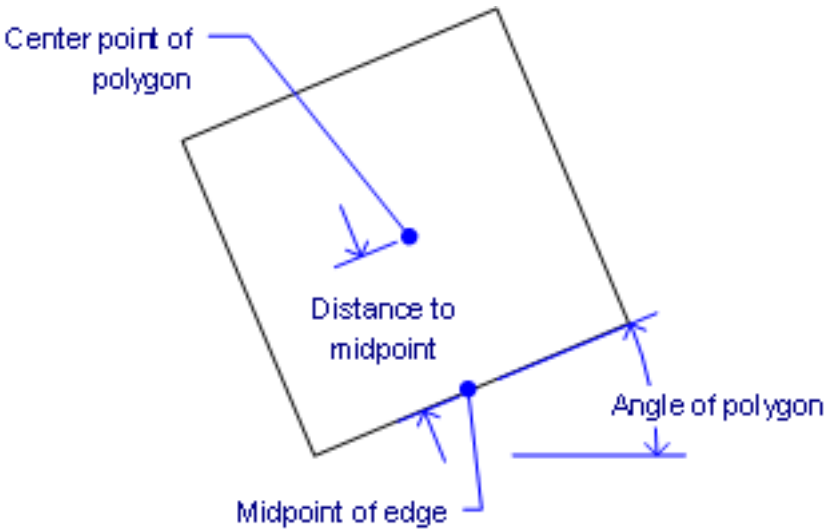


Figure 6.1736

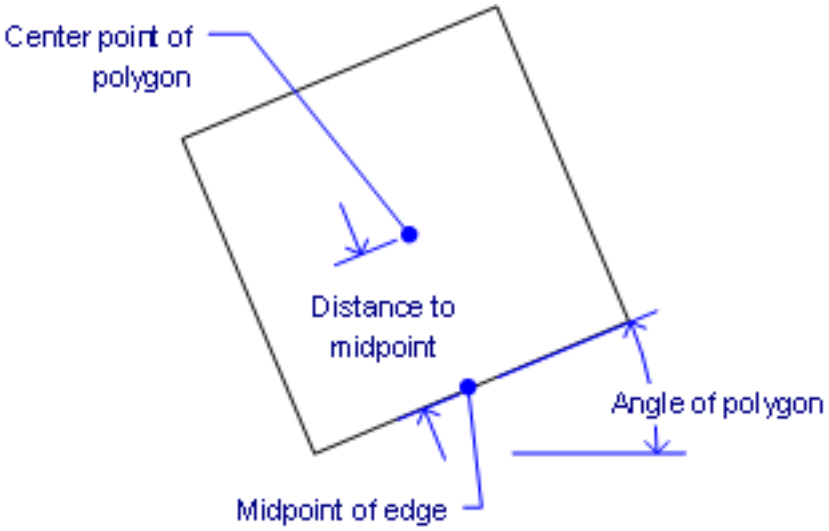


Figure 6.1737

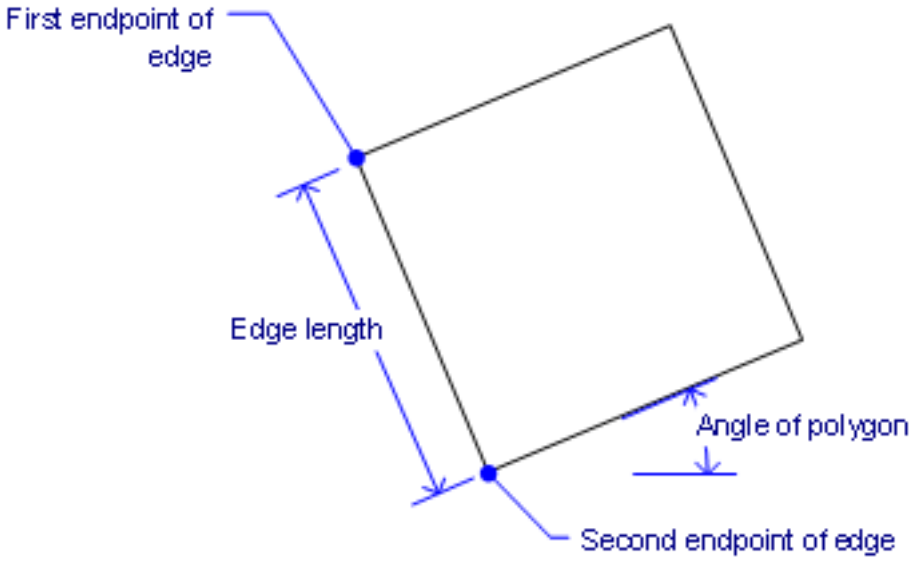


Figure 6.1738

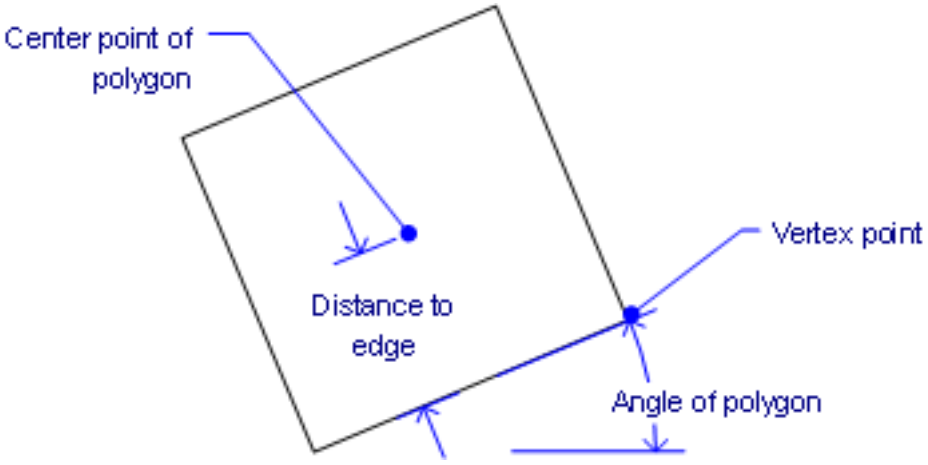
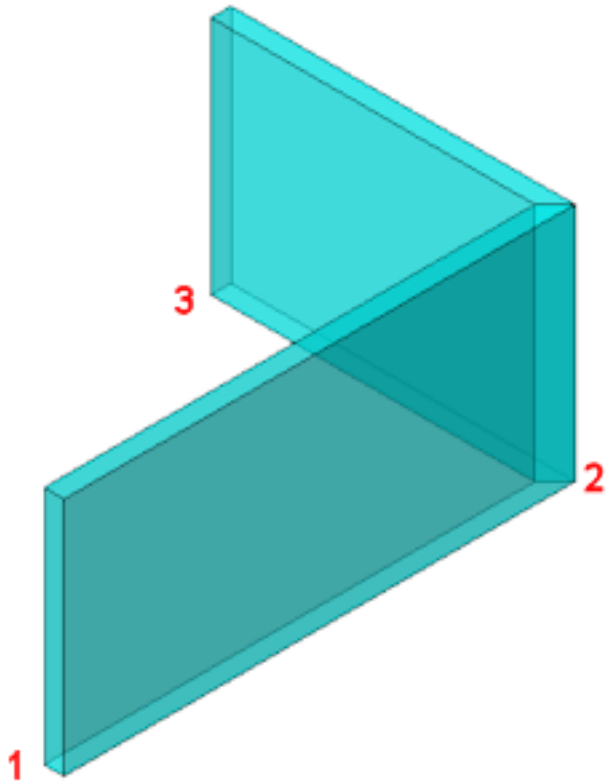


Figure 6.1739



Figure 6.1740



6.24.44 PolySolid

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Quickly 3D solids that look like walls and roofs. (\vref {fig:C:/Dev/Task-2448/images/wwwbrics

Accessing the Command

 keyboard: **polysolid** alias: **pso** menu bar: **Model | 3D Solids | Polysolid** ribbon: **Model | Create 3D** toolbar: **Solids** quad tab: **Model** : **PolySolid Prompts you in the command bar:** Start point or [Height/Width/Justification/Entity/Separa solids/Dynamic] <Entity>: **(Specify a point, such as #1 in the figure above, or enter an option)** Set next point or [draw Arcs/Distance]: **(Specify a point, such as #2, or enter an option)** Set next point or [draw Arcs/Distance/Follow/Undo]: **(Specify a point, such as #3, or press Enter to specify the height)** Set next point or [draw Arcs/Close/Distance/Follow/Undo]: **(Specify a point, enter an option, or press Enter)** Height of polysolid <8.00>: **(Enter a height, or else press Enter accept the default height)** The height of the polysolid is perpendicular to the XY-plane of the current **coordinate system** (WCS or UCS) or dynamic UCS.

Command Options

Option	Description
Start point	Pick the starting point of the polysolid, or enter x,y coordinates.
Set next point	Pick the next vertex of the polysolid,or enter x,y coordinates.

Height	Sets the height of the polysolid; prompts you: **Height of polysolid <8>** - **(enter a value.)** The default height is saved to the PSOLHEIGHT variable for reuse by this command.This default height is used when the Dynamic option is off.
Width	Sets the width of the polysolid; prompts you: **Width of polysolid <0.5>**- **(enter a value.)** The default width is saved to the PSOLWIDTH system variable forreuse by this command.
Justification	Defines the position of the polysolid entity with respect to the base line; prompts you: **Select justification [Left/Right/Center] <Center>** - **(choose an option)**: • **Left** - the width of the polysolid is drawn to the left of the center line, which is defined by your pick points • **Right** - the width of the polysolid is drawn to the left of the center line • **Center** - width is drawn on both side of the center line; this is the default justification When Hot Key Assistant is on (**HKA** on the status bar), then the following widget appears on the screen; tap the **Ctrl** key to switch dynamically between left, center, and justification: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P
Entity	Draws the polysolid along an existing 2D entity; prompts you: **Select polysolid base** - **(select a line, open or closed polyline, arc, circle, ellipse, elliptical arc, or splines.)**
Separate solids	Determines whether the 3D solids drawn by this command are created separately; prompts you: **Separate solids On/OFF <On>** - **(enter an option)**: • **On** - draws separate solids between vertices (at left, below) • **OFF** -draws the entire polysolid as a single, multi-segment entity; bisector connection lines are removed in sections depending on the Union Section (2.6.10) property of the material(s) in the composition that is assigned to the connected solids (at right, below (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P

Dynamic	Determines whether the height of the polysolid is fixed; prompts you: **Dynamic height On/OFF <On>** - **(enter an option)**: • **On** - you will be prompted to specify the height of the polysolid entity; prompts you: **Height of polysolid <8.00>** - **(enter a value, press Enter to accept the default height; or click a point in the drawing.)** • **OFF** - uses the the height defined by the PSOLHEIGHT (1.13.11)variable You can also use dynamic dimensions (**DYN** on the status bar) todefine the width and height of the polysolid in the dynamic entry fields; press the **Tab**key to switch to the other field, and then press **Enter** to confirm the values. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P
draw Arcs	Starts drawing arc segments; prompts you: **Set end of arc or [draw Lines/CLose/Direction/Second point/Undo]** - **(enter an option)**: • **End of arc** - enter a point to make the arc tangent to the previous segment • **Draw Lines** - switches back to drawing linear segments • **Direction** - specifies the direction of the tangent at the start point of the new arc segment • **Second point** - prompts you to specify a point along the arc's circumference
Distance	Specifies the length and angle of a linear segment; prompts you: 1. **Distance for segment** - **(type a value.)** 2. **Angle of segment** - **(type a value.)**
Follow	When switching from arc segments to linear segments, creates the linear segment tangent to the arc segment.
Close	Creates a closing segment between the previous point and the start point of the polysolid.If you snap to the start point of the currently created chain, the chain is closed automatically.This option is available after creating two polysolid segments.
Undo	Removes the previous segment.

Procedures

- Creating a single wall (2.4.9)
- Creating a chain of L-connected walls (6.20.19)

Related Commands

BimConnect (6.20.19) - Creates an L-connection between minor faces of two solids. BimDrag (6.7.8) - drags major faces of solids while preserving connections with minor faces. Drags minor faces; optionally connects minor faces to major faces of other solids. BimQuickDraw (6.7.42) -Allows to quickly create and edit rectangular and L-shaped rooms and buildings.

Figure 6.1741



Figure 6.1742

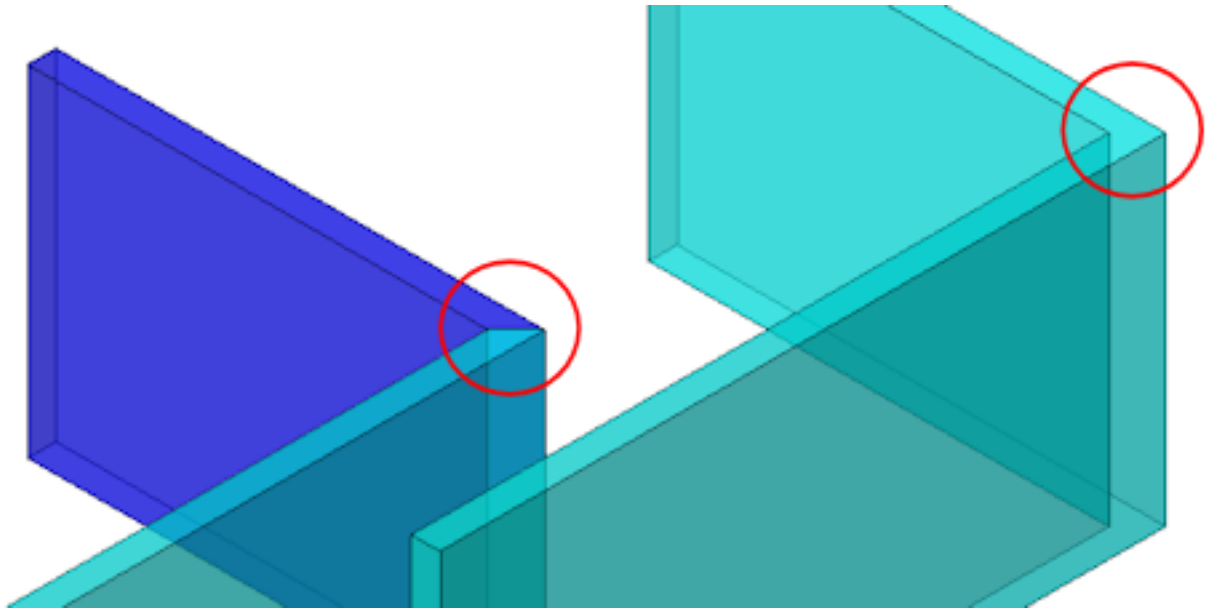


Figure 6.1743

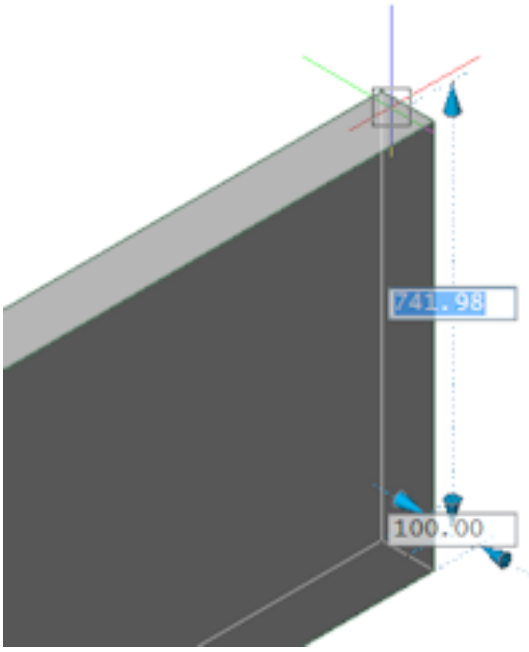


Figure 6.1744



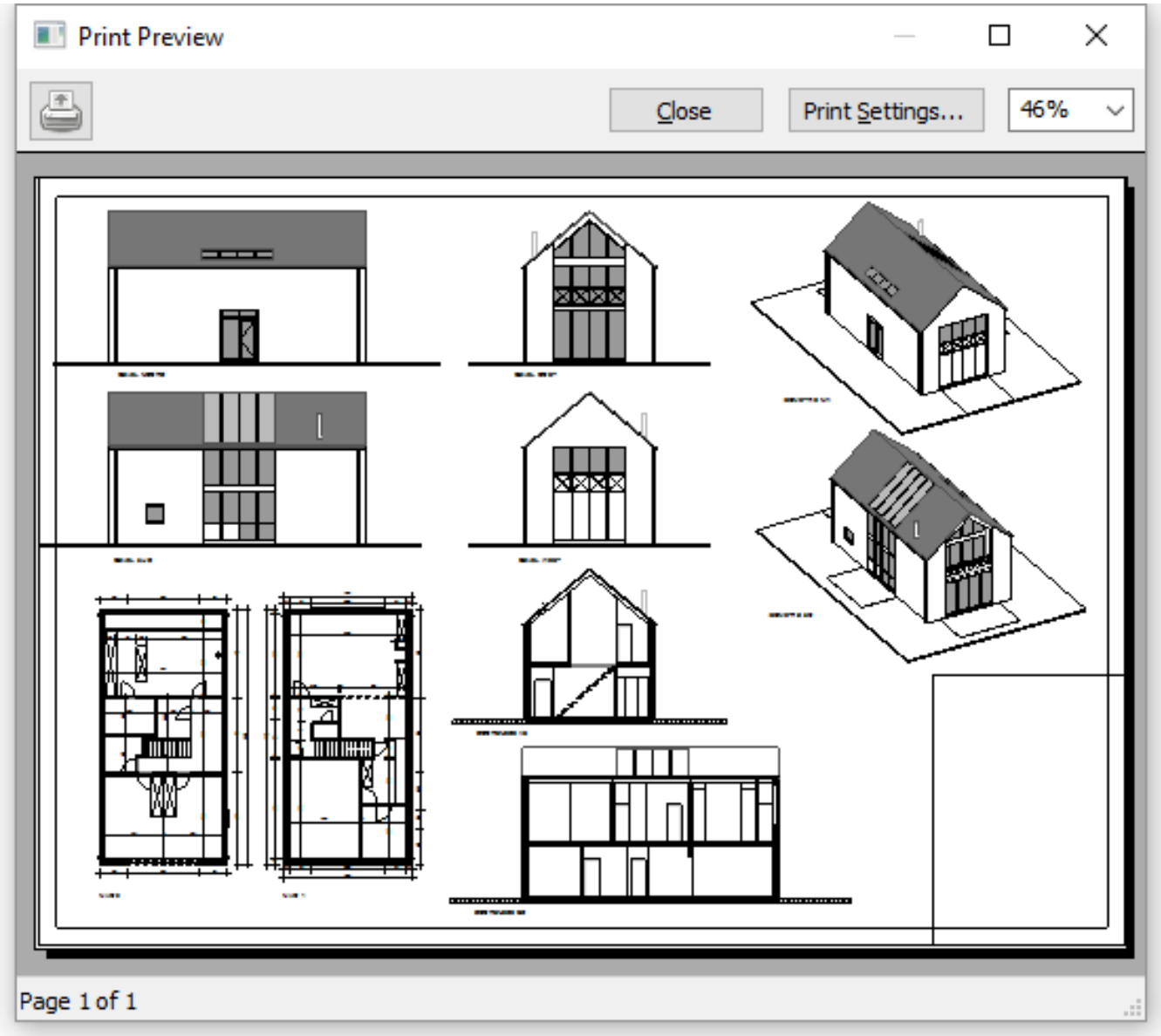
6.24.45 Preview

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Displays the Preview window so that you can check how the drawing looks before you plot it. **Note:** This command does not work with the "None" printer; use the Print (6.24.46) orPageSetup (6.24.1) command to first specify a printer.

Accessing the Command


command bar: **preview** aliases: **ppreview** **pre** menu bar: **File | Print Preview** ribbon: **Home | Print/Plot | Preview** toolbar: **Standard |**  : **Preview** Displays a window: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU Use the mouse scroll wheel to zoom, and use the scroll bars to pan. Click **Print** to plot the drawing, or click **Close** to return to BricsCAD.

Figure 6.1745



Command Options

Option	Description
Print	Prints the drawing; does not display the Print dialog box. See Print (6.24.46) command.
Close	Closes the preview window and returns to the BricsCAD drawing window.
Print Settings	Displays the Print Settings dialog box. See PageSetup (6.24.1) command.
Zoom	Changes the size of the preview; use the scroll bars to pan.

Related Commands

Print (6.24.46) - prints drawings through a dialog box. Plot (6.24.26) - plots drawings through the command bar.

6.24.46 Print

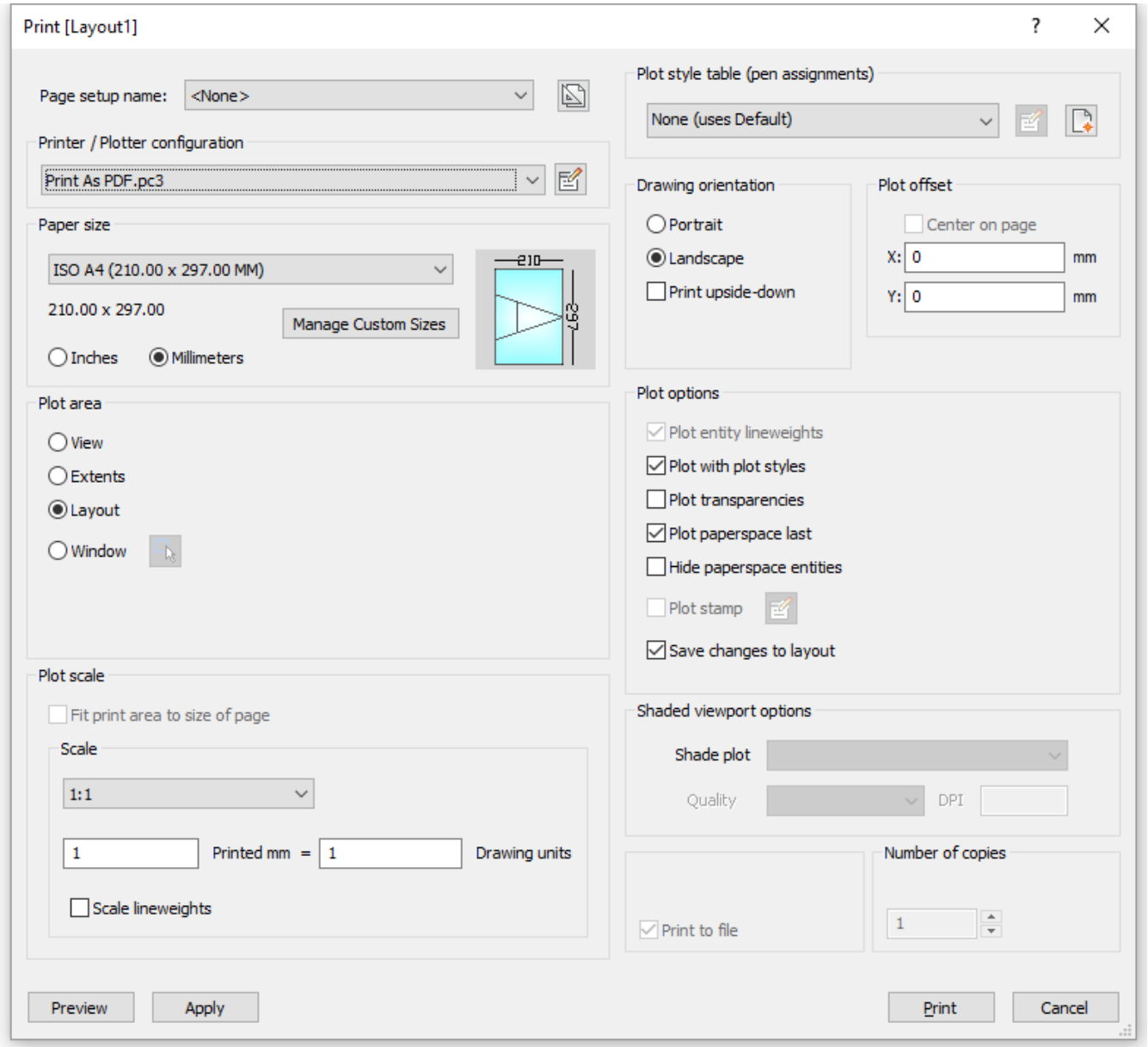
Prints and previews drawings to plotters and files through a dialog box. **Note:**When a selection set is activeupon starting this command, the selection set is printed.

Accessing the Command


command bar: **print** shortcut: **Ctrl+P** menu bar: **File | Print** ribbon:**Home | Print/Plot | Print** toolbar: **Standard | Print** :**Print** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sour

1. Choose options.
2. Click**Preview**to preview the plot on simulated paper.
3. Click**Apply**to apply the changes.
4. Click**Print**to print the drawing, or click**Cancel**to close the Print dialog box.

Figure 6.1746



Command Options

Option	Description
Page Setup Name	Recalls plot settings used previously: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P • **<None>**- uses options saved in the current layout or model space • **<Previous Plot>**- uses options that were saved from the last time this dialog box was used • Names of other page setups stored in the drawing
 Save current settings to page setup	Create new page setups; see thePageSetup (6.24.1)command.
Printer/Plotter Configuration	Selects the printer or plotter. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i The program works with any output device installed on the system, including networked printers, print to file, faxes, and PostScript devices -- as well aspredefined printer parameters stored in PC3 files.
 Edit plotter configuration	Opens the printer's Properties dialog box for customizing the printer's parameters and creating PC3 files; see thePlotterManager (6.24.30)command.
Paper Size	Specifies the size of paper. You can select standard sizes listed by the droplist; these are sizes supported by the printer. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i While printer may appear to support many different sizes, you should select only the size of paper that is actually in the printer. IfUser Definedappears in the list, the paper's dimensions are specified by the printer's Properties dialog box. The image indicates the size, position and orientation of the current plot area on the selected paper size. • **Inches** - uses imperial units for measurements involving printing • **Millimeters** -uses metric units for measurements involving printing **Manage Custom Sizes** -(PDF output only.) Manages custom paper sizes whenPrint as PDF.pc3is selected in thePrinter/Plotter Configurationlist; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C

Plot Area

Specifies which area of the drawing to print: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

- **View**- prints the current view or a named view. Choose a view from the droplist; use the View (6.31.1) command to create named views.
- **Extents**- prints the extents of the drawing, which ensures every visible entity is printed. Entities on frozen layers are not taken into account when calculating the extents.
- **Limits**- prints the limits of the drawing, as specified by the Limits command.
- **Window**- prints a rectangular area of the drawing. You define the rectangular area by entering x,y coordinates or by clicking the**Select Area to be Printed**  button:

1. Click the button; you are prompted:
2. **Select first corner of window**- **(specify one corner of the rectangle by picking points orentering x,y coordinates.)**
3. **Select second corner of window**- **(specify the other corner.)**

The Print dialog box returns, with the pair of x,y coordinates added to the Windows Plot Area boxes; you can edit the values: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Plot Scale

Scales the drawing to fit it to the paper: **Fit print area to the size of paper** -scale is calculated automatically by the program, taking into account the print area of the drawing. When this option is turned on, you cannot specify the scale factor.:

- **Printable area** is the size of the paper, minus the margins.
- **Margins** are the strips along the four edges that the printer uses for handling the paper.

Scale - allows you to specify the scale factor to use for plotting; choose a scale factor from the droplist, or choose **Custom**. The scale factor shown can be edited withtheScaleListEdit (6.27.7)command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

- For scale factors like 1:5, the drawing is printed smaller.
- For scale factors like 5:1, the drawing is printed larger.

Custom - lets you enter in your own scale factors: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P **Scale lineweights** - when on,lineweights are scaled with respect to the plot scale.

Plot Style Table (Pen Assignment)

Specifies the plot style table to use for the printed output, which assigns properties to ”pens,” colors, and entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

- When the drawing does not use plot styles, then only CTB (color-based table) files are listed.
- When the drawing uses plot styles, then only STB (style-based table) files are listed.

When you switch from**None**to a named plot style, BricsCAD prompts you: **Assign plot style table to all layouts?**- **(enter an option)**:

- **Yes**- assigns the CTB or STB plot style file to all layouts.
- **No**- assigns the plot style file to the current layout only.

 Edit Plot Style

Opens the Plot Style Table editor dialog box; see thePlotStyle (6.24.29)command.This button is available only when the drawing uses STB plot styles.

 Create New Plot Style

Creates new plot styles; runs the Add Plot Style Table wizard; see theStylesManager (6.27.55)command.

- When the drawing does not use named plot styles, then the wizard creates new color-dependent tables (CTB files).
- If the drawing uses named plot styles, then the wizard creates new named plot style tables (STB files).

Drawing Orientation

Specifies the orientation of the drawing on rectangular paper: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

- **Portrait**- The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.
- **Landscape**- The drawing or layout x-axis is aligned with the longest edge of the selected paper size.
- **Print upside down**- Prints the drawing upside-down. This is useful when paper with a drawing border is loaded backwards in the printer.

Plot Offset	Specifies the offset distance for the print. **Center On Page** - centers the print on the page, taking into account margins.Together with**Fit Print Area to Size of Page**, this option is excellent for prints where scale is unimportant. **X** and **Y** - enter positive or negative distances to move the drawing in the x and/or y distances: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Positive values** - moves the drawing up and to the right. • **Negative values** - moves the drawing down and to the left. The lower left corner of the drawing is moved by the specified distance. This is useful when the paper has a title block area that might interfere with the drawing.

Plot Options	**Plot entity lineweights** - toggles the use the lineweights; this option is unavailable whenPlot with Plot Stylesis turned on\: • **On**- mimics lineweights assigned to the drawing in the print. • **Off**- ignores lineweights. **Plot with Plot Styles** - toggles the use of plot styles • **On**- uses plot styles to determine the look of the printed drawing, andoverrides lineweight settings. • **Off**- does not use plot styles. **Plot Transparencies** -plots entities who have been assigned the theTransparency (1.17.8)property, either by-entity or by layer. **Plot Paperspace Last** - specifies theprinting order;this option is available only when printing layouts, and so is unavailable in model tab: • **On**- prints model space entities, followed by paper space entities. • **Off**- prints paper space entities first, followed by entities in model space. **Hide Paperspace Entities** - when on, removes hidden lines from 3D entities in paper space.This option is disabled when printing model space. **Plot Stamp** - toggles use of the plot stamp; click the **Edit Plot Stamp** button to change plot stamp data. See the PlotStamp (6.24.28) command • **On**- applies plot stamp data to the print. • **Off**- does not apply plot stamps. **Save Changes to Layout** - determines if the options you changed in this dialog box are saved;the next time you use this dialog box, you can select ”Layout” from theUse Plot SettingsFrom droplist. • **On**- saves changes made to this dialog box with the layout. • **Off**- does not save changes.
Shaded Viewport Options	Override the visual style of the current view when printing model space: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S **Shade Plot** - choose a shade style plot the drawing with. This option is disabled when printing a paper space layout.TheShade Plotmode of a paper space viewport is defined in theShade Plotproperty of the viewport.The quality of theRenderedoption is defined through the current render preset; see theRenderPresets (6.26.24)command.
Print to File	When checked, redirects the print output to a *.plt file, which can be processed by certain types of software.
Number of Copies	Specifies the number of copies to print.Enter a number or click the buttons to change the value.
Preview	Displays a preview of the print.In the preview window, click Print Settings button to return to this dialog box.See thePreview (6.24.45)command.

Apply	Applies the changes you made to this dialog box. The changes are remembered the next time you use this command.This is like using thePageSetup (6.24.1)command.
Print	Prints the drawing, and then returns to the drawing editor
Cancel	Returns to the drawing editor without printing the drawing.

Procedures

- General procedure for printing (1.25.1)
- To open the Print dialog (1.25.1)
- To choose the paper size (1.25.1)
- To choose the paper units (1.25.1)
- To choose the drawing orientation (1.25.1)
- To define the plot offset (1.25.1)
- To specify the area to plot (1.25.1)
- To define the plot scale (1.25.1)
- To set the plot options (1.25.1)
- To define the plot stamp (1.25.1)
- To specify the number of copies (1.25.1)
- To see a preview of the plot output (1.25.1)

Related Commands

Plot (6.24.26)- plots drawings through the command bar. QPrint (6.25.3)- plots drawings using the current settings. Preview (6.24.45)- previews drawings before they are plotted. ConvertCtb (6.9.42)- converts CTB plot style files to STB files. ConvertPStyles (6.9.46)- converts drawings from CTB to STB plot styles. ExportPdf (6.13.17)- exports layouts in a drawing to a pdf docucment. Layer (6.20.2)- assigns plot styles to layers. PageSetup (6.24.1)- assigns plot style to drawings. PlotStyle (6.24.29)- sets the current plot style. Properties (6.24.49)- changes plot styles assigned to entities. StylesManager (6.27.55)- creates and attaches plot styles to drawings. ScaleListEdit (6.27.7)- edits the scale list of a drawing, used inPlot Scalesection of thePrintandPage Setupdialogs and theStandard scaleproperty of a paper space viewport. Publish (6.24.55)- allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file. -Publish (6.24.56)- prints a saved sheet list (= a list of model space or paper space layouts).

Figure 6.1747



Figure 6.1748

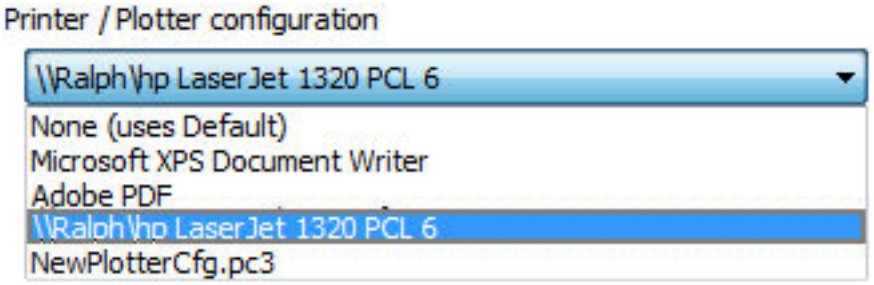


Figure 6.1749

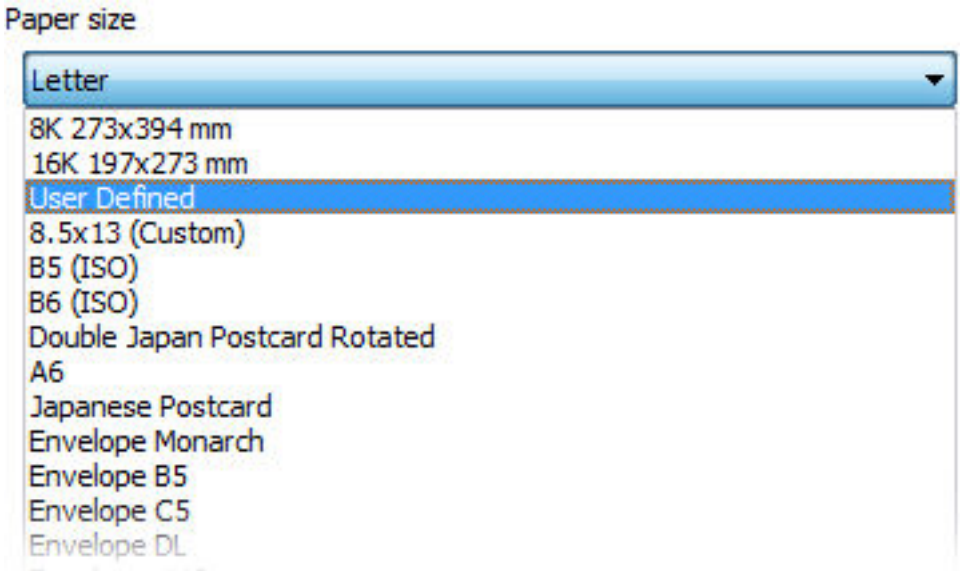


Figure 6.1750

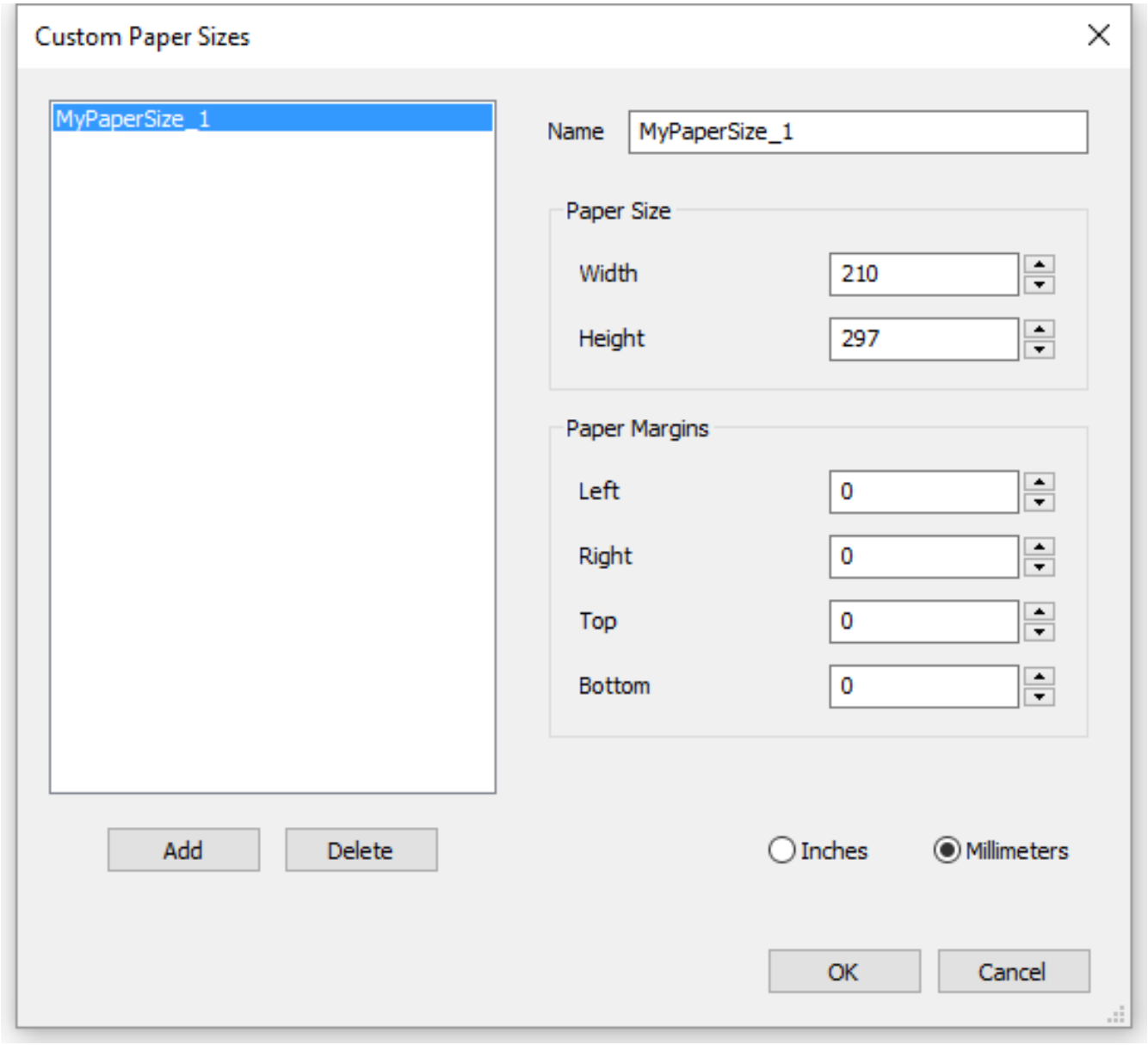


Figure 6.1751

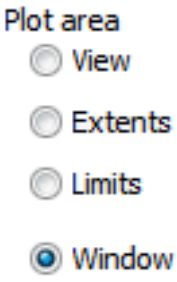


Figure 6.1752

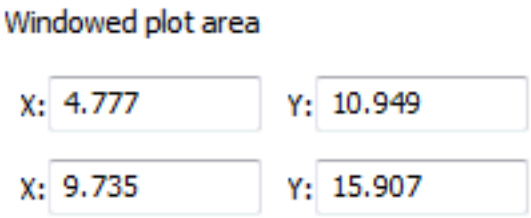


Figure 6.1753

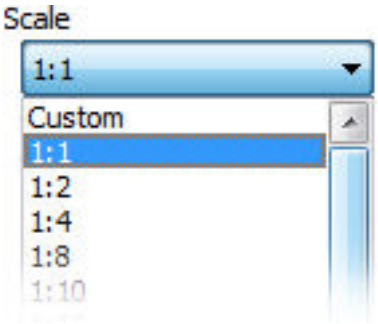


Figure 6.1754

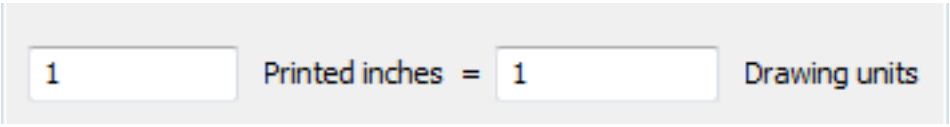


Figure 6.1755

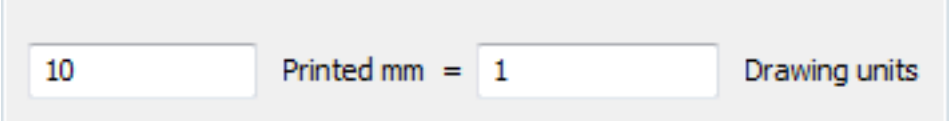


Figure 6.1756

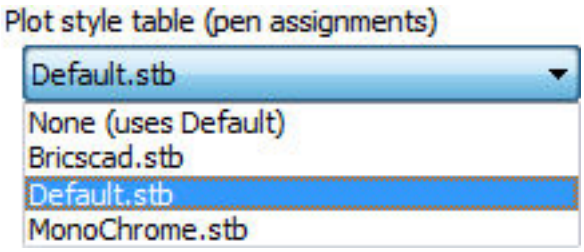


Figure 6.1757

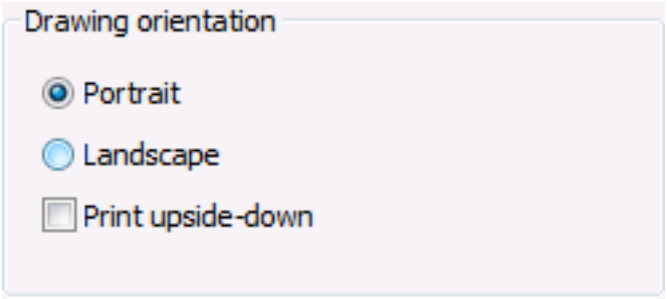


Figure 6.1758

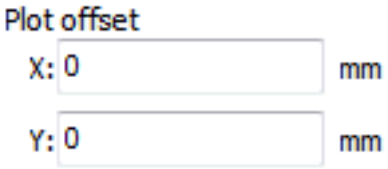


Figure 6.1759

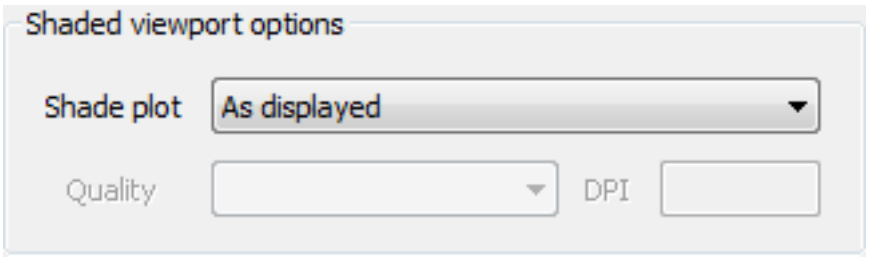


Figure 6.1760



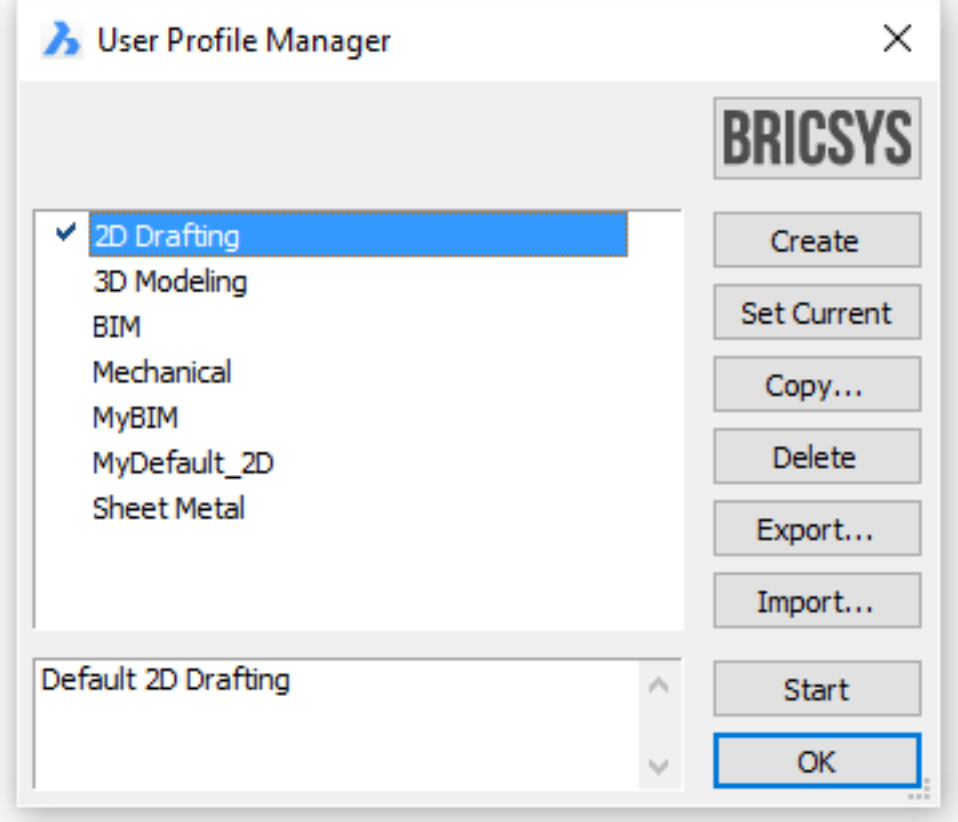
6.24.47 ProfileManager

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the current user profile; also creates, copies, deletes, imports, and exports profiles through a dialog box.

Accessing the Command

command bar: **profilemanager** transparent:'**profilemanager** menu bar: **Tools | User Profile Manager** : **ProfileManager** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/userProfi

Figure 6.1761



Command Options

Option	Description
Create	Creates a new fresh profile, using built-in default settings; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C 1. **Profile name**- type a name for the new profile 2. **Description**- (optional) type a description for the new profile 3. Click **OK** to close and return to the previous dialog box
Set Current	Select a profile in the list, then click the **Set Current** button to set a profile current.The next time you start up the program, it uses this profile.The current profile is marked in the list with a check mark.

Copy	Creates a new profile as a copy of an existing profile: 1. Choose a profile name 2. Click **Copy**; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so 3. **Profile name**- type a name for the copied profile 4. **Description**- (optional) type a description for the copied profile 5. Click **OK** to close and return to the previous dialog box
Delete	Deletes profiles: 1. Select a profile in the list 2. Click**Delete**to delete the profile
Export	Exports profiles: 1. Select a profile in the list 2. Click**Export** button 3. Specify the export folder and ARG file name 4. Click **Save** Exporting, then importing, a profile allows you to migrate user preferences from one computer to another.
Import	Imports saved profiles: 1. Click **Import** button 2. In the Import Profile dialog box, select an ARGfile 3. Click **Open**
Start	Starts another BricsCAD session using the selected profile: • Double click the profile in the list • Select the profile, and then click the **Start** button. If the SingletonMode user preference is set to Yes it is not possible to have multiple BricsCAD sessions open simultaneously.
OK	Closes the User Profile Manager dialog box.

Procedures

- Creating user profiles (1.13.9)
- Starting BricsCAD with a specific user profile (1.13.9)
- Restarting BricsCAD using a different user profile (1.13.9)
- Managing user profiles (1.13.9)
- To export a user profile (1.13.9)
- To import a user profile (1.13.9)
- To import a user profile from another user (1.13.9)

- Using the Welcome dialog box (1.14.1)

Related Commands

Customize (6.9.58) - customizes the user interface of BricsCAD: menus, toolbars, keyboard shortcuts, aliases, and shell commands.

Figure 6.1762

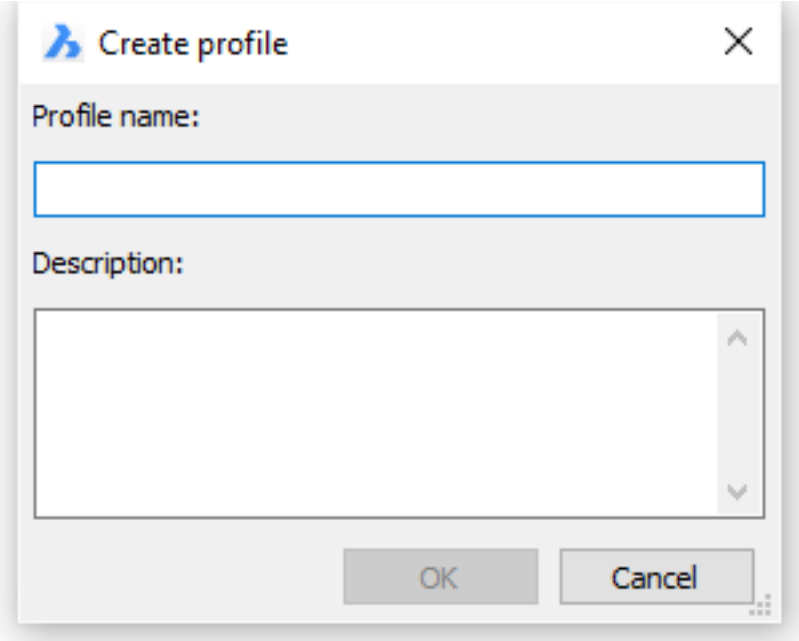


Figure 6.1763

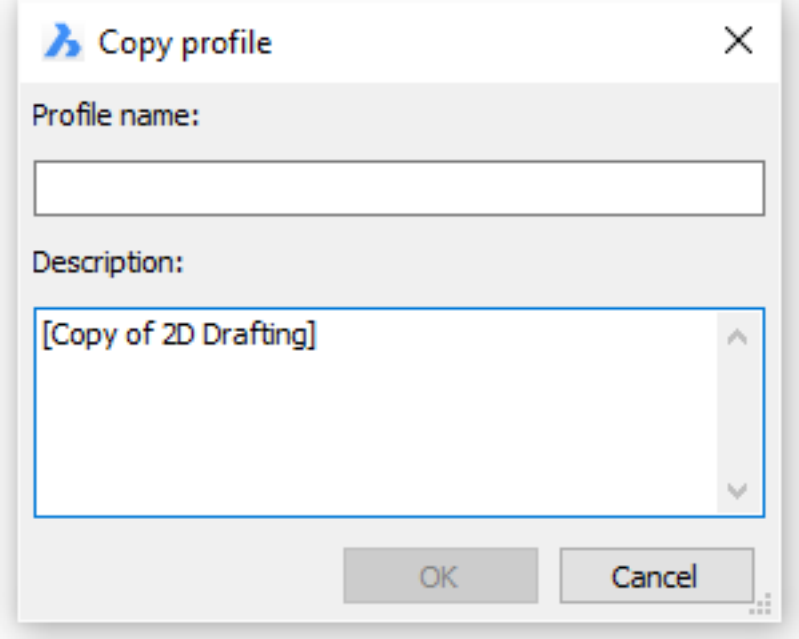
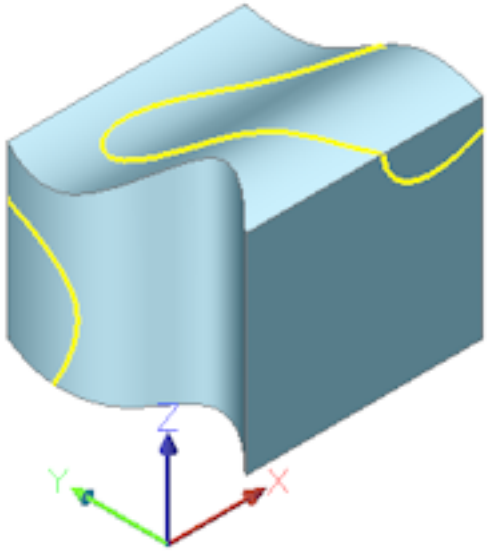


Figure 6.1764



Figure 6.1765



6.24.48 ProjectGeometry

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Projects 2D geometry onto regions, surfaces, or 3D solids to create additional edges (line work). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ProjectGeometryResult.png})

Accessing the Command


command bar: **projectgeometry** menu bar: **Draw | Project Geometry** toolbar: **Draw |**  : **ProjectGeometry**
Prompts you in the command bar: Select 2d entities to be projected or specify [PROjection direction]: **(Select a 2D entity, #1 in the figure below, or choose an option)** Entities in set: 1 Select 2d entities to be projected or specify [PROjection direction]: **(Select more 2D entities or press Enter)** Select a region, surface or 3d solid to project onto: (**Select a region, surface, or 3D solid, #2 in the figure)** 1 object(s) projected successfully. The program draws entities where the 2D entity projects onto the 3D solid, surface, or region and placed on the current layer. (1) 2D entity; (2) 3D solid; blue line = projection direction; pink tube = projection; yellow curves = new edges (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ProjectGeometryStart.png}) (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/ProjectGeometryVolume.png}) (\vref {fig:C:/Dev/T
2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/ProjectGeometryResult.png})

Figure 6.1766

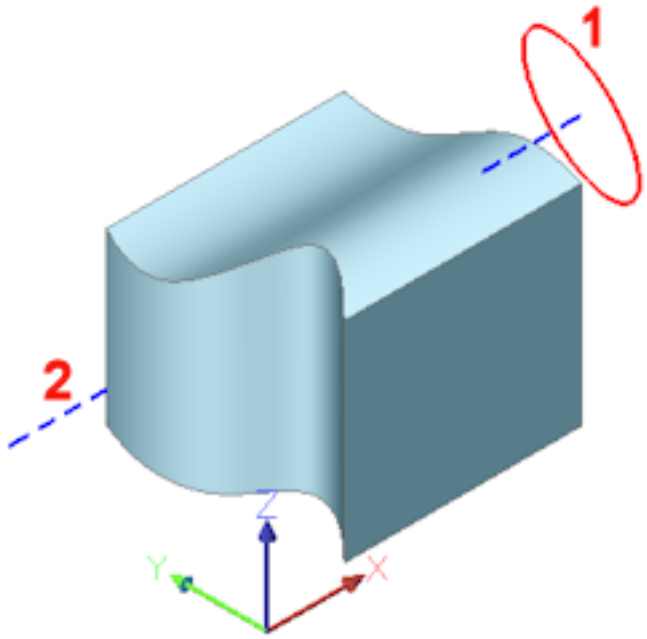


Figure 6.1767

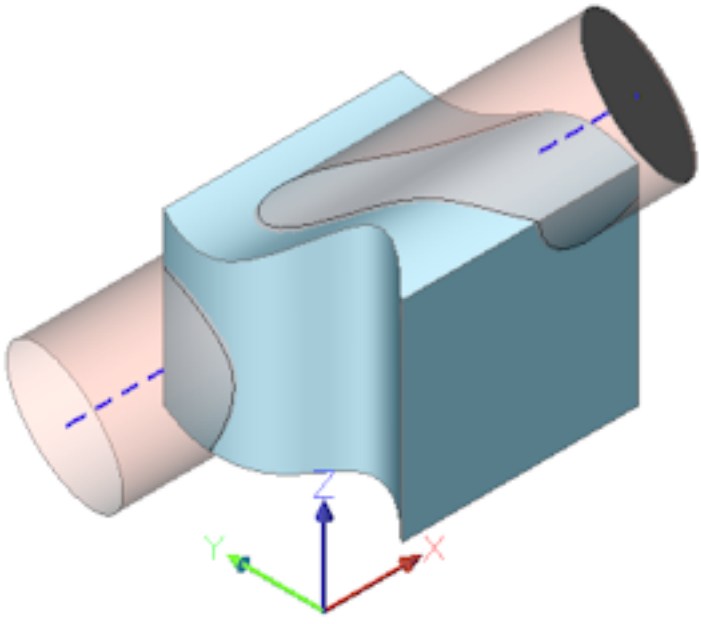
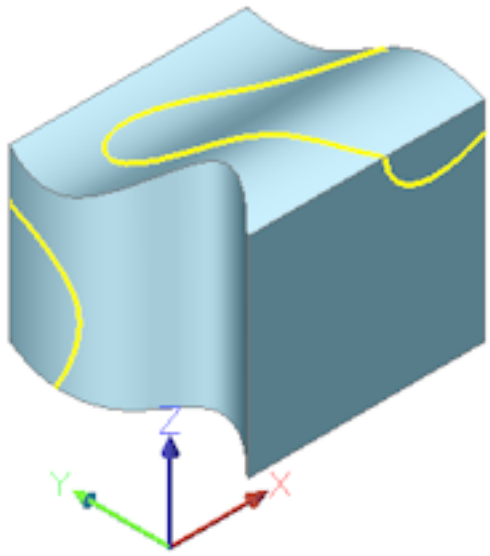


Figure 6.1768



Command Options

Option	Description
Select 2d entities to be projected	Select one or more entities; the entity can be any open or cloesd 2D entities, such as line, circle, ellipse, and open or closed spline.
Select a region, surface or 3d solid to project onto	Select a region, surface or 3D solid onto which to project the 2D entities.
Projection direction	Defines the projection direction.By default 2D geometry is projected orthogonally onto the receiving entity.Prompts you: **Specify the direction [View/Ucs/Points] <View>** - **(enter an option):** • **View** - sets the projection direction perpendicular to the current view orientation. • **UCS**- uses the Z-axis of the current UCS. • **Points**- prompts you to specify the start and end point of the projection direction

Procedures

- Use ”PROJECTGEOMETRY (1.6.30)” to markout your models.
- Using Project Geometry (1.6.29) to create a corrugate Iron roof profile.

Related Commands

Imprint (6.17.12) - imprints 2D entities onto planar faces of 3D solids and surfaces for additional edges on planar faces. XEdges (6.34.3) - creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, surfaces, etc. and regions.

Figure 6.1769



6.24.49 Properties

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the Properties panel with either the default properties or the properties of selected entities.

Accessing the Command


command bar: **properties** aliases: **ch pr props mo ddchprop ddmodify** keystrokes:**Ctrl+1 Ctrl+Shift+P** menu bar:**View | Properties Modify | Properties** toolbar: **Standard** |  : **properties** Displays a palette with varying content:

- When no entities are selected, displays the following content: **General** - reports the current entity property settings **View** - reports the current view settings and visual style **Misc** - reports the current annotation scale and default lighting status

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/properties-1-panel.png})

- When a single entity is selected, displays the following content: **General** - reports the properties of the entity **3D Visualization** - reports the entity's material **Geometry** - reports the geometry of the entity

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/properties-2-panel.png})

- When two or more entities are selected, reports the shared properties of the selected entities; where properties are different, the panel reports *varies*.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/properties-3-panel.png})

Figure 6.1770

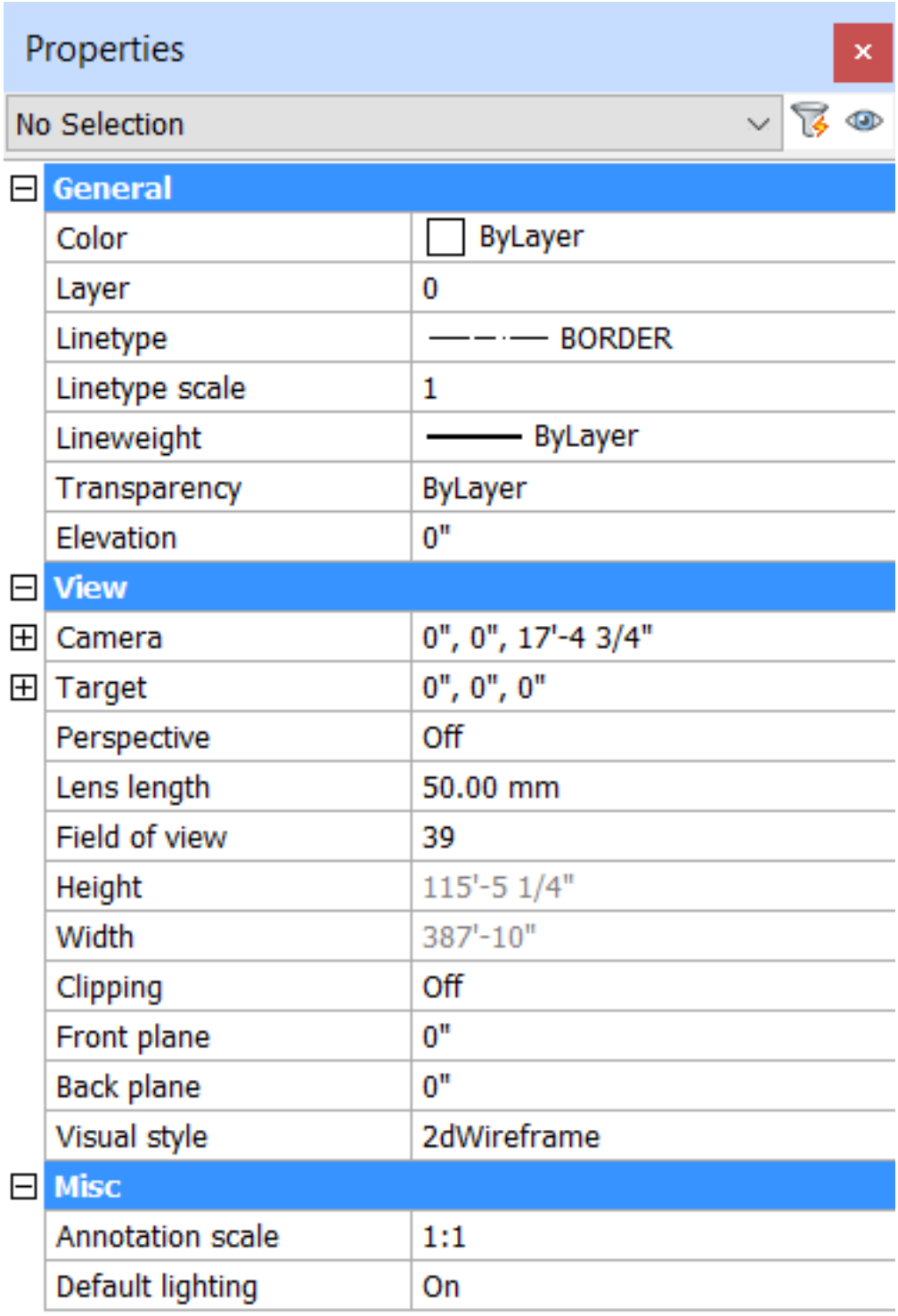


Figure 6.1771

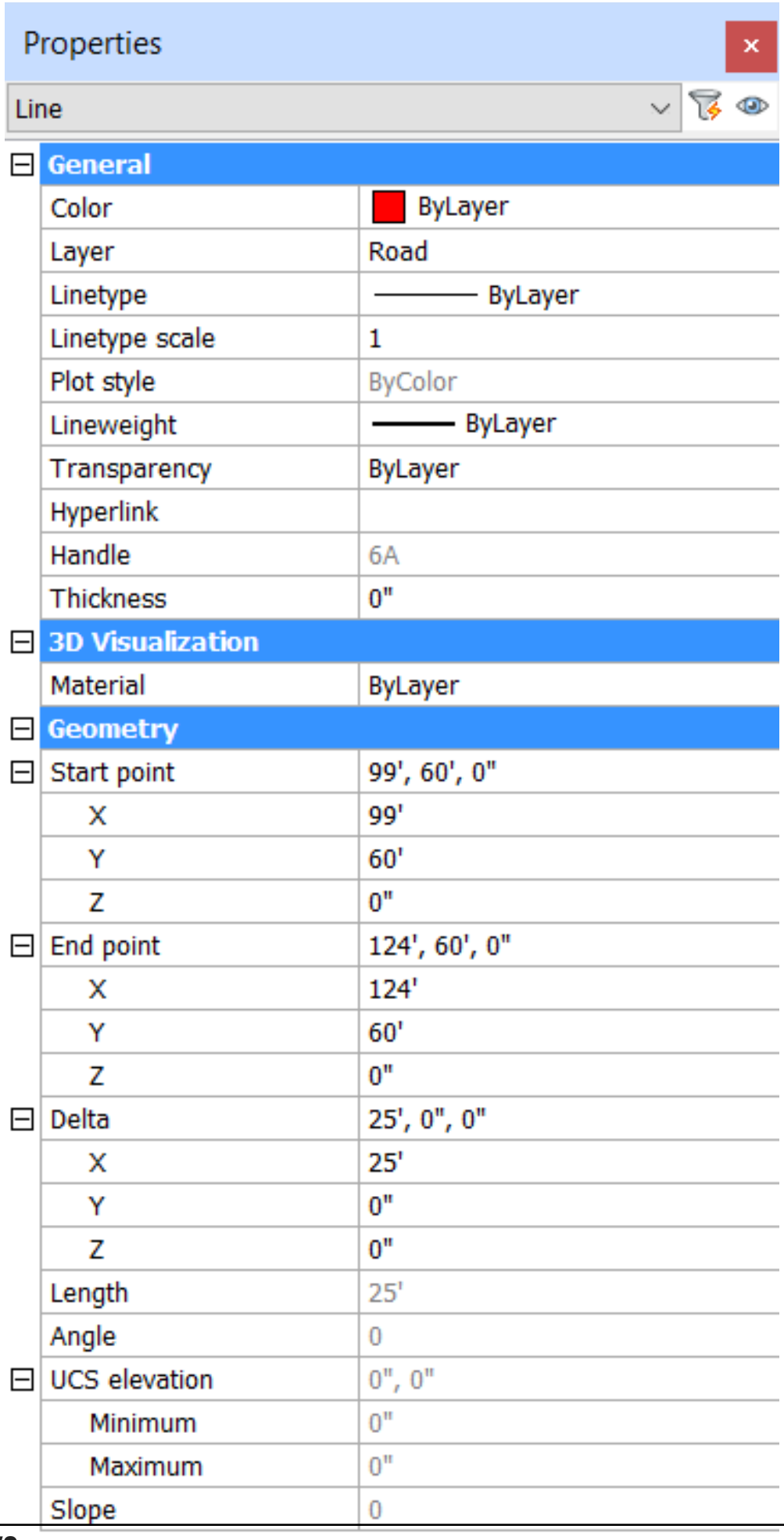
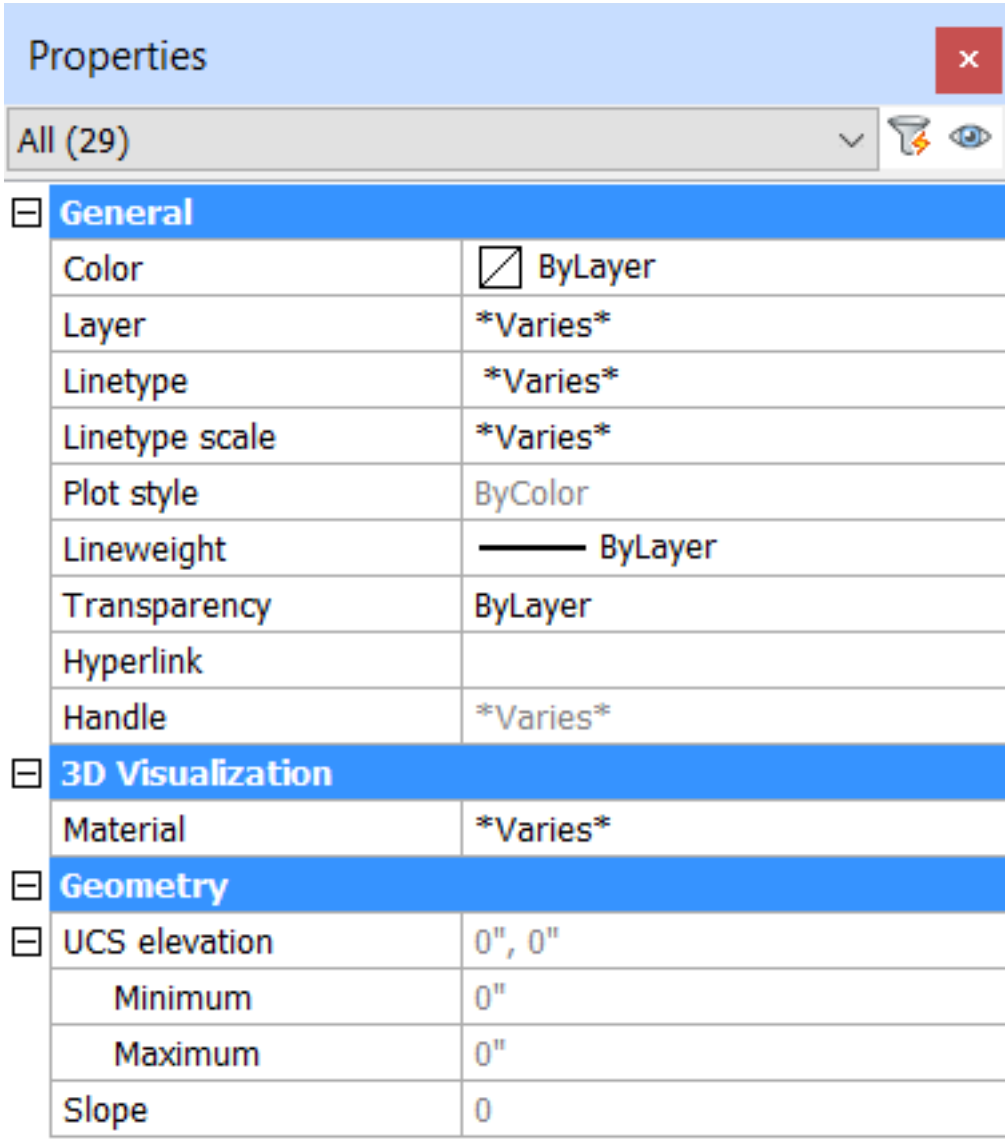


Figure 6.1772



Command Options

Options	Description
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Properties1.stButtonpng})	Reports the number of selected entities.Click to see the number of each entity type in the current selection set: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P Select an entity type in the list to see the shared properties of all entities of this type in the current selection set.
Quick Select	Switches between **Properties** and **Quick Select** mode (See also the Qselect (6.25.5) command).
Property Preview	When on, the Properties panel previews values when you hover the cursor over a property.
General	
Color	Reports the current color, and allows you to change the color of selected entities: • Click the droplist, and then choose another color. • Choose **Select Color** to access the Colors dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i See theColor (6.9.33) command.
Layer	Reports the current layer name. To change the layer of selected entities: 1. Click the droplist, and then choose another name. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i See theLayer (6.20.2) command.
Linetype	Reports the current linetype. To change the linetype of selected entities: 1. Click the droplist, and then choose another name. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so 2. Or, choose **Load** to access the Load Linetypes dialog box. See theLinetype (6.20.33) command.
Linetype Scale	Reports the current linetype scale factor. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i To change the linetype scale, enter a new value. See LTSCALE system variable,
Plot Style	Reports the current plot style. To change the plot style of selected entities, click the droplist and then choose another name. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i This property is only availablewhen plot styles are enabled for the drawing. See PlotStyle (6.24.29) command.

Lineweight	Reports the current lineweight. To change the lineweight of selected entities, click the droplist and then choose another width. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i See LWeight (6.20.41) command.
Transparency	Reports the current level of transparency. To changetransparency, click the droplist, and then choose a settings: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pro transparency-droplist.png}) • Select ByLayer or ByBlock • Select a preset value, such as 10 or 90 • Enter a value between 0 (opaque) and 90 (nearly transparent)
Hyperlink	Reports the current hyperlink. To change the hyperlink of selected entities: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • Or else click the  button to access the Edit Hyperlinks dialog box. See Hyperlink (6.16.12) command.
Thickness	Reports the current entity thickness. To change the thickness of selected entities, enter a new value. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i See Elev (6.13.4) command.
Handle	Reports the hexadecimal value by which the entity is known in the DWG database; read-only, and so you cannot change it.
3D Visualization	
Material	Reports the current material definition of the selected entity.To change, click the field, then select a material from the drop-down list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pro material-droplist.png}) See the Materials (6.21.11) command to manage material definitions.
View	
Camera	Reports and edits the current position of the camera's position.See the Camera (6.9.2) command.
Target	Reports and edits the current position of the camera's target position.See the Target command.
Perspective	Reports the current value of the PERSPECTIVE system variable; switchesperspective view mode on and off.
Lens Length	Reports the current value of the LENSLENGTH system variable; edits the value of the LENSLENGTH system variable.
Field of View	Reports the current view angle; editsthe camera view angle. Editing this value changes the value of the LENSLENGTH system variable.
Height, Width	Reports the dimensions of the current view in drawing units; cannot be edited. These values are changed when the Field of View property is modified.
Clipping	Toggles view clipping on and off; when clipping is on, parts of the view before the front clipping plane and behind the back clipping plane become invisible.
Front, Back Plane	Specifies the locations of the front and back clipping planes; available only when the Clipping property is on.

Visual Style	Reports the current visual style, and selects a different one. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pro visuastyle-droplist.png}) See the VisualStyles (6.31.13) command to edit and create visual styles.
Misc (Miscellaneous)	
Annotation Scale	Reports the current value of the CANNOSCALE system variable, and allows youto choose another one from the list defined in the drawing are available (see the ScaleListEdit (6.27.7) and -ScaleListEdit (6.27.8) commands).
Default Lighting	Toggles default lighting status on and off: • **On** - the program uses default lighting, which substitutes when no lights are defined in the scene • **Off** - the program uses lights defined in the scene

Other properties may appear depending on the entity selected.

Procedures

- To open or close a panel (1.12.7)
- To dock a panel (1.12.7)
- To dock a panel with respect to another docked panel (1.12.7)
- To stack panels in a tabbed panel (1.12.7)

Related Commands

Area (6.4.13) - reports the area, circumference, and perimeter of entities. AttEdit (6.4.36) - edits attribute values and properties. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. Dist (6.10.31) - reports the 2D and 3D distances between two points. EAttEdit (6.13.1) - edits attribute values and most properties. Id (6.17.1) - reports the x,y,z coordinates of picked points. Layer (6.20.2)- assigns properties to layers. List (6.20.35) - lists the area and other parameters of entities in the command bar. MassProp (6.21.4) - reports the properties of solid models and regions. PropertiesClose (6.24.50) - closes the Properties panel. -ToolPanel (6.29.21) - controls the display of tool panels, such as the Properties panel, Ribbon and the Sheet Sets panel from the command line.

Figure 6.1773



Figure 6.1774

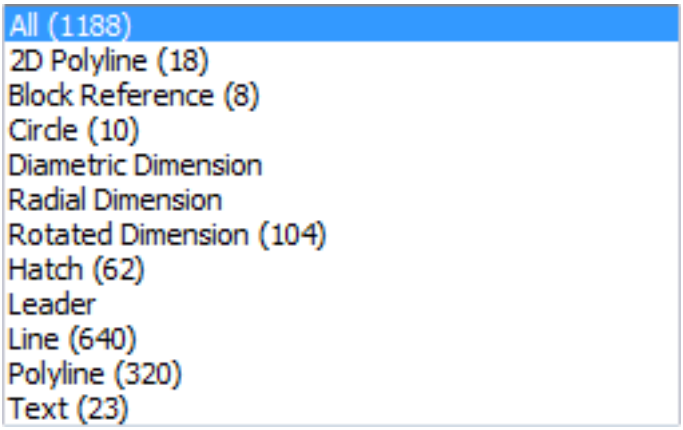


Figure 6.1775

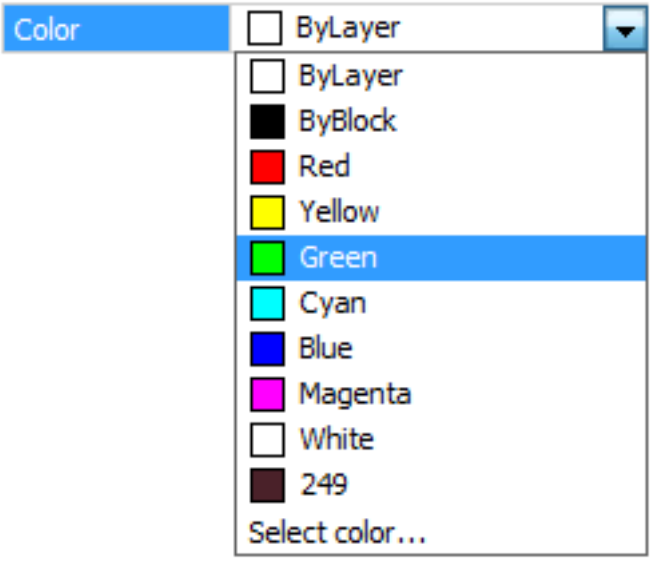


Figure 6.1776

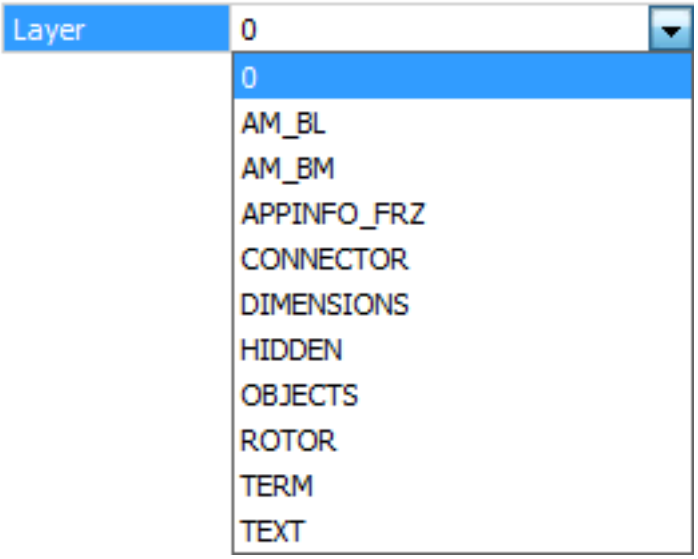


Figure 6.1777

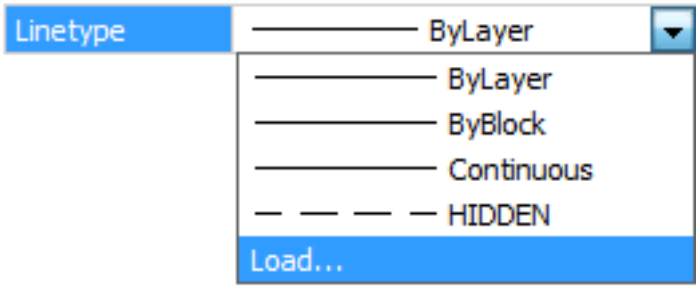


Figure 6.1778



Figure 6.1779

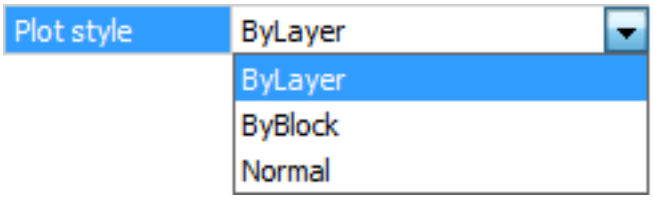


Figure 6.1780

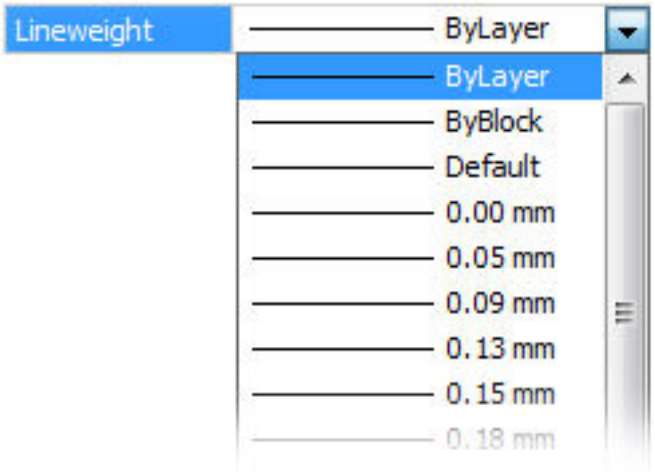


Figure 6.1781

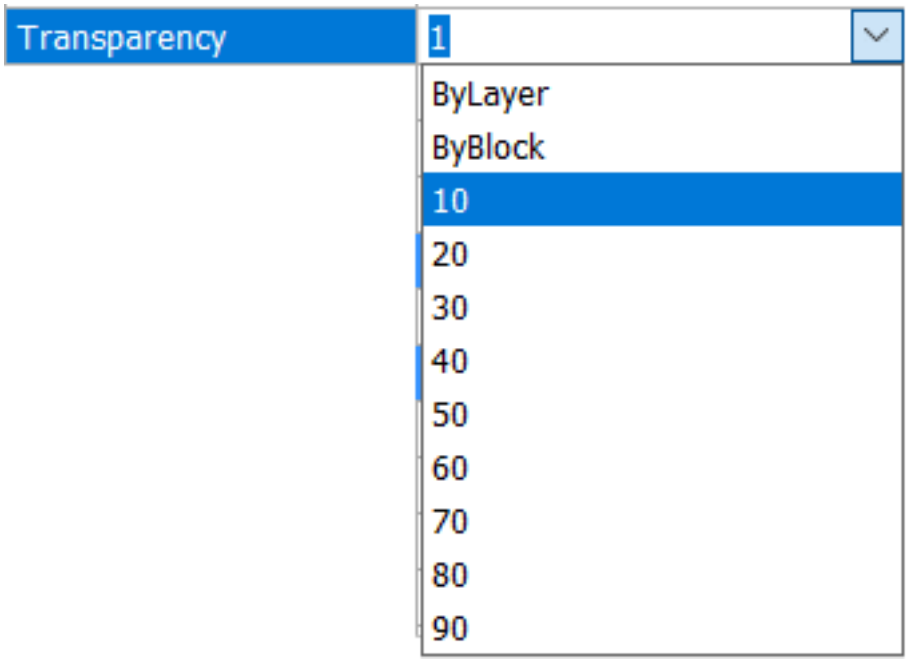


Figure 6.1782



Figure 6.1783



Figure 6.1784

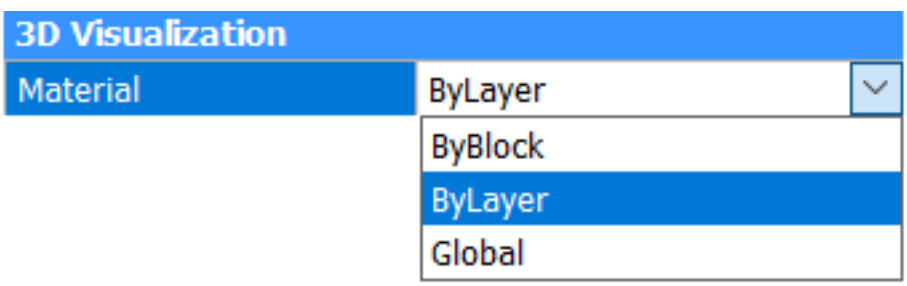


Figure 6.1785

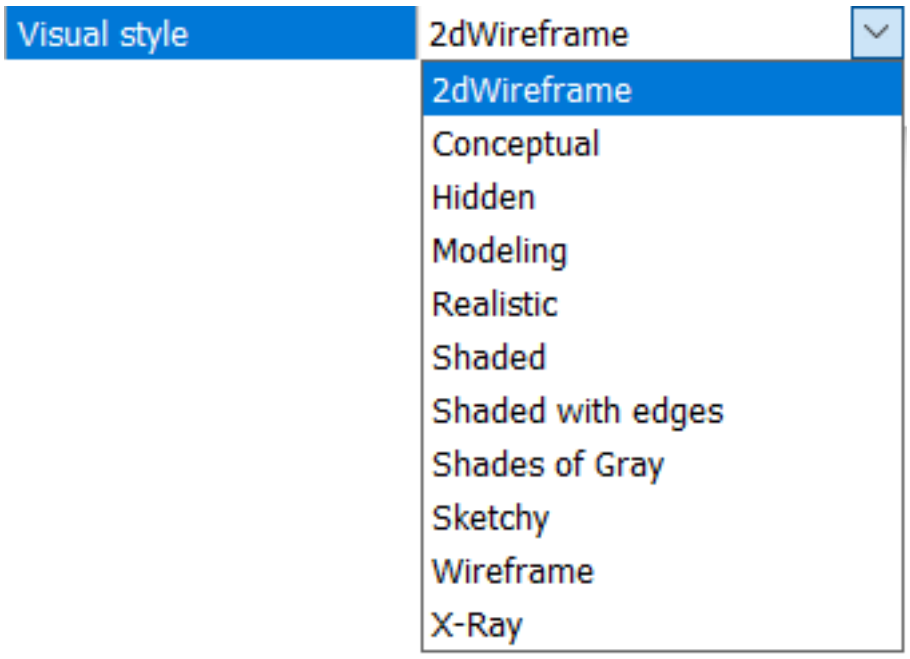


Figure 6.1786



6.24.50 PropertiesClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes the Properties panel.

Accessing the Command

command bar: **propertiesclose** alias: **prc** keystrokes:**Ctrl+1 Ctrl+Shift+P** menu:**View | Properties** : **propertiesclose** Closes the Properties panel. To open the Properties palette, use the Properties (6.24.49) command.

Command Options

No options.

Related Command

Properties (6.24.49) - opens the Properties panel. -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Figure 6.1787



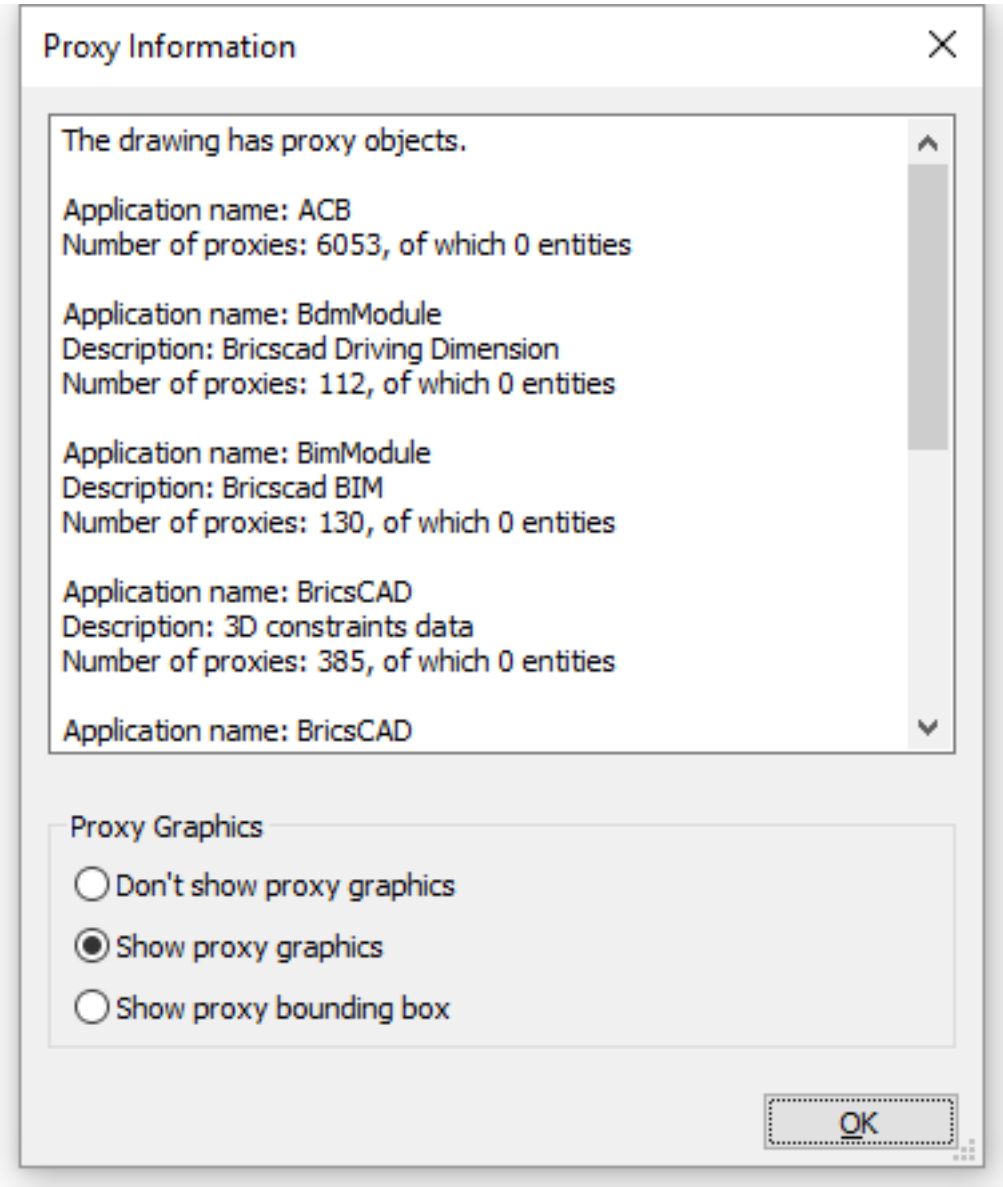
6.24.51 ProxyInfo

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays information about proxy entities and toggles their display in the current drawing. **Note:** Proxy graphics are CAD entities that are unknown to BricsCAD and so are shown in the drawing as geometry that can be edited only in the most basic manner.

Accessing the Command

command bar: **proxyinfo** : **proxyinfo** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

Figure 6.1788



Command Options

Option	Description
Proxy Graphics	Toggles the display of proxy graphics in the drawing: • **Don't show proxy graphics** - turns off the display of proxy graphics • **Show proxy graphics** - shows proxy graphics • **Show proxy bounding box** - displays a rectangle or cube, instead of the proxy

Related Commands

Audit (6.4.32) - repairs drawings that are open. Recover (6.26.3) - recovers damaged drawings.

Figure 6.1789



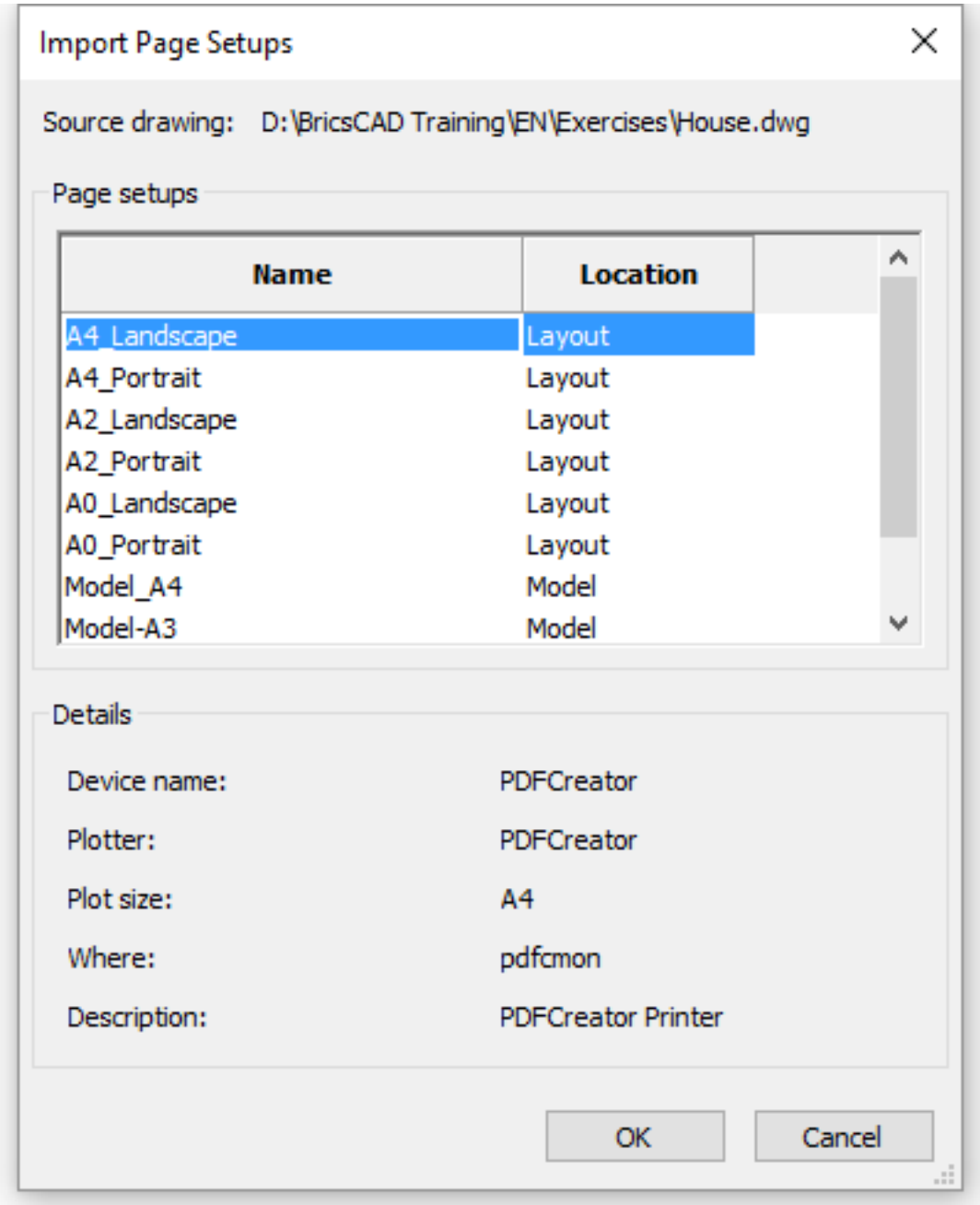
6.24.52 PSetupIn

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Imports page setup definitions from other drawings through a dialog box. **Note:** Page setups define how a drawing is printed, and are used by the Print (6.24.46) and Publish (6.24.55) commands.

Accessing the Command

command bar: **psetupin** : **psetupin** Displays a file dialog: select the drawing from which you want to import page setups, and then click **Open**. Displays a dialog: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/ Select one or more page setups in the list. Hold down the **Ctrl** key to select multiple page setups, then click**OK**to import the selection into the current drawing.

Figure 6.1790



Command Options

Option	Description
Name	Specifies the names of page setups in the drawing; click the Name header to short the names alphabetically. When the drawing contains no page setups, then the dialog box is blank.
Location	Specifies the layout and model locations of page setups; click the Location header to short them alphabetically
OK	Imports the page setups into the drawing
Cancel	Cancels the command and does not import anything

Related Commands

-PsetupIn (6.24.53) - imports page setup definitions from another drawing through the command bar. PageSetup (6.24.1) - creates and edits page setups for plotting drawings in the Drawing Explorer. Layout (6.20.14) - creates, copies, renames, and deletes layouts.

Figure 6.1791



6.24.53 -PSetupIn

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Imports page setup definitions from another drawing through the command bar. **Note:** Page setups define how a drawing is printed, and are used by the Print (6.24.46) and Publish (6.24.55) commands.

Accessing the Command

command bar: **-psetupin** : **-psetupin** Displays a file dialog: select the drawing from which you want to import page setups, and then click **Open**. **Prompts you in the command bar:** Enter user defined page setup(s) to import or [?]: **(Type the name(s) of the page setup(s) you want to import or type ? to list the available page setups)**

Command Options

Option	Description
Enter user defined page setup(s) to import	Type the name(s) of the page setup(s) you want to import; separate setup names by commas
?	Lists the names of page setups in the drawing

Related Commands

PsetupIn (6.24.52) - Imports page setup definitions from another drawing through a dialog box. PageSetup (6.24.1) - creates and edits page setups for plotting drawings in the Drawing Explorer.

Figure 6.1792



6.24.54 PSpace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Switches from model to paper space in a layout tab (short for "paper space"). **Note:** This command operates only in a layout tab.

Accessing the Command

command bar: **pspace** shortcut: in a layout tab, double-click outside a viewport border alias: **ps** status bar: **M:Layout** : **pspace** Displays no prompts in the command bar.

Command Options

No options.

Status Bar Menu

Right-click P:Layout on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image794.png})

Shortcut Menu	Description
Paper Space	Switches to paper space in the current layout.
Model Space	Switches to model space in the current layout.
Layout1	Switches the drawing editor to Layout1.
Model	Switches the drawing editor to Model.
Layout2	Switches the drawing editor to Layout2. When the drawing contains other layouts, they are also listed in the shortcut menu.

Figure 6.1793

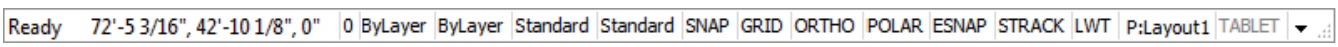
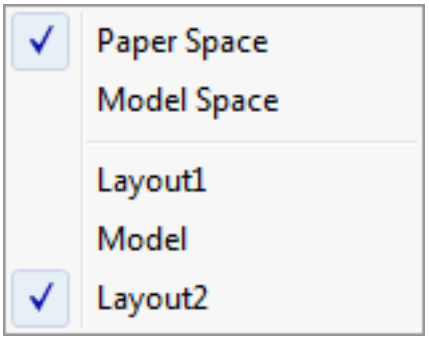


Figure 6.1794



Related Commands

ChSpace (6.9.19) - Moves entities from paper space to model space and vice versa. MSpace (6.21.34) - switches to model space. MView (6.21.37) - creates viewports in paper space. VpClip (6.31.17) - clips viewports. VPorts (6.31.20) - creates viewports in model space. Zoom (6.35.7) - utilizes the XP option to specify the zoom factor in paper space relative to model space.

Figure 6.1795



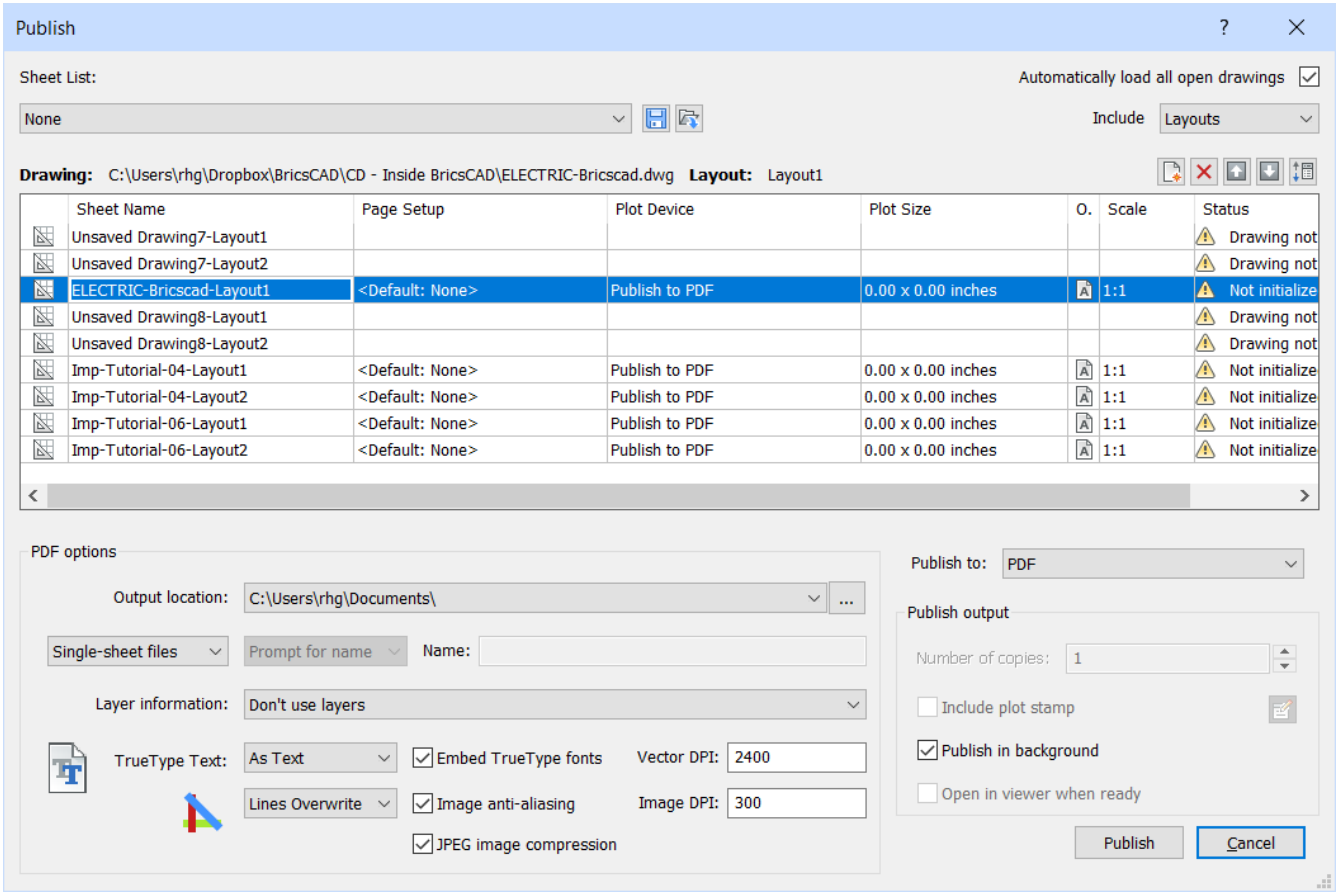
6.24.55 Publish

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sends one or more drawings. layouts, and sheets to the printer, or exports them in PDF format; useful for batch printing of ”books” of drawings. **Note:** To print a batch of drawings when BricsCAD starts, use the **/pl** switch in the program’s properties.

Accessing the Command


command bar: **publish** menu bar: **File | Publish** ribbon:**Home | Print/Plot | Publish** toolbar: **Standard |**  : **publish** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/publish-dlg.png})

Figure 6.1796



Command Options

Option	Description
Load Sheet List	Loads a saved sheet list through a file dialog box. (If the current list is not saved yet, you are prompted to save it.) 1. Select a *.dsd (Drawing Set Description) file. 2. Click**Open**button. 3. Choose whether to **Replace** all sheets in the list or to **Append**sheets to the list. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L
Save Sheet List	Saves the current sheet list to a a *.dsd (Drawing Set Description) file.
Automatically load all open drawings.	Determines how open drawings are handled: • **On** (checked) - adds all model and/or paper space layouts to the list of sheets to be plotted • **Off** (unchecked) - adds the model and/or paper space layouts of the current drawing only This option also sets the value of the PUBLISHALLSHEETS system variable.

Include	Determines which sheets must be included: • **Model** - include only model tabs of open drawings • **Layouts** -include only layout tabs of open drawings • **Model & Layouts** -include both model and layout tabs of open drawings
Add Sheets	Adds sheets to the current list; displays a filedialog box: 1. Select a drawing. 2. All layouts in the selected drawing are added to the sheet list. 3. If a sheet name already exists, a dialog box displays prompting you to enter a new name. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/D
Remove Sheets	Removes the selected sheet(s) from the current list.
Arrange Sheets	Moves the selected sheets up or down, or reverses the sheet order (useful for printers and plotters that print in reverse order).
Drawing	Reports the path and name of the drawing of the currently selected sheet; ready-only.
Layout	Reports the name of layout of the currently selected sheet; read-only.
Sheet List	
Sheet Name	Lists the names of layouts in the sheet set.
Page Setup	Specifies thePage Setup (6.24.1) name of each sheet: • Click the down arrow on the droplist to select a different page setup: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P • Or else choose **Import...** to import page setups of another drawing: 1. In the file dialog box, select a drawing. 2. Click the **Open** button. 3. Choose a page setup from thelist. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P
Plot Device	Selects the name of the plot device, as defined in the Page Setup (6.24.1) of the sheet.
Plot Size	Specifies the size of the paper in the currently selected page setup.
Orientation	Selects the paper orientation in the currently selected page setup: Landscape ( ) or Portrait ( ).
Scale	Specifies the scale of the currently selected page setup.
Status	Reports the status of the currently selected page setup, including errors detected.

Publish to:

Choose the output target of the Publish command:

- Plotter named in page setup . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so
- Number of copies: sets the number of copies for each sheet to be printed.
- Include plot stamp: if checked adds a plot stamp. Click the **Edit Plot Stamp** button () to change plot stamp data.
- Turning Drawings into PDFs Youtube (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so

Publish in background	Determines if the sheet is published in the background: • **Yes** - sheets are published in the background, which takes longer to produce the prints, but allows you to continue to work in the program • **No** - sheets are published in the foreground, which is faster, but prevents you from working with the program until the print job is completed The BACKGROUNDPLOT system variable controls whether documents are published in the foreground or background.
Open in viewer when ready	Determines if the sheet set should be viewed; this option is available when **Publish To** is set to ”PDF” and **Publish in Background** is ”Off.”
Publish	Publishes the current sheet set to the selected output target.

Related Commands

ExportPdf (6.13.17) - exports drawings to PDF documents. -Publish (6.24.56) - prints a saved sheet list. PageSetup (6.24.1) - assigns plot style to drawings. Plot (6.24.26) - plots drawings through the command bar. Print (6.25.3) - plots drawings through a dialog box. SheetSet (6.27.31) - manages sheet sets through the Sheet Sets panel.

Figure 6.1797

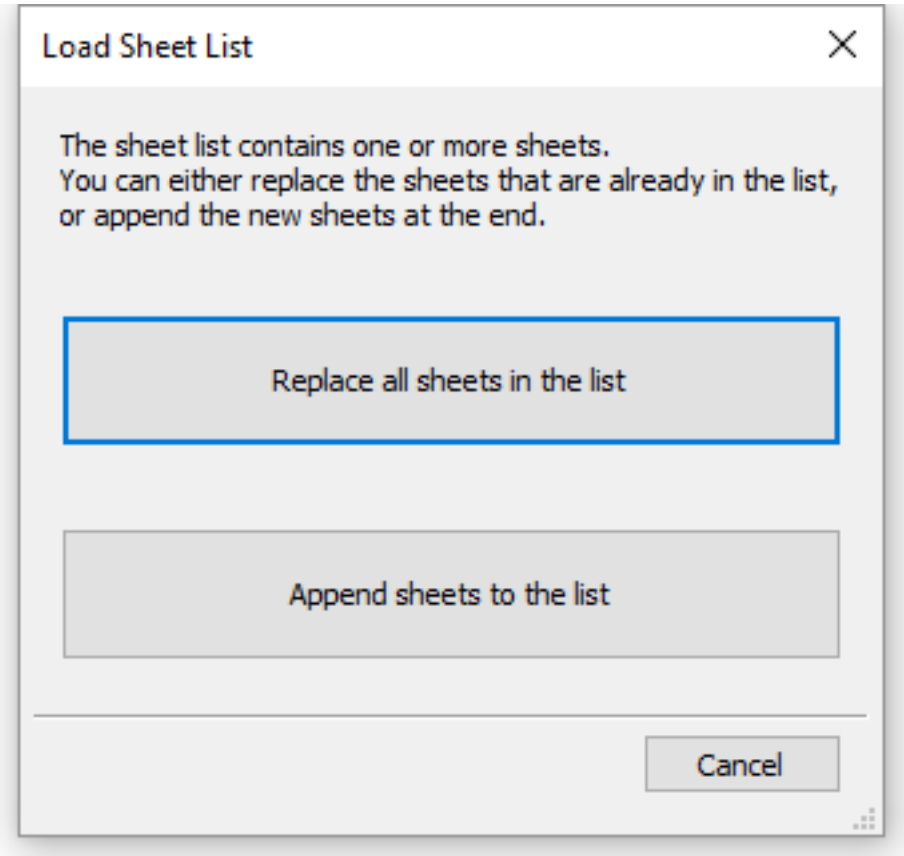


Figure 6.1798

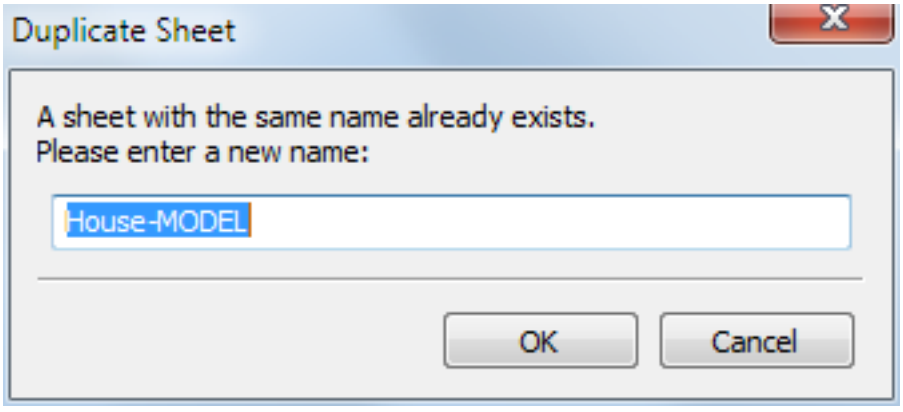


Figure 6.1799

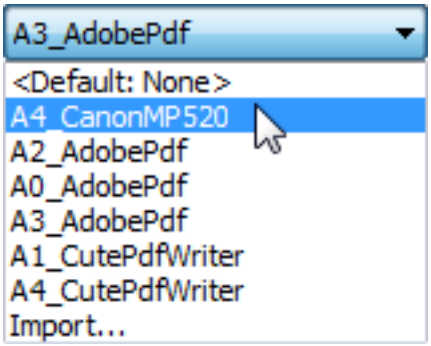


Figure 6.1800

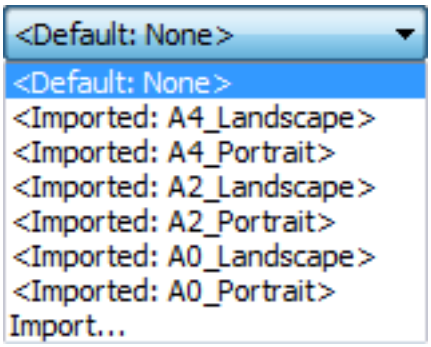


Figure 6.1801



Figure 6.1802

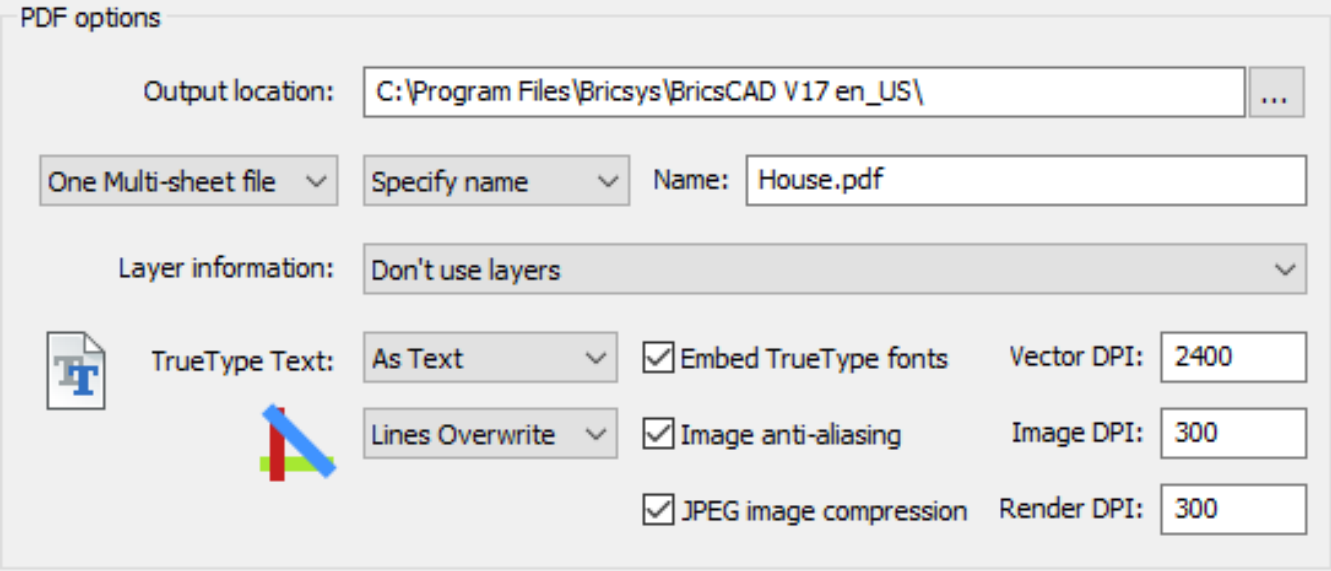


Figure 6.1803



Figure 6.1804

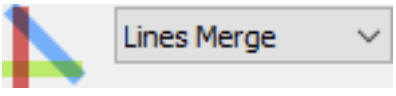


Figure 6.1805



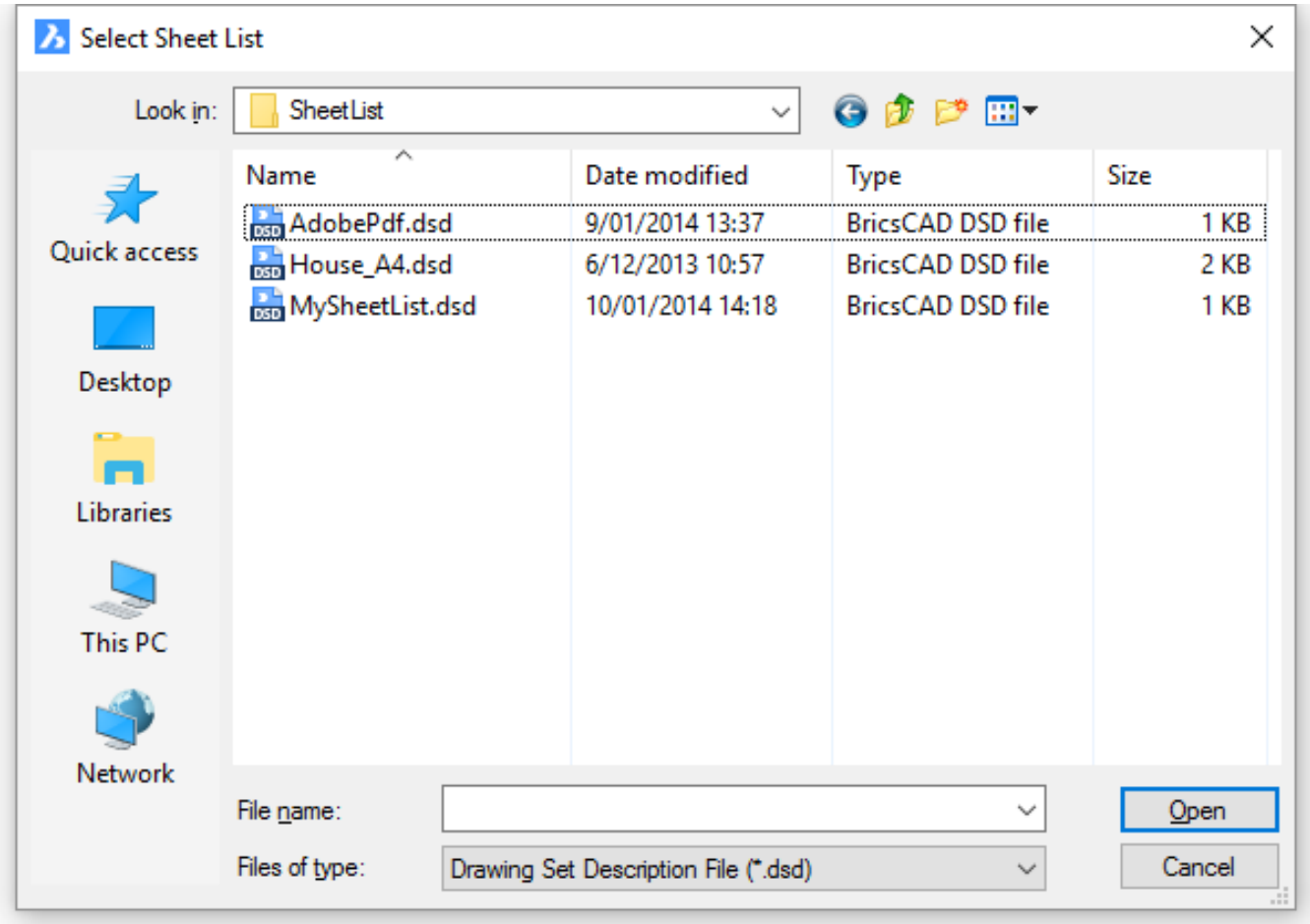
6.24.56 -Publish

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Prints the content of a DSD file, a saved list of drawings, sheets, and layouts, at the command line. **Note:** Create the DSD file with the Publish (6.24.55) command.

Accessing the Command

command bar: **-publish** : **-publish** Displays a File dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
Select a DST file in the list, then click the **Open** button. If the BACKGROUNDPLOT (1.13.11) system variable is 2 or 3, the sheets are printed in the background.

Figure 6.1806



Command Options

No options.

Related Commands

ExportPdf (6.13.17) - Exports layouts in a drawing to a pdf docucment. Publish (6.24.55) - prints a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file. PageSetup (6.24.1)- assigns plot style to drawings. Plot (6.24.26) - plots drawings through the command bar. Print (6.25.3) - plots drawings through a dialog box. SheetSet (6.27.31) - manages sheet sets. Opens the **Sheet Sets** panel.

Figure 6.1807



6.24.57 Purge

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Removes unused named entities from drawings, such as unused layers and linetypes.

Accessing the Command

 command bar: **purge** alias: **pu, -purge** menu bar: **File | Purge** ribbon:**Home | File | Purge** : **purge Prompts you in the command bar:** Purge [BAtch all/purge All/Blocks/DEtail view styles/Dimension styles/Groups/LAyers/LineTypes/MA styles/MUltileader styles/Plot styles/Regapps/SEction view styles/SHapes/Table styles/text STyles/Visual styles/Zero-length geometry/Empty text entities/Orphaned data]: **(Enter an option)**

Command Options

Option	Description
BAtch all	Purges all unused named entities from the drawing without prompts.
purge All	Purges all unused named entities from the drawing with the following prompt: **Enter name(s) to purge <*>: (Enter an entity name, or press Enter to have the program check all named entities) Verify each name to be purged? [Yes/No] <Yes>:** • **Yes** - purges each unused named entity after you confirm • **No** - purges all unused named entities without confirmation
Blocks	Removes unused block definitions from the current drawing; prompts you: **Enter name(s) to purge <*> - (enter the names of block definitions to purge, or press Enter to purge all unused ones.)**
DEtail view styles	Removes used detail view styles used by the ViewBase and related commands: **Enter name(s) to purge <*> - (enter the names of styles to remove, or else press Enter to purge all unused ones.)**
Dimension styles	Removes unused dimension styles from the current drawing; prompts you: **Enter name(s) to purge <*> - (enter the names of dimension styles to purge, or press Enter to purge all unused ones.The "Standard" dimension styles cannot be purged.)**
Groups	Removes unused named groupss from the current drawing; prompts you: **Enter name(s) to purge <*> - (enter the names of groups styles to purge, or press Enter to purge all unused ones.)**
LAyers	Removes unused layers from the current drawing; prompts you: **Enter name(s) to purge <*> - (enter the names of layers to purge, or press Enter to purge all unused ones.Layer "0" cannot be purged.)**

LineTypes	Removes unused linetypes from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names of linetypes to purge, or press Enter to purge all unused ones. Linetype "Continuous" cannot be purged.)**
MAterials	Removes unused materials from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names of materials to purge, or press Enter to purge all unused ones.The "Global" material cannot be purged.)**
MLine styles	Removes unused multiline styles from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names of multiline styles to purge, or press Enter to purge all unused ones.Multiline style "Standard" cannot be purged.)**
MUltileader styles	Removes unused multileader styles from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names of mleader styles to purge, or press Enter to purge all unused ones.MLeader style "Standard" cannot be purged.)**
Plot styles	Removes unused plot styles from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names of plot styles to purge, or press Enter to purge all unused ones.)**
Regapps	Removes unused regapp keys (1) from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names registered application keys to purge, or press Enter to purge all unused ones.)** Regapp (registered application) keys are used to attach additional information to entities in drawings by third-party applications. The keys are saved with the drawing, and unused ones can be purged by this command.
SEction view styles	Removes unused section view styles (used by the ViewBase and related commands) from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names of section view styles to purge, or press Enter to purge all unused ones.)**
Shapes	Removes unused shapes (loaded into the drawing with the Load command) from from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names of shapes to purge, or press Enter to purge all unused ones)**
Table styles	Removes unused table styles from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names of table styles to purge, or press Enter to purge all unused ones. Table style "Standard" cannot be purged.)**
text STyles	Removes unused text styles from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names of styles to purge, or press Enter to purge all unused ones. Text style "Standard" cannot be purged.)**
Visual styles	Removes unused user-defined visual styles from the current drawing; prompts you: **Enter name(s) to purge <*>** - **(enter the names of visual styles to purge, or press Enter to purge all unused ones.)** Only user defined visual styles can be purged; hard-coded visual styles cannot be purged.
Zero-length geometry	Removes geometry with no length, without a prompt.

Empty text entities	Removes empty text entities from the current drawing, such as those from imported DGN design files. No prompt is displayed.
Orphaned data	Performs a drawing scan and removes obsolete DGN linestyle data.No prompt is displayed.

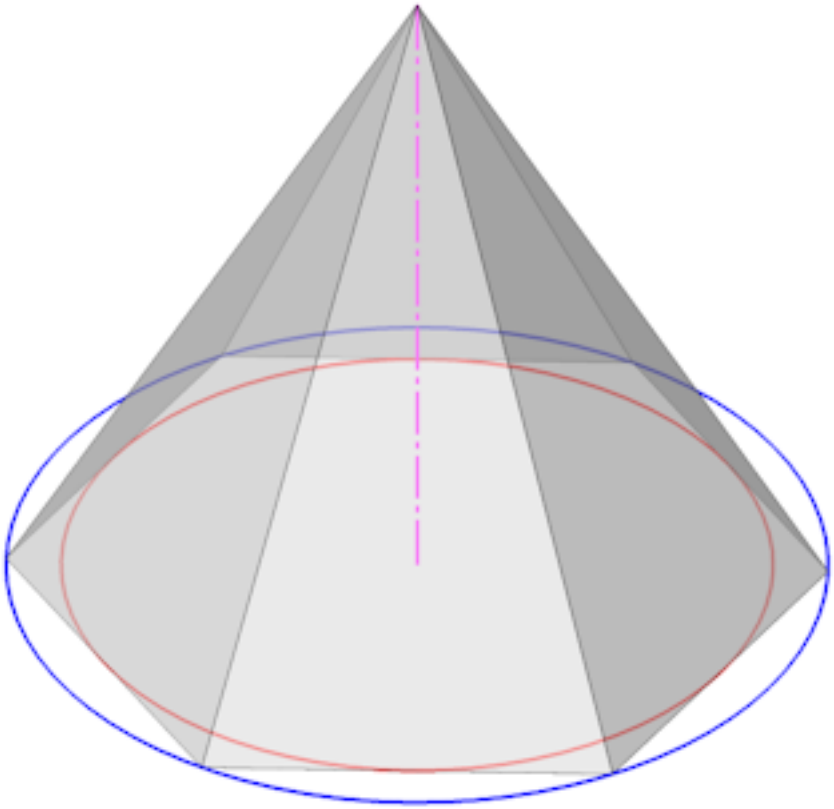
Related Command

Explorer (6.13.14) - purges unused entity names through the Drawing Explorer.

Figure 6.1808



Figure 6.1809



6.24.58 Pyramid

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws 3D pyramids:

- Classic: pyramid is drawn from a 3D polyface mesh.
- Shape, Pro and Platinum: pyramid is drawn as a 3D solid.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Pyramid.png})

Accessing the Command


command bar: **pyramid** alias: **pyr** menu bar: **Model | 3D Solids | Pyramid** ribbon:**Solid | Create | Primitive | Pyramid** toolbar: **3D Solids |** : **pyramid**

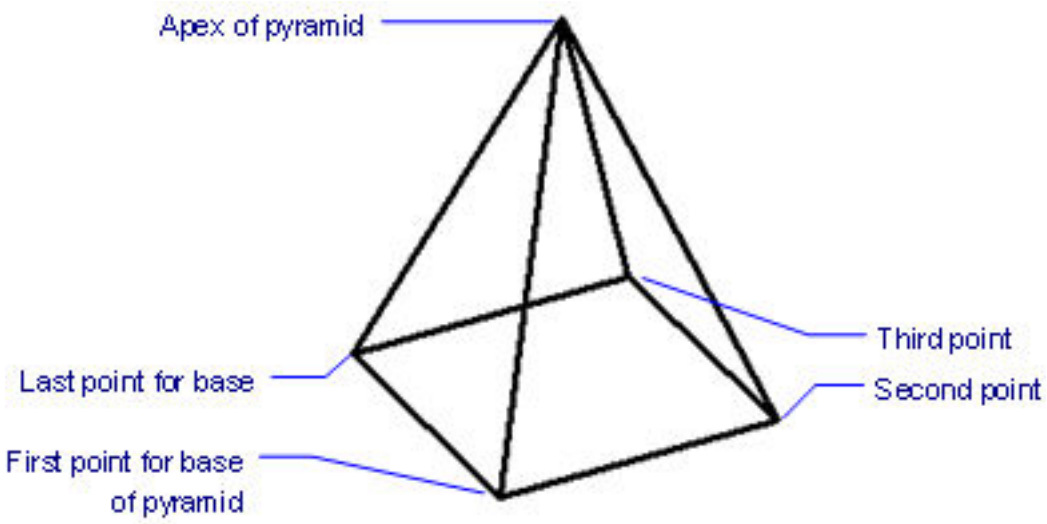
- Command sequence in Pro and Platinum:

You are prompted: Specify center point or [Edge/Sides]: **(Pick a point, enter coordinates, or choose an option)** Specify base radius or [Inscribed]: **(Type a value, pick a point, or choose an option)** Specify height or [2Point/Axis endpoint/Top radius]: **(Type a value, pick a point, or choose an option)**

- Command sequence in Classic:

You are prompted: First point for base of pyramid: **(Pick a point, or enter coordinates)** Second point: **(Pick another point, or enter coordinates)** Third point: **(Pick a third point, or enter coordinates)** Tetrahedron/<Last point for base>: **(Pick the last point, enter coordinates, or type T)** Set apex of pyramid or [Ridge/Top surface]:**(Enter an option, pick a point, or enter coordinates)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sour

Figure 6.1810



Command Options in Pro and Platinum

Pro and Platinum Option	Description
Specify center point	**Specifies the center of the pyramid's base:(pick a point or enter x,y,z coordinates.)**
Sides	Specifies the number of sides (or vertices of the base); prompts you: **Enter number of sides <4>** - **(Type a number, or press Enter to accept the default)**
Edge	Specifies the length of the edges of the base face; prompts you: 1. **Specify first endpoint of edge -(specify a point)** 2. **Specify second point of edge** - **(specify a point)** As you move the cursor for the second point, the program ghosts an image of the pyramid.
Specify base radius	Specifies the radius of a circle inscribed by the base of the pyramid; prompts you: **Specify base radius or [Circumscribed]** - **(enter a value, specify a point, or else enter C to switch to defining a circumscribed base) Inscribed** means that the size of the base is the double the distance from the center point to one of the midpoint of the base: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/py incribed-eg.png})
Circumscribed	Specifies the radius of a circle circumscribed by the base of the pyramid; prompts you: **Specify base radius or [Inscribed]- (enter a value, specify a point, or else enter I to switch to defining an inscribed base)** Circumscribed means that the size of the base is determined by double the distance from the center point to one of the vertices of the base: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/py circumscribed-eg.png})
Specify height	Sets the height by picking a point or entering x,y,z coordinates

2Point	Set the height of the pyramid by specifying two points, such as along another entity; prompts you: 1. **Specify first point** - **(pick a point or enter x,y,z coordinates)** 2. **Specify second point** - **(pick another point or enter x,y,z coordinates)**
Axis endpoint	Sets the length and orientation of the pyramid's axis by specifying the top of the pyramid, creating a rotated pyramid; prompts you: **Specify axis endpoint - (specify a point or enter x,y,z coordinates)**
Top radius	Allows to create a truncated pyramid; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P 1. **Specify top radius <0.0>**- **(type a value or specify a point to define the radius of the circle the top face of the pyramid is inscribed in)** 2. **Specify height or 2Point/Axis endpoint**- **(type a value or specify a point)**

Command Options in Classic

See the Ai_Pyramid (6.5.10) command.

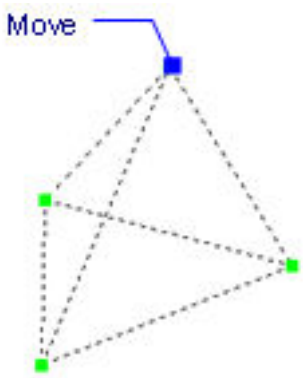
Grips Editing

Pyramids made of solid models can be edited directly through grips:

1. Select the pyramid. Notice that it has several grips.
2. Drag a grip to move the pyramid. (All grips perform the same function: move.)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image439.jpg})

Figure 6.1811



Related Commands

Box (6.6.17) - creates 3D boxes as solid models. Cone (6.9.63) - creates 3D cones as solid models. Cylinder (6.9.60) - creates 3D cylinders as solid models. Dish (6.10.30) - creates the lower half of 3D balls as solid models. Dome (6.10.34) - creates the upper half of 3D balls as solid models. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Pyramid (6.24.58) - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic) Revolve (6.26.30) - revolves 2D entities into 3D solid models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Sphere (6.27.64) - creates 3D balls as solid models. Subtract (6.27.56) - removes one solid model from a second. Torus (6.29.22) - creates 3D donuts as solid models. Union (6.30.6) - joins two or more solid models into a single entity.

Figure 6.1812

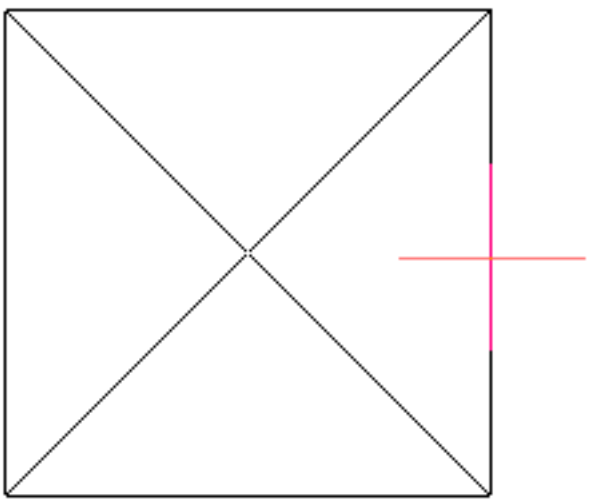


Figure 6.1813

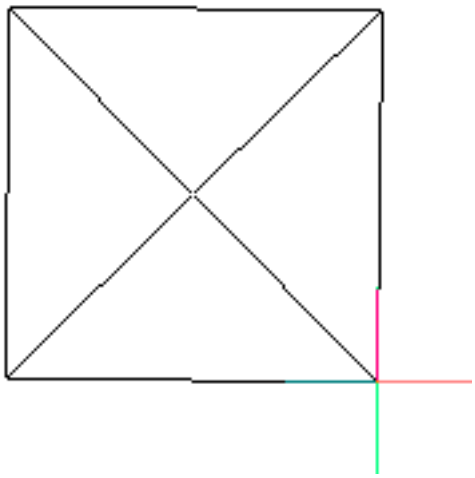


Figure 6.1814

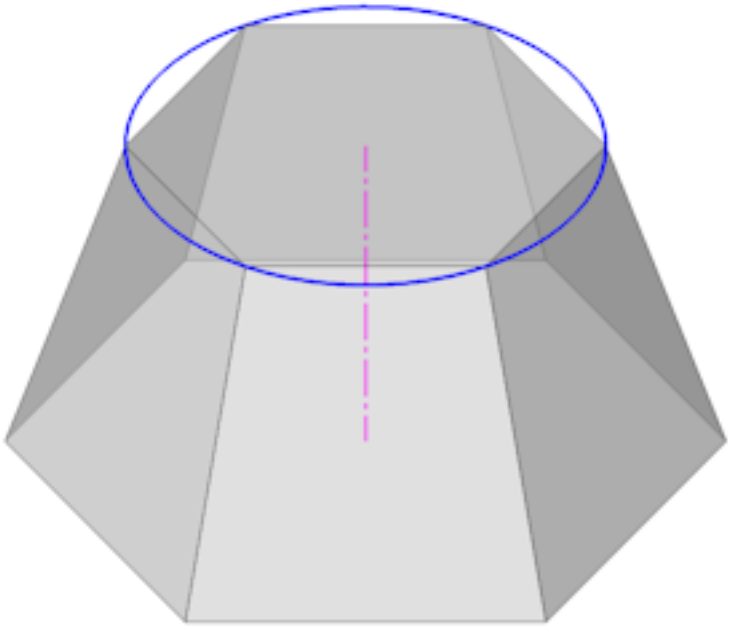


Figure 6.1815



6.24.59 PointCloudPointSize_Plus

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Increases the size of points in point clouds.

Accessing the Command

 Keyboard: **pointcloudpointsize_plus** Toolbar: **Point Cloud :PointCloudpointsize_plus Prompts you:** Increased point size to 1.000000

Command Options

None. The program increases the size of points in cloud by one.

Related Commands

PointCloudAttach (6.24.34) - opens a BPT point cloud file, and then attaches it to the current drawing. PointCloudPointMax (6.24.37) - defines the maximum amount of points shown per point cloud. PointCloudPointSize (6.24.39) - specifies the size of points. PointCloudPointSize_Minus (6.24.40) - decreases the size of points. PointCloudPreprocess (6.24.60) -converts point cloud files to BricsCAD BPT format.

Figure 6.1816



6.24.60 PointCloudPreprocess

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Converts point cloud files to BricsCAD BPT format

Accessing the Command

 Keyboard: **pointcloudpreprocess** Toolbar: **Point Cloud :PointCloudPreprocess** Displays the Point Cloud Data File dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pointcloudpreprocopen.png}) Choose a LAS, PTX, or PTX file, and then click **Open**. The program displays the Give Name for Preprocessing Output File dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pointclsave.png}) Choose a folder in which to place the BPT file, and then click Save. **The program prompts you:** Start preprocessing of raw point cloud data? This is a necessary step, but will take ~1 minute per million points. [Yes/No] <Yes>:**(Press Enter to continue)** Give unit for point coordinates. [Inches/Feet/Miles/miLlimeters/Centimeters/mEters/Kilomet <mEters>: **(Specify a unit type)** Chosen units are Meters Preprocessing data: C:\Users\ralphg\Desktop\Dropbox\dwg\Point Clouds\3D part.bpt -> This will take apprx 1 minute per million points source data. The program processes the points in the background so that you can carry on working on drawings. The processing time can take from several seconds to several minutes. When done, the program displays a dialog box with the results of the conversion: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/P/pointcloudpreprocess-done.png}) When the program reports the conversion is completed, use the PointCloudAttach (6.24.34) command to attach the BPT point cloud file to the current drawing.

Figure 6.1817

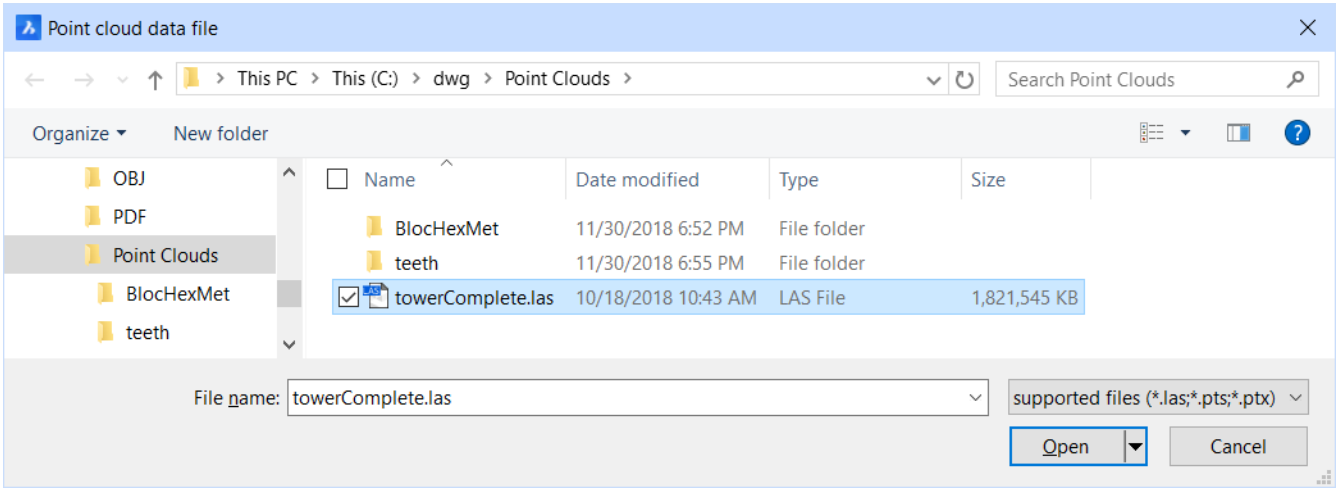


Figure 6.1818

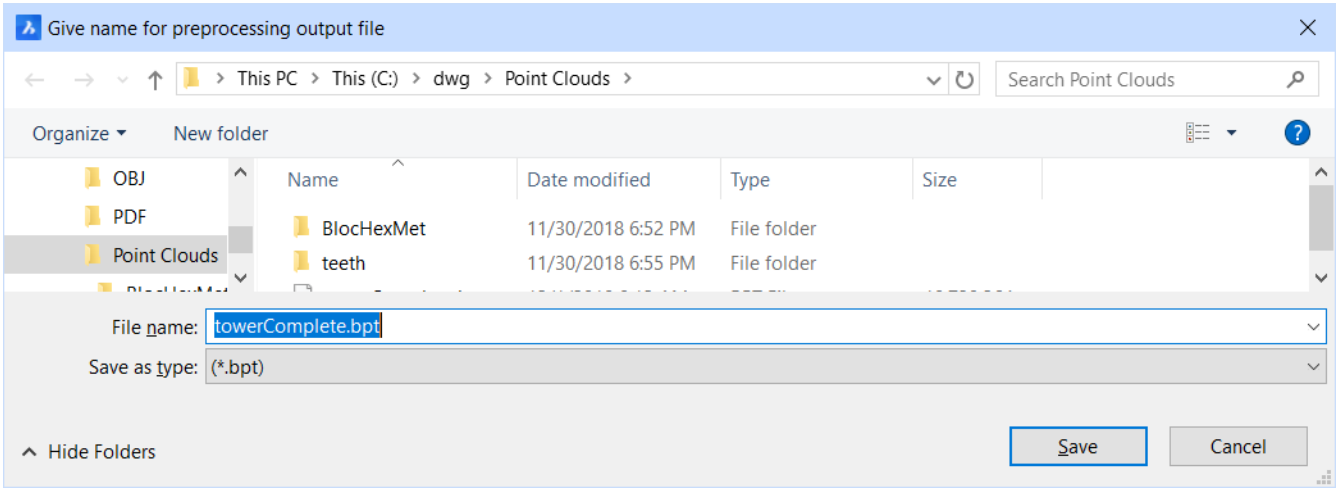
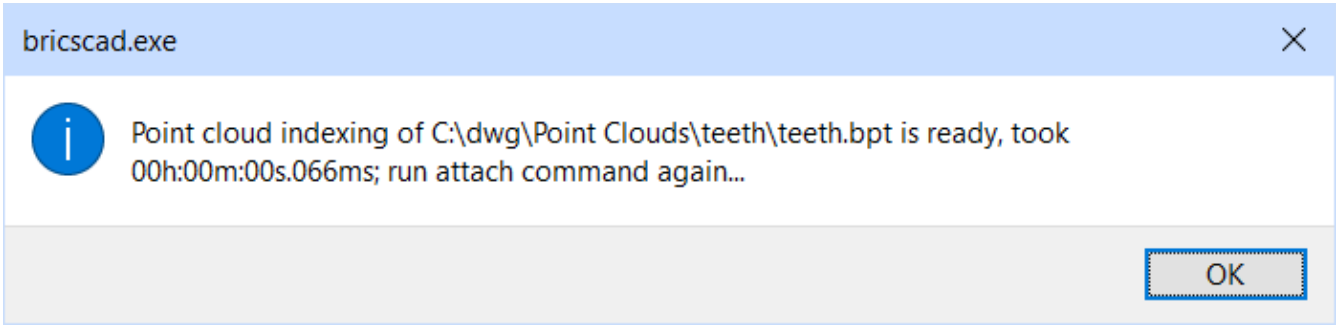


Figure 6.1819



Command Options

Option	Description
File name	Specifies the name of the point cloud file to process
Supported formats	Specifies the point cloud file formats supported by BricsCAD: • **LAS** - LASer (lidar) format • **PTS** - Leica PoinTS format (contains only x,y,z points) • **PTX** - Leica format enhanced with luminosity data
Open	Opens the point cloud file, and then prompts you for options.
Start preprocessing of raw point cloud data?	Large files can take a long time to process, and so you have the choice of not proceeding with the conversion: • **Yes** - carry on converting the point cloud file to BPT format • **No** - cancel the conversion process

Give unit for point coordinates.	Specify a type of unit, in case the point cloud needs to match a model in the drawing: • **Inches** • **Feet** • **Miles** • **miLlimeters** • **Centimeters** • **mEters** - default units • **Kilometers** • **micrOinches** • **milS** • **Yards** • **Angstroms** • **Nanometers** • **micRons**

Once the processing is finished, the resulting BPT file is attached to drawings with the PointCloudAttach (6.24.34) command.

Related Commands

AttachmentsPanelOpen (6.4.24) - displays the Attachments panel from which point clouds and other attachments can be loaded and controlled. PointCloudAttach (6.24.34) - opens a BPT point cloud file, and then attaches it to the current drawing. PointCloudColorMap (6.24.36) - colorizes the points based on Intensity, Elevation, and Classification of the point cloud. PointCloudPointSize (6.24.39) - specifies the size of points.

Figure 6.1820



6.24.61 PointCloudUncrop

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Removes all croppings from a point cloud.

Accessing the Command

Keyboard:**pointclouduncrop : pointclouduncrop Prompts you:** Select point cloud entity: (Select the point cloud.) All croppings of the selected point cloud are removed right away.

Command Options

No command options.

Related Commands

PointCloud (6.24.32)- opens the Drawing Explorer to the Point Cloud section; converts, loads, and unloads point cloud file attachments. PointCloudCrop (6.24.38) -crops a point cloud entity, displaying only the part of the point cloud inside or outside a prismatic or cylindrical volume.

6.25 Q

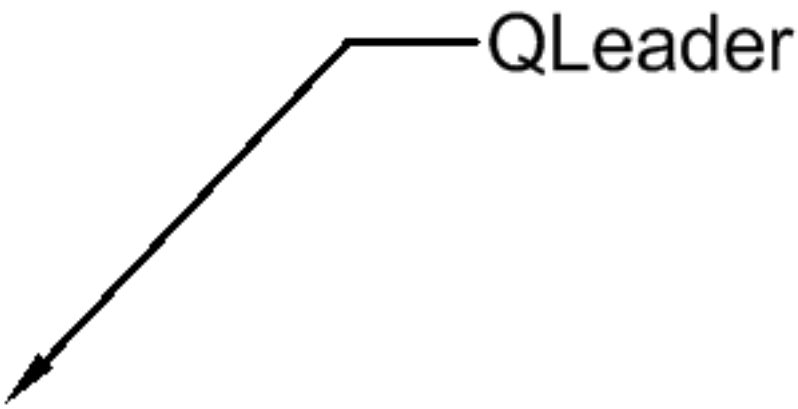
6.25.1 QLeader

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws leaders with a variety of annotation types; allows you to specify the properties through a dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Q/qleader-eg.png})

Figure 6.1821



Figure 6.1822



Accessing the Command

 command bar: **qleader** : **QLeader Prompts you in the command bar:** Start of leader <Settings>: **(Pick a point, enter x,y coordinates, or else press Enter for the settings dialog box)** Next point: **(Pick another point)** To point: <Undo>: **(Pick another point, or enter U to undo the last leader segment)** Specify text width: <0>: **(Enter a width for the text bounding box; enter 0 for no limit on the width)** First line of annotation text: **(Type the first line of the annotation text)** Next line of annotation text: **(Type the another line of text, or elsepress Enter to end the command)**

Command Options

Option	Description
Start of leader	Specifies the start of the leader, the end at which the arrowhead is displayed: • Pick a point • Enter x,y coordinates
Settings	Displays the QLeader Settings dialog box; see below
Next point	Specifies the vertex of the leader, the start of the second leader line: • Pick a point • Enter x,y coordinates
To point	Specifies the end of the leader, also known as the landing line): • Pick a point • Enter x,y coordinates Hold down the **Shift** key to draw a horizontal landing line.
Undo	Undoes (un-draws) the last-drawn leader lin
Specify text width	Specifies the width for the text bounding box: • Enter a number in drawing units • Enter 0 for no limit on the width When the length of the text exceeds the bounding box width, the text flows automatically to the next line.

First line of annotation text	Specifies the text for the first line of leader annotation: • Enter text • Press **Enter** to carry on to the second line of text
Next line of annotation text	Specifies the text for the second line of leader annotation. The prompt continues until you press **Enter** without first entering text.

Dialog Box Options

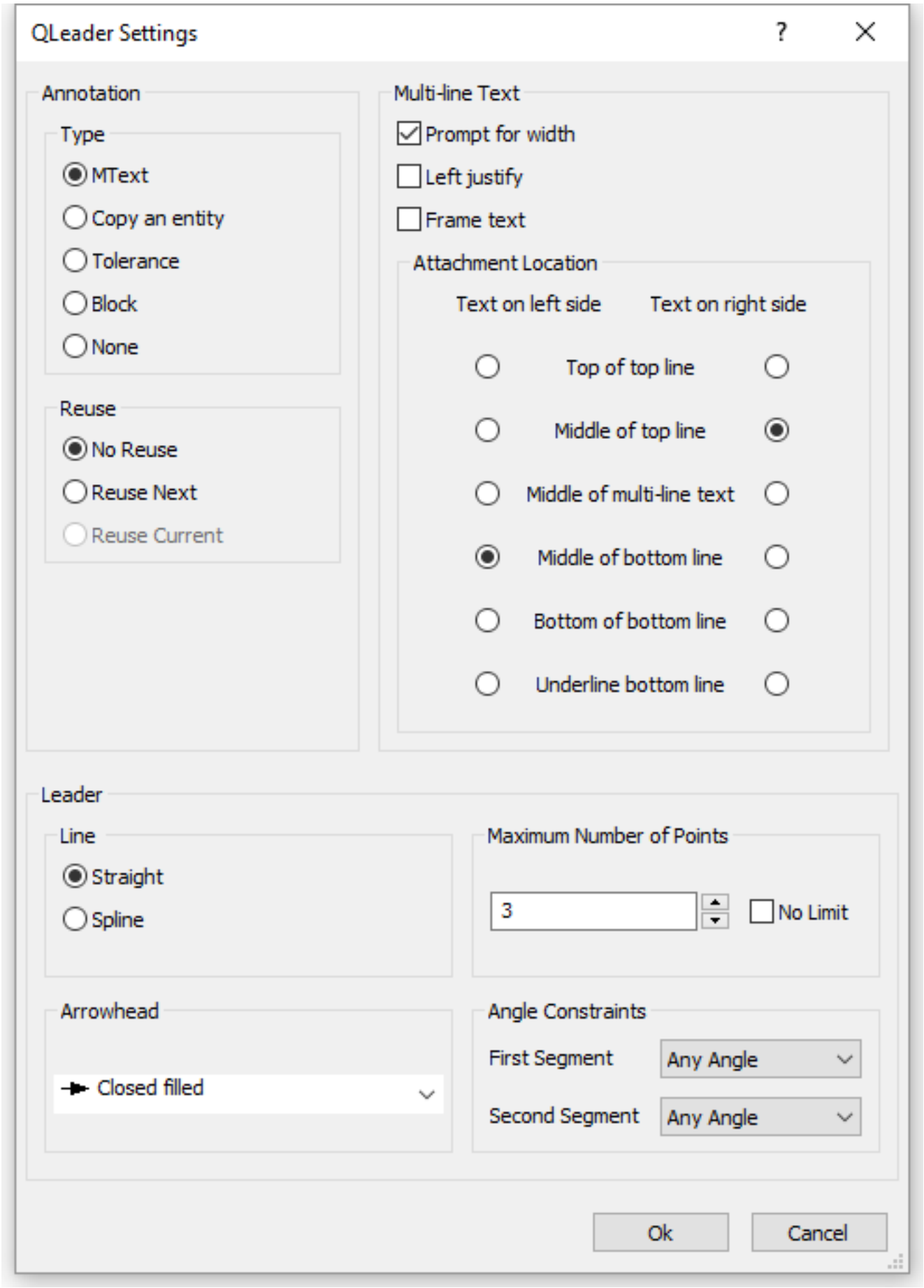
Enter the **Settings** options to see the dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdF

Option	**Description**
Annotation type	Specifies the type of annotation: • **Mtext** - text is placed as the leader line's annotation; a leader can have one or more lines of text. Although the mtext editor does not appear when entering text (as with the MLeader command), the mtext editor does appear when editing the annotation (edit the text by double-clicking it) • **Copy of an entity** - uses any other entity in the drawing as the annotation; prompts you: **Select an entity** - choose an entity, other than the leader itself. • **Tolerance** - uses tolerance notation for the annotation; displays the Geometric Tolerance dialog box. See theTolerance (6.29.17)command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • **Block** - uses a block as the annotation; displays prompts identical to that of the -Insert command: **? to list blocks in drawing/~ to open the file dialog/<Block to insert>** - **(enter an option.)** • **None** - no annotation is used

Multi-line Text	Specifies some multi-line text options: **Prompt for width** - toggles the use of the 'Specify text width' prompt: • **On** - displays the 'Specify text width' prompt and so wraps the text tot he specified width • **Off** - disables the prompt **Left justify** - determines the justification of the text: • **On** - forces the text to be left-justified always • **Off** - the text is left or right-justified, depending on the orientation of the landing line **Frame text** - toggles the use of a frame: • **On** - places a rectangle around the text • **Off** - no frame **Attachment location**- determines the relative location of where the mtext is attached to the landing line (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Q/ql 2-eg.png})
Reuse	Specifies how to reuse the annotation: • **No reuse** - does not reuse the previous annotation text; you will be prompted to enter new text each time you use this command • **Reuse next** - you will be prompted to enter new text, which is then reused for subsequent leader(s). • **Reuse current** - the current text is reused for each new leader, and you will not be prompted to enter new text.
Leader line	Specifies the style of leader line: • **Straight** - draws the leader lines as straight segments • **Spline** - draws the leader line as a smooth continuous spline
Maximum number of points	Specifies the maximum number of vertices for the leader, before it stops drawing them automatically: • Enter a number; the minimum number is 2 • **No limit** - keeps drawing leader segments until you press **Enter** at the 'To point' prompt
Arrowhead	Specifies the type of arrow at the start of the leader; choose from the droplist: • Choose an arrowhead type • **None** - no arrow is drawn • **Select arrow** - choose a block from those int he current drawing; displays the Select Custom Arrow Block dialog box To create a custom arrow, draw it to your liking, and then use the Block (6.6.11) command to turn it into a block.

Angle Constraints	Set an angle to constrain the first and second leader segments. The angles are increments; for instance, choose 15 allows you to draw the leader at 0, 15, 30, 45, and so on degrees. Click the droplist, then choose an option: • **Any Angle** - no constraint; hold down the **Shift** key to draw the leader at 90 degrees • **Horizontal** - forces the leader line to be drawn at 90 degrees only • **90°** - draws leader lines at 90-degree increments • **45°**- draws leader lines at 45-degree increments • **30°** - draws leader lines at 30-degree increments • **15°** - draws leader lines at 15-degree increments
OK	Saves the changes you made to the settings, closes the dialog box, and then prompts you to specify the first point of the leader.

Figure 6.1823



Related Commands

Leader (6.20.20) - places leaders with text, blocks, entities, or tolerances. DimLeader (6.11.13) - places leaders with text, blocks, entities, or tolerances. MLeader (6.21.20) - creates multileader entities using the current multileader style.

Figure 6.1824

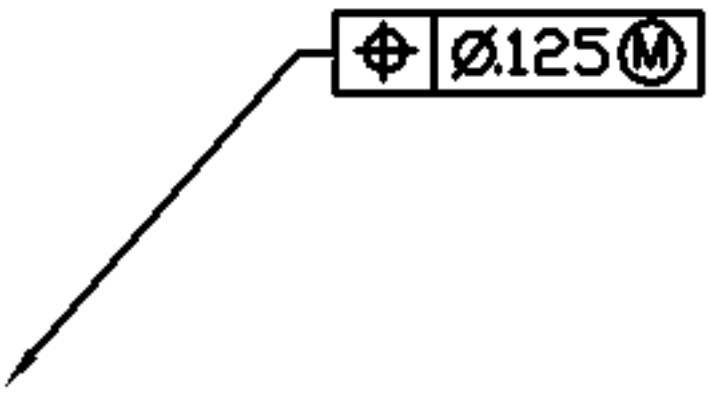


Figure 6.1825

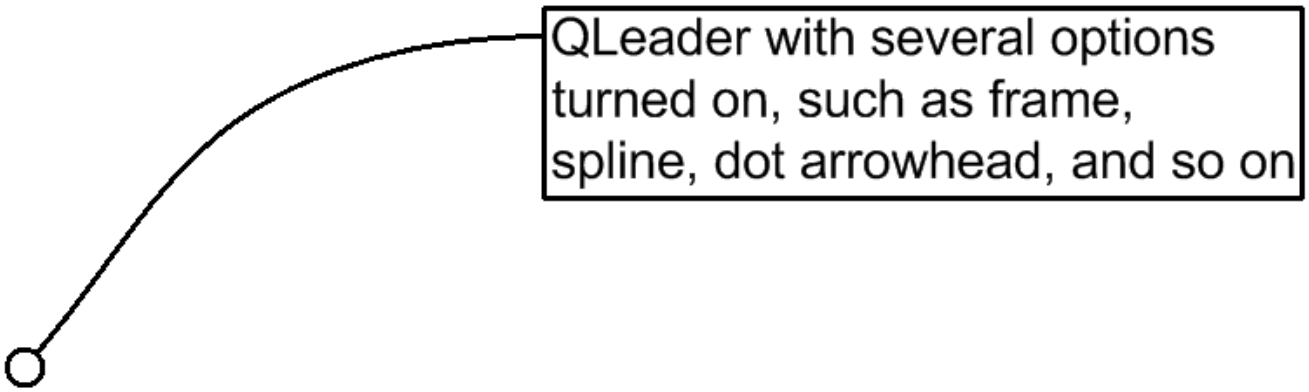


Figure 6.1826



6.25.2 QNew

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Starts a new drawing (short for "quick new") based on the default template file. **Note:** The New (6.22.4) command start a new drawing after prompting you to select a template file.

Accessing the Command

 command bar: **qnew** alias: **n** menu bar: **File | New** ribbon:**Home | File | QNew** toolbar: **Standard |**  : **QNew** Opens a new document tab based on the default template and user profile.

Command Options

No options.

Procedures

- To create a drawing using a default template drawing (1.14.1)
- To set the default template drawing (1.14.1)
- To create a drawing from scratch (1.14.1)
- To set the Plot style policy (1.14.1)
- To create a drawing using a template drawing (1.14.1)
- To create a drawing using the New command (1.14.1)
- To set the Template Path variable (1.14.1)
- To create a drawing using a wizard (1.14.1)

Related Commands

Units (6.30.8) - sets units through the Settings dialog box. -Units (6.30.9) - sets units through the command bar. NewWiz (6.22.6) - sets units and other parameters for new drawings. New (6.22.4) - opens new drawings.

Figure 6.1827



6.25.3 QPrint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Prints the drawing using the default print configuration, without displaying the Print dialog box (short for "quick print"). **Note:**This command does not work when the default printer is set to "None." For full control over the printing process, use the Print (6.24.46) command.

Accessing the Command

 command bar: **qprint** ribbon:**Home | Print/Plot | Print | Print** toolbar: **Standard |**  : **QPrint** Displays no prompts in the command bar; prints the drawing. As this command does not work when the default printer is set to "None," you will want toconfigure the drawing's print properties ahead of time with the PageSetup (6.24.1) command.

Command Options

No options.

Related Commands

Print (6.24.46) - plots drawings through a dialog box. Plot (6.24.26) - plots drawings through the command line. Preview (6.24.45) - previews drawings before they are plotted. PageSetup (6.24.1) - assigns plot style to drawings. PlotStyle (6.24.29) - sets the current plot style. StylesManager (6.27.55) - creates and attaches plot styles to drawings.

Figure 6.1828



6.25.4 QSave

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Saves the drawing immediately, without displaying the Save dialog box (short for ”quick save”).

Accessing the Command

 command bar: **qsave** shortcut: **Ctrl+S** menu bar: **File | Save** toolbar: **Standard |**  : **QSave**

- When the drawing has never been saved before
- When the drawing is opened as read-only

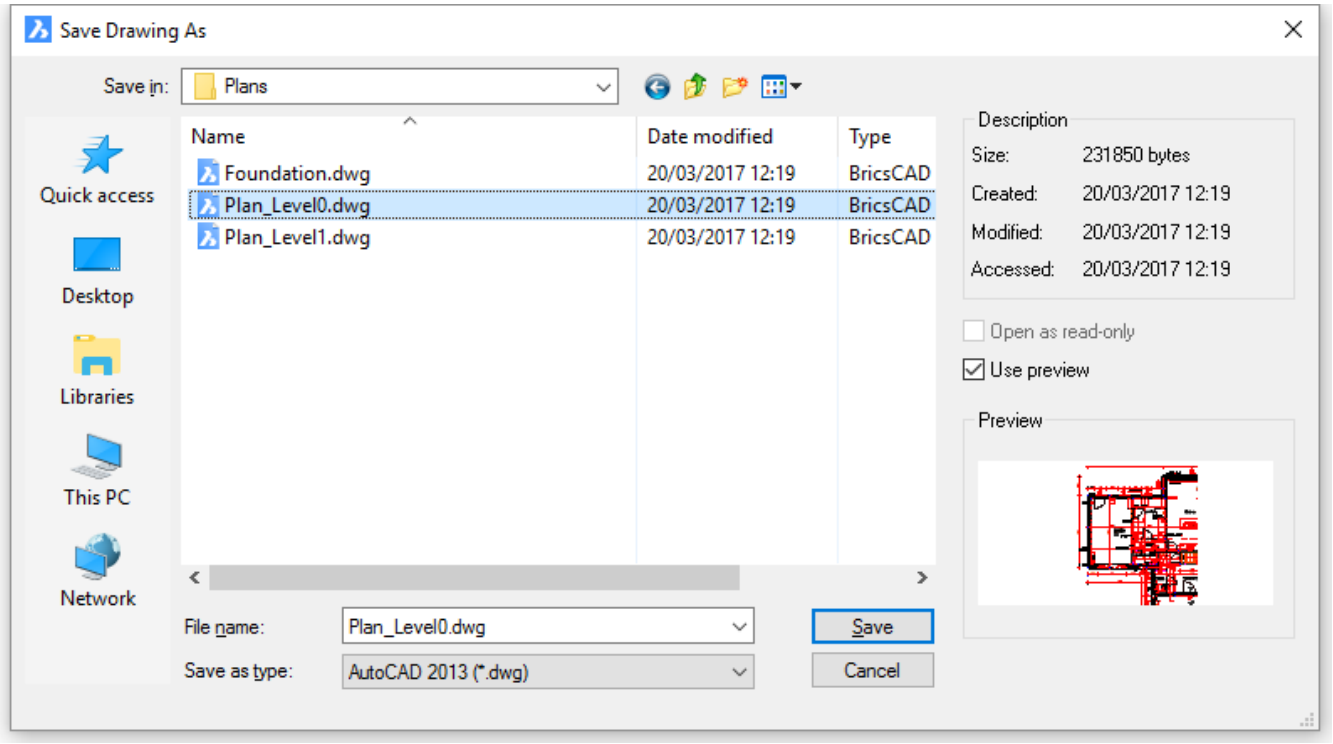
Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SaveAsDialog

1. (Optional) Enter a file name or accept the default name generated by the program, such as ’Drawing1.dwg’.
2. (Optional) Choose a folder in which to save the drawing.
3. (Optional) choose a file format from the **Save As Type** droplist.
4. **Click Save.**See theSave (6.27.1) command.

- When the drawing has been already saved at least once:

Displays no dialog box or prompt in the command bar.To save the drawing by another name, use the Save command.

Figure 6.1829



Command Options

No options.

Related Commands

Save (6.27.1) - saves the current drawing through a dialog box. SaveAll (6.27.2) - saves all open drawings. SaveAs (6.27.3) - saves drawings by another name. SaveAsR12 (6.27.4) - saves drawing in older formats. Export (6.13.15) - exports drawings in a variety of vector and raster formats.

Figure 6.1830



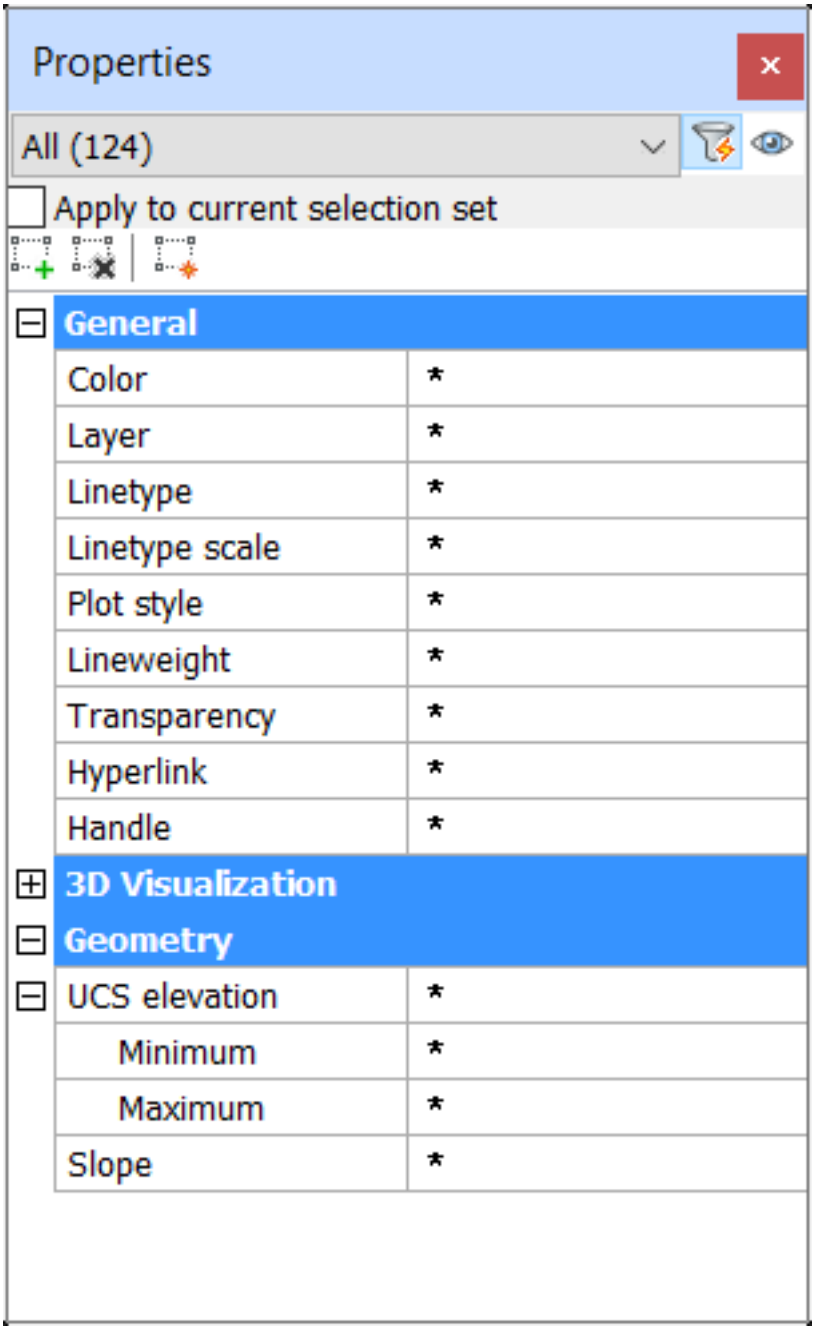
6.25.5 QSelect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Composes a selection set of entities based on their **properties**. **Note:** Tocreate selection set of entities based on their **location** in the drawing, see the Select (6.27.17) command.

Accessing the Command

command bar: **qselect** properties panel:  : **QSelect** Opens the Properties panel, switches to Quick Select mode, and then selects all entities in the drawing: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands panel.png}) The panel does not open whenthe drawing contains no entities or all layers are frozen.

Figure 6.1831



Panel Options

Option	Description
Entities	Specifies the entities to be included in the selection set;click the droplist, then choose an entity type in the list: • **All** - includes all non-frozen entities in the drawing • **EntityName** - limits the selection set to specific entities • **(3)** - reports the number of entities of that type in the drawing.
Apply to Current Selection Set	Toggles whether the filter is continuously applied to the current selection set: • **On** - entities that match the properties are added to the current selection set • **Off** - entities are not added Applies the filter to the current selection set; the program highlights the selected entities in the drawing.
 Add to Current Selection Set	Adds the entities that match the filters to the current selection set.
 Remove fromCurrent Selection Set	Removes the entities that match the filters from the current selection set.
 Add to New Selection Set	Creates a mew selection set from the the entities that match the filters, highlights the entities in the drawing, and then switches back to the Properties mode of the panel.
 Quick Select Mode	Toggles the Properties panel between properties and quick select mode;see the Properties (6.24.49) command
Property Preview	Toggles whether properties are previewed in real time: • **Yes** - properties of entities and settings change to show the property over which the cursor is hovering (it may take a second or two for the preview to appear) • **No** - properties are not previewed

Filter Options	Description

Entity Properties	The common properties of the entities in the drawing or in the current selection set are available. 1. Click the property field, such as Color, then do choose a value from the list, or type a value in the entry field. 2. Click * to choose an operator: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so The properties listed in the panel vary depending on the entity types selected.

Procedures

- Creating a selection set (1.23.7).

Related Commands

dmSelect (6.12.26) - selects edges and faces of 3D solids or surfaces based on their geometric properties. Select (6.27.17) - selects one or more entities; can be accessed later with the Previous option. SelectSimilar (6.27.21) - selects all entities of the same type as the current selection, based on a set of matching properties, stored in the **SELECTSIMILARMODE** system variable DdSelect (6.10.20) - sets selection properties in the **Entity Selection** section of the **Settings** dialog box. SelGrips (6.27.22) - selects entities and displays grips.

Figure 6.1832

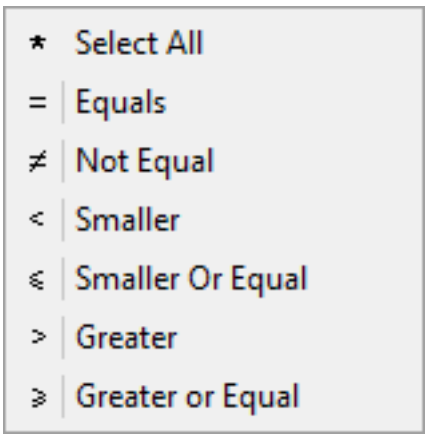


Figure 6.1833



6.25.6 QText

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Changes the display of text between text and rectangles (short for "quick text"). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Q/qtext-on-eg.png})

Accessing the Command

command bar: **qtext** alias: **qt** : **QText Prompts you in the command bar:** QTEXTMODE is currently off: ON/Toggle/<Off>: **(Enter an option.)** The change in text does not occur until you enter the Regen (6.26.15) command: : **Regen Note:** This command is useful when drawings contain much text, thereby slowing down the display of the drawing, but you still need to see the location of the text.

Command Options

Option	Description
ON	Turns on quick text mode, rendering all text -- text, mtext, attributes, dimension text, and so on -- as rectangles: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Q/qt on-eg.png}) Remember to run the Regen command.
Toggle	Changes quick text mode: • If on, turns it off • If off, turns it on
Off	Turns off quick text mode, returning text to its normal display: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Q/qt off-eg.png}) Remember to run the Regen command.

Related Commands

Text (6.29.12) - places single-lines of text in drawings. MText (6.21.35) and -MText (6.21.33) - places paragraph text. AttDef (6.4.27) - defines attribute text. DdEdit (6.10.14) - edits text. Style (6.27.53) - creates and modifies text styles through the Drawing Explorer. -Style (6.27.54) - creates and modifies text styles through the command bar. TxtExp (6.29.27) - explodes text into polylines. Properties (6.24.49) - modifies the properties of text. Find (6.14.8) - finds and replaces text.

Figure 6.1834

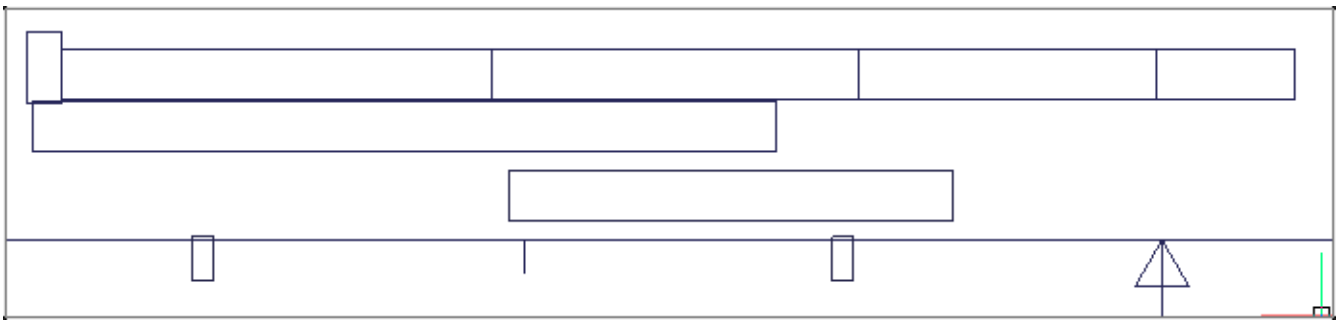


Figure 6.1835

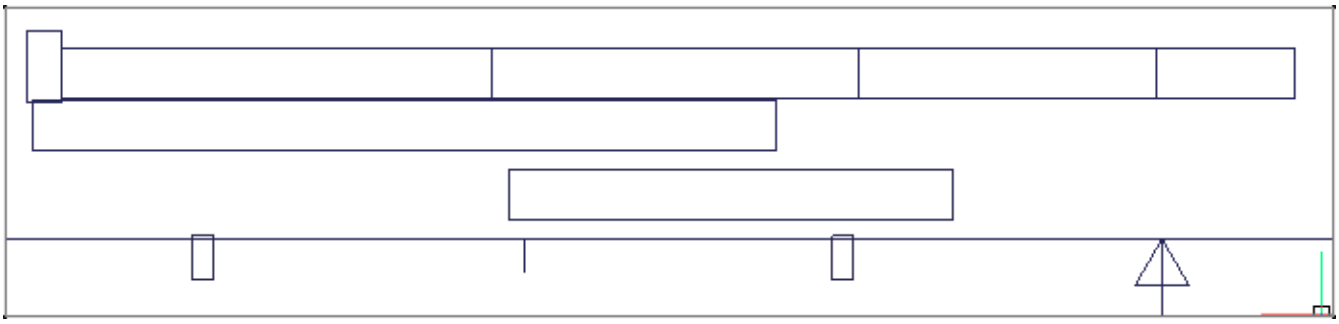


Figure 6.1836

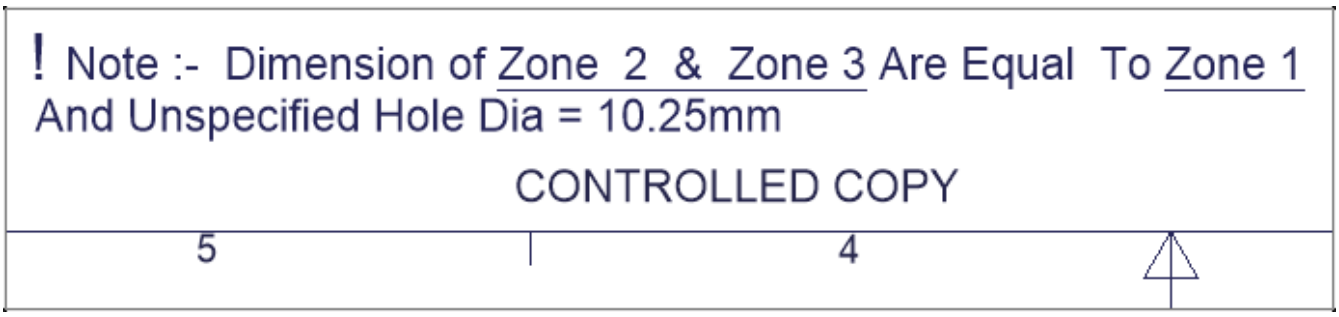
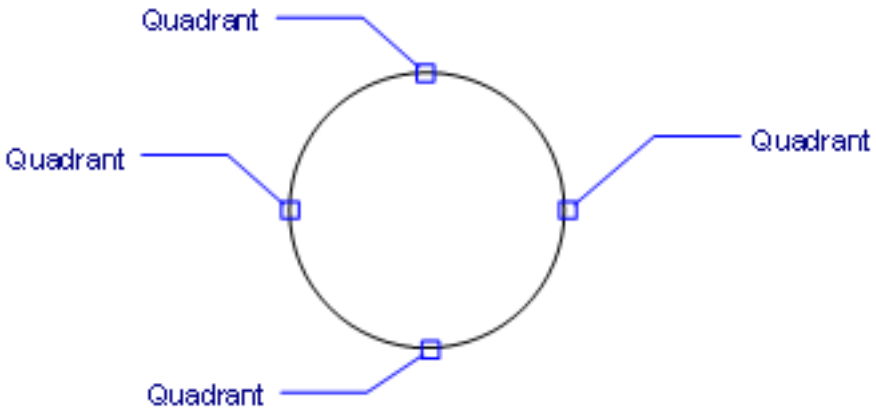


Figure 6.1837



Figure 6.1838



6.25.7 Quadrant

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Snaps to the quadrant points of circles, arcs, and polyarcs -- located at the 90-, 180-, 270-, and 0-degree points of the curve. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sourc

Accessing the Command



command bar: **quadrant** transparent: **qua** ribbon:**Home | Entity Snaps | Snap to Quadrant** toolbar: **Entity Snaps |**  : **Quadrant** Displays no prompts in the command bar.The quadrant points are located at the 90-degree points of circular entities.When quadrant snap is on, the cursor snaps to the 90-degree points of circles, arcs, and polyarcs.

Command Options

Option	Description
Quadrant	Entering this command toggles Quadrant entity snap. It snaps to the 90-degree quadrant points on curved entities. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. Extension (6.13.20) - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Tangent (6.29.9) - toggles tangent snap mode.

Figure 6.1839



6.25.8 Quit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Closes all open drawings and exits BricsCAD; offers the opportunity to save unsaved drawings.

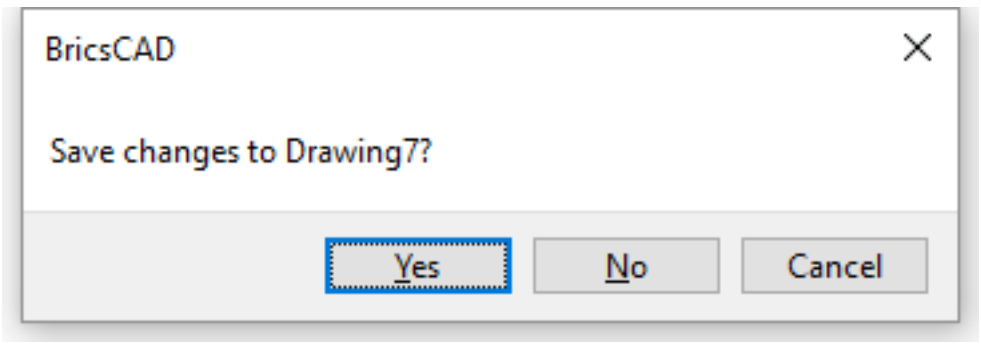
Accessing the Command

command bar: **quit** shortcuts: **Ctrl+Q** **Alt+F4** alias: **exit** menu bar: **File** | **Exit** : **Quit**

- When no drawings need to be saved, this command exits without displaying a dialog box or prompts in the command bar.
- If drawings are unsaved, displays this dialog box for each drawing that needs to be saved to disc:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/QuitDialog.png})

Figure 6.1840



Command Options

Option	Description
Yes	Saves the drawing, and then exits BricsCAD.
No	Does not save the drawing, and then exits BricsCAD.
Cancel	Cancels the command; BricsCAD does not exit.

Related Commands

Open (6.23.8) - opens drawing files. Close (6.9.25) - closes the current drawing. WClose (6.33.4) - closes the current window.

6.26 R

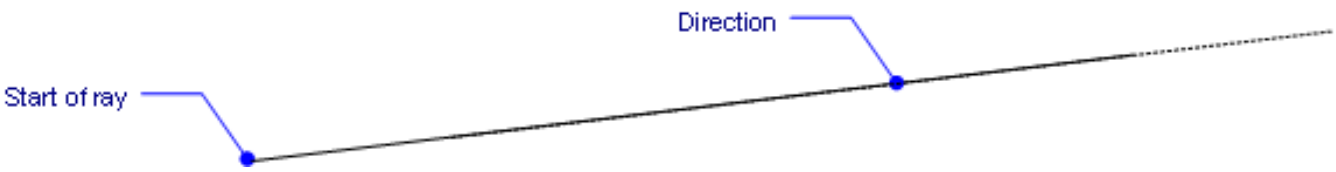
6.26.1 Ray

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws rays, semi-infinite lines with start point at one end and then continue to beyond infinity at the other. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/im

Figure 6.1841



Figure 6.1842



Accessing the Command

 command bar: **ray** menu bar: **Draw | Ray** ribbon:**Draw | Draw | Line | Ray** toolbar: **Draw | Line |**  : **Ray**
Prompts you in the command bar: Infinite ray: [Bisect/Horizontal/Vertical/Angle/Parallel] <Start of ray>: **(Pick a point, or else enter an option.)** Direction: **(Pick a point, or else enter x,y coordinates)** Direction: **(Pick another point, or else press Enter to exit command)** The program draws a ray from the starting point to ”infinity.” **Infinity** is defined as always appearing in the drawing, no mater how back back you zoom.

Command Options

Option	Description
Start of ray	Specifies the starting point of the ray: • Pick a point in the drawing • Enter x,y coordinates
Direction	Specify the direction of the ray from the starting point: • Pick a point in the drawing • Enter x,y coordinates • Press **Enter** to end the command The ’Direction’ prompt repeats so that you can draw multiple rays from the same starting point; press **Enter** to exit the command.

Bisect	Draws a ray that bisects entities perpendicularly; prompts you: **Entity/<Vertex point>** - **(enter an option)**: **Entity** - draws rays perpendicular to the midpoints of lines and polyline segments, and the center point of arcs, polyarcs, and splined polylines; prompts you: 1. **Select line,arc, or polyline segment to bisect** - **(select a line, an arc, a polyline segment, or a polyarc.)** Line and polyline segments receive rays perpendicular to the segment's midpoint: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so Arcs and polyarcs receive rays drawn perpendicular to the arc's center point and through its midpoint: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so 2. **Select side for ray bisection** - **(choose the side on which the ray should emerge.)** 3. **Select line,arc, or polyline segment to bisect** - **(the command repeats its prompts so that you can draw rays on other entities, or else press Enter to exit the command.)** This option works with splined polylines, but not with spline entities.
Vertex point	Draws rays that bisect two points. The vertex is at the initial 'Set vertex point' prompt, and then you are prompted to pick two more points, which the ray will bisect: 1. **Set vertex point or [Entity]**- **(pick a point to start the ray.)** 2. **Bisection angle start point** - **(specify one part of the bisection angle.)** 3. **Bisection angle end point** - **(specify the other part of the bisection angle.)** 4. **Bisection angle end point** - **(specify another bisection, or else press Enter to end the command.)**
Horizontal	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Draws rays horizontal to the current UCS; prompts you: 1. **Location** - **(pick the starting point of the ray, or enter x,y coordinates.)** 2. **Location** - **(the prompt repeats so that you can draw more horizontal rays with different starting points; press Enter to exit the command.)**

Vertical	Draws rays vertical to the current UCS; prompts you: 1. **Location** - **(pick the starting point of the ray, or enter x,y coordinates.)** 2. **Location** - **(the prompt repeats so that you can draw more vertical rays with different starting points; press Enter to exit the command.)**
Angle	Draws one or more rays at a specified angle; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **Reference/<Enter angle (0)>** - **(enter an angle.)** 2. **Location** - **(pick the starting point of the ray, or enter x,y coordinates.)** 3. **Location** - **(the prompt repeats so that you can draw more angled rays with different starting points; press Enter to exit the command.)** **Reference** - draws one or more rays at an angle relative to another entity; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **Select reference entity** - **(choose an entity, such as a line or another ray, to provide the reference (”base”) angle.)** 2. **Enter angle** - **(specify the angle, will will be measured relative to the entity = angle of the reference entity + angle.)** 3. **Location** - **(specify the angle’s start point; need not coincide with the entity’s end points.)** 4. **Location** - **(the prompt repeats so that you can draw more angled rays with different starting points; press Enter to exit the command.)**

Parallel	Draws rays parallel to lines and polyline segments only; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Set offset distance for parallel infinite line or [Through point] <Through>** - **(enter an option)**: 1. **Offset distance for parallel infinite line** - specifies the offset distance; enter a value or pick two points; prompts you: 2. **Select entity for parallel infinite line** - **(choose the entity to offset with a ray.)** 3. **Side for parallel infinite line** - **(specify the side on which to place the ray; the ray's start point is parallel to the line's start point.)** 4. **Select entity for parallel infinite line**- **(the prompts repeat themselves for drawing more rays at different offset distances; press Enter to exit.)** **Through point** - **(specify the point through which to draw the ray; this point also specifies the ray's start point.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **Select entity for parallel infinite line** - choose the entity to offset with a ray. 2. **Side for parallel infinite line** - specify the side on which to place the ray; the ray's start point is parallel to the line's start point. The two prompts repeat for drawing multiple rays at different offset distances; press **Enter** to exit.

Procedures

- Drawing rays (1.17.10)

Grips Editing

Rays can be edited directly through grips:

1. Select the ray. Notice that it has two grips.
2. Drag the grips to perform the following actions:

- Drag the endpoint grip to move the ray.
- Drag the other grip to rotate the ray.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image762.png})

Figure 6.1843



Related Commands

XLine (6.34.5) - draws infinite lines. Properties (6.24.49) - modifies the properties of rays and xlines.

Figure 6.1844

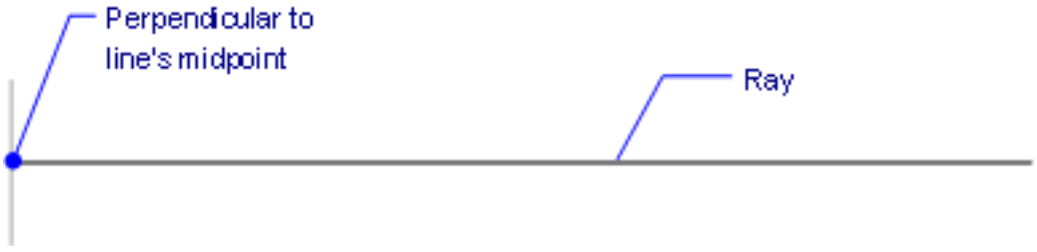


Figure 6.1845

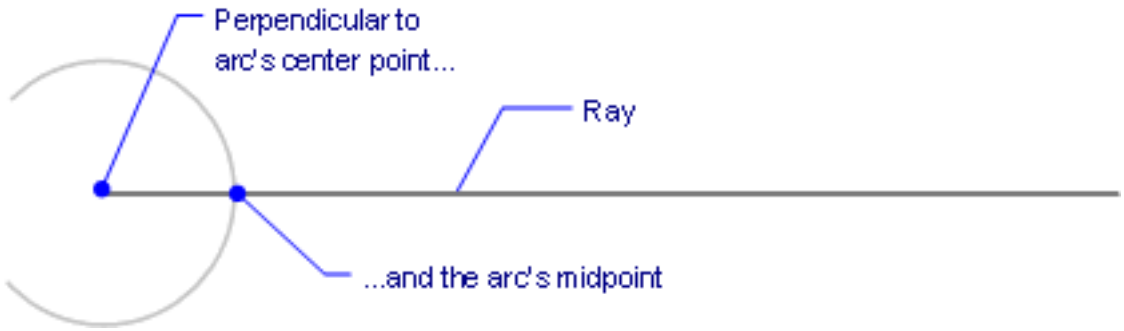


Figure 6.1846

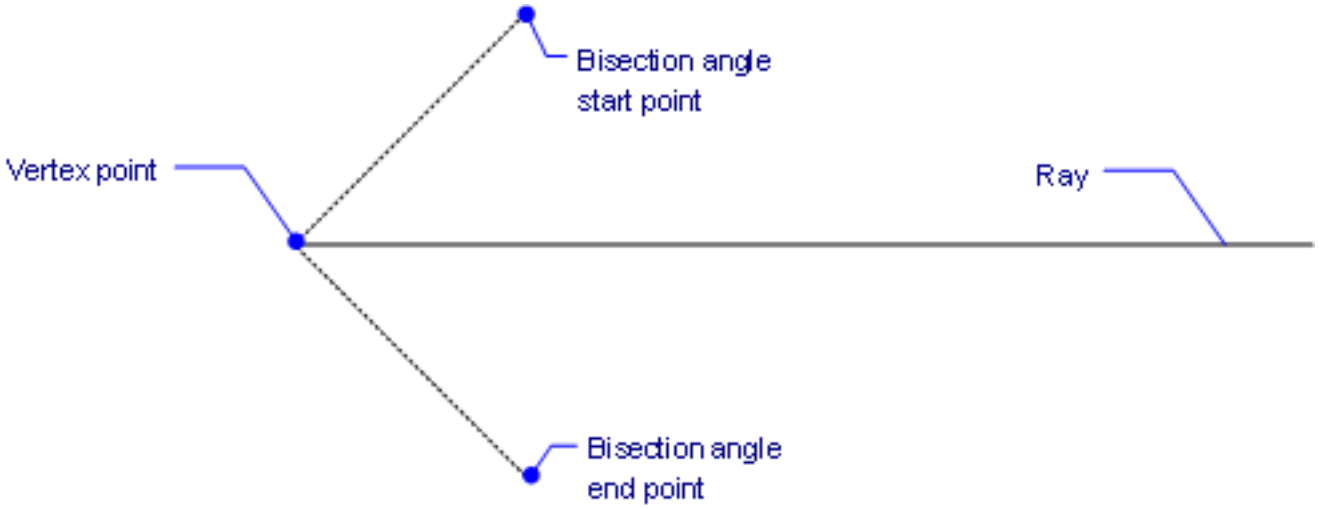


Figure 6.1847

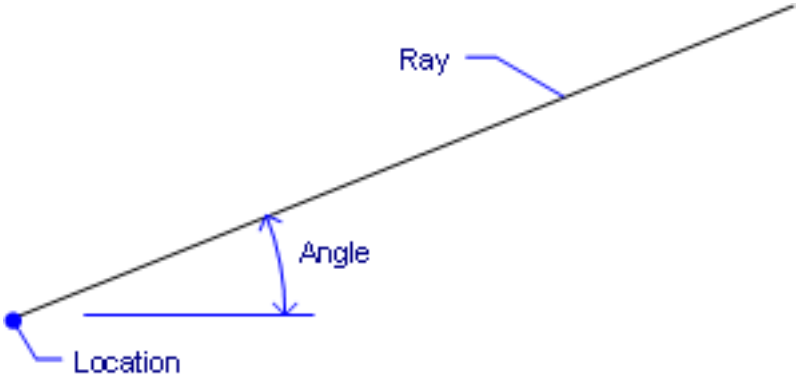


Figure 6.1848

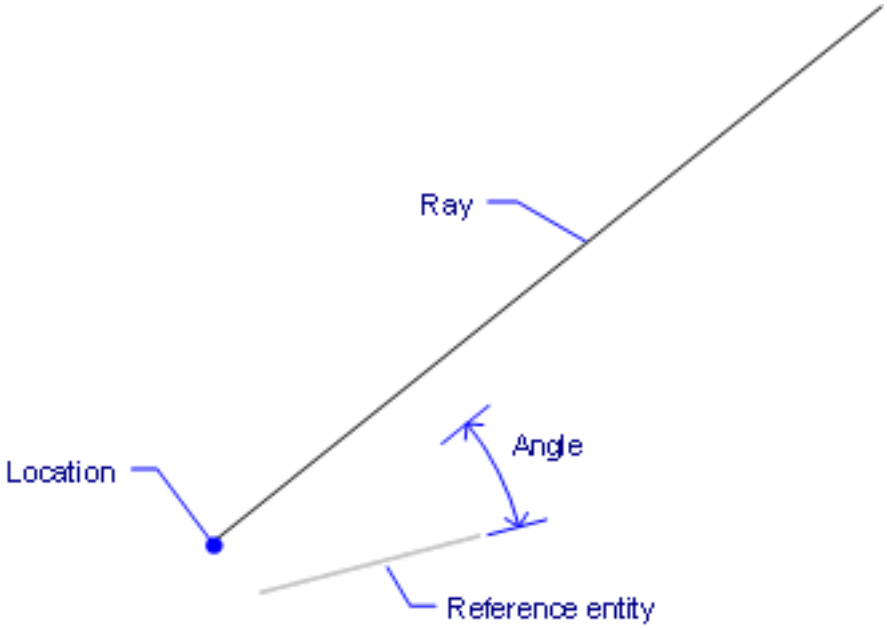


Figure 6.1849

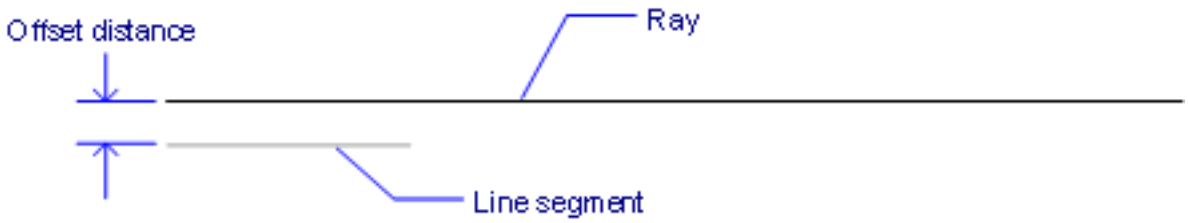


Figure 6.1850

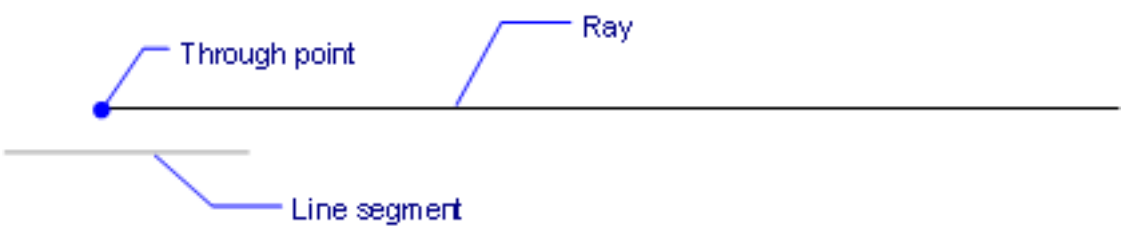


Figure 6.1851



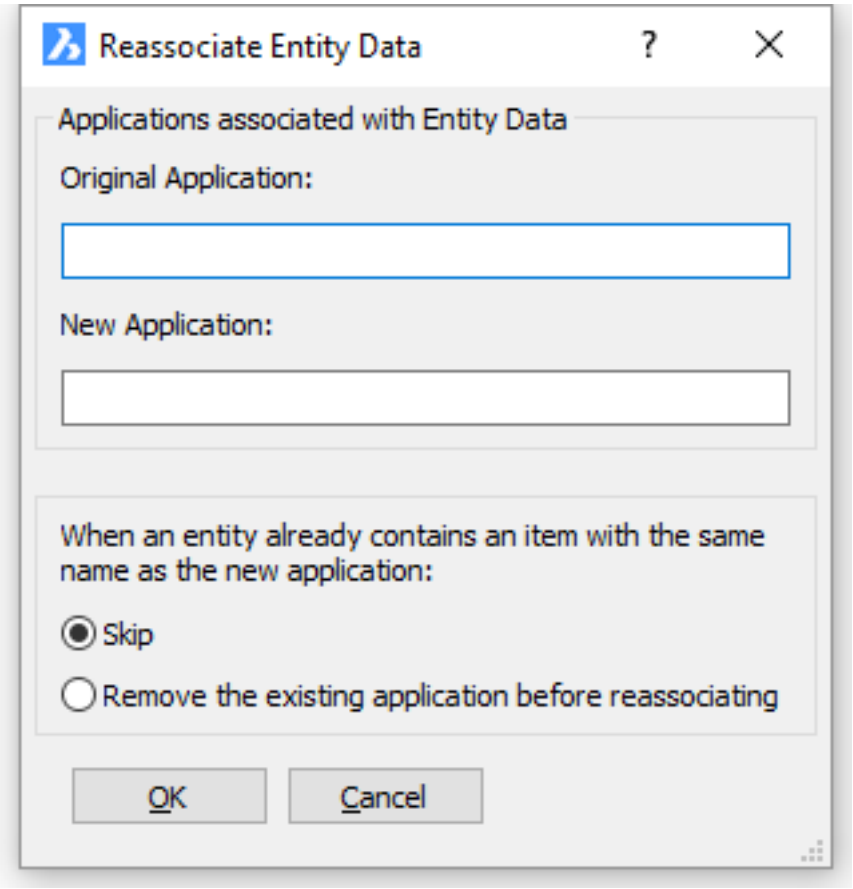
6.26.2 ReAssocApp

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Associates extended entity data with a specific application (short for "re-associate application").

Accessing the Command

 command bar: **reassocapp** menu bar: **Tools | Entity Data | Reassociate Entity Data** toolbar: **Entity Data |** 
: **ReAssocApp** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/
Enter the names of original and new applications, and then click **OK**.

Figure 6.1852



Command Options

Option	Description
Original Application	Specifies the name of the original application.
New Application	Specifies the name of the new application. Entities will be reassociated with this application name.
Skip	Ignores the change.
Remove the existing application before reassociating	Replaces the application name.

Related Commands

CopyEData (6.9.53) - copies extended entity data to other entities. DelEData (6.10.25) - erases extended entity data from entities. EditEData (6.13.3) - creates and edits extended entity data. MoveEData (6.21.32) - moves extended entity data to other entities.

Figure 6.1853



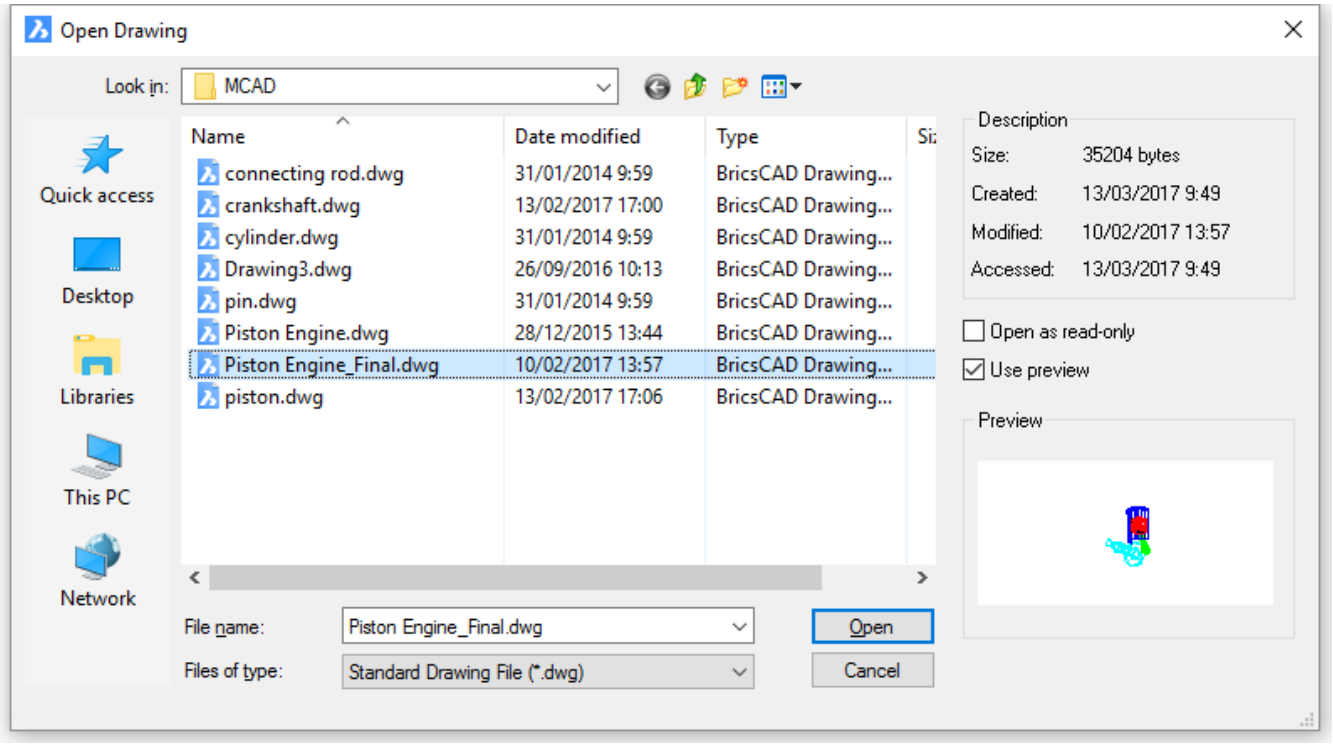
6.26.3 Recover

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Recovers data from a damaged DWG file.

Accessing the Command

 command bar: **recover** menu bar: **File | Recover** ribbon:**Home | File | Recover** : **Recover** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/RecoverDialog.png}) Choose a file, and then click **Open**. When the drawing file contains damaged data, the program reports the changes it made, automatically.For example Name: Dwg file Value: CRC does not match in Object (167); Name: AcDbPolygonMeshVertex(130); Value: Owner Id (0); Validation: Invalid; Replaced by: Set to AcDbPolygonMesh(F0). When the drawing file contains no errors, no report is given. When the recovery is complete, the drawing is displayed.

Figure 6.1854



Command Options

No options.

Procedures

- To analyze the integrity of the current drawing (1.14.9)
- To repair a drawing (1.14.9)

Related Commands

Audit (6.4.32) - checks drawings that are open. ProxyInfo (6.24.51) - displays a information about proxy entities, contained in the current drawing, in a dialog box. RecoverAll (6.26.4) - recovers a drawing with all nested XRef drawings, each drawing is saved with .BAK copy and .ADT report.

Figure 6.1855



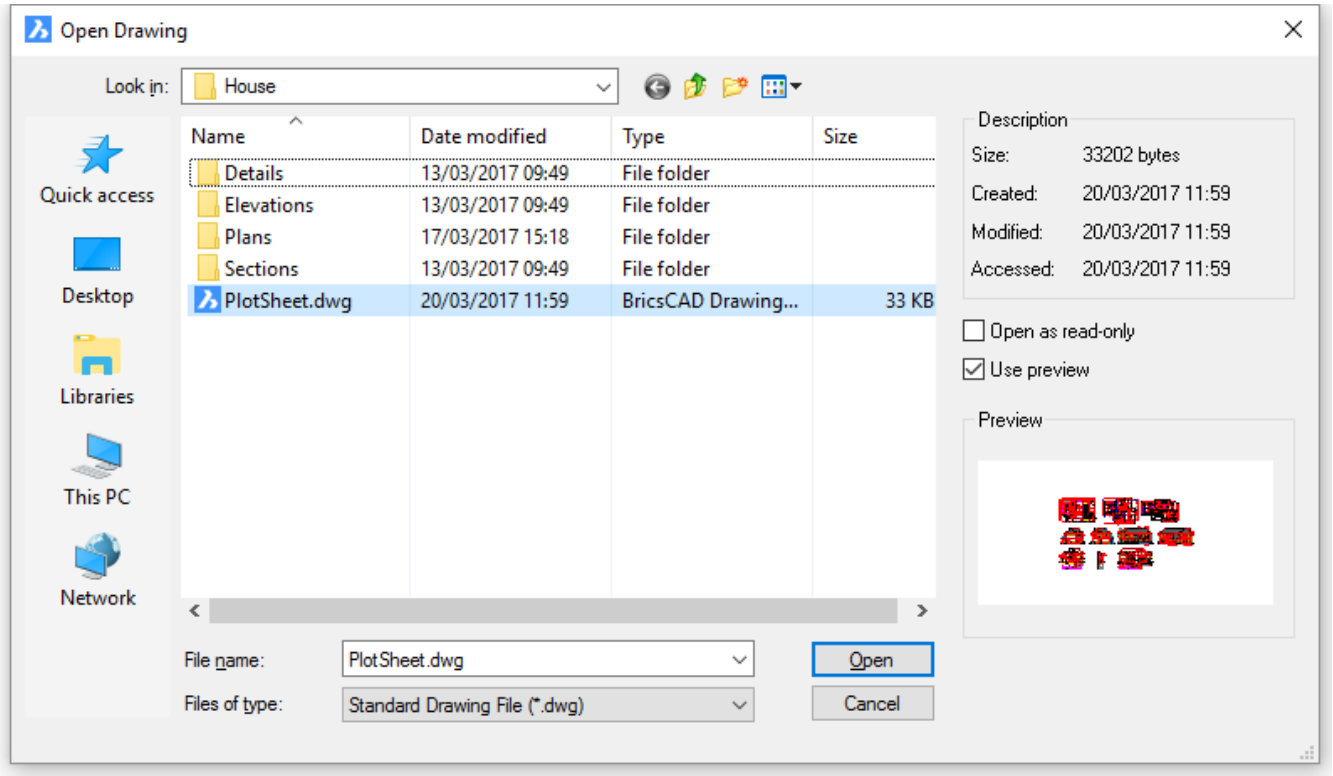
6.26.4 RecoverAll

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Recovers a drawing file, along with all nested xref'ed drawings; each drawing is saved with .BAK copy and .ADT report.

Accessing the Command

command bar: **recoverall** : **RecoverAll** Displays a File dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/
Select the drawing, the click the **Open** button. The selected drawing file and all attached xrefs, including all nested xrefs, are opened, repaired, saved and closed. The report of the audit report of the repair is saved to a *.adt file.

Figure 6.1856



Command Options

No command options.

Related Commands

Audit (6.4.32) - repairs drawings that are open. Recover (6.26.3) - recovers data from damaged DWG files.

Figure 6.1857



6.26.5 RecScript

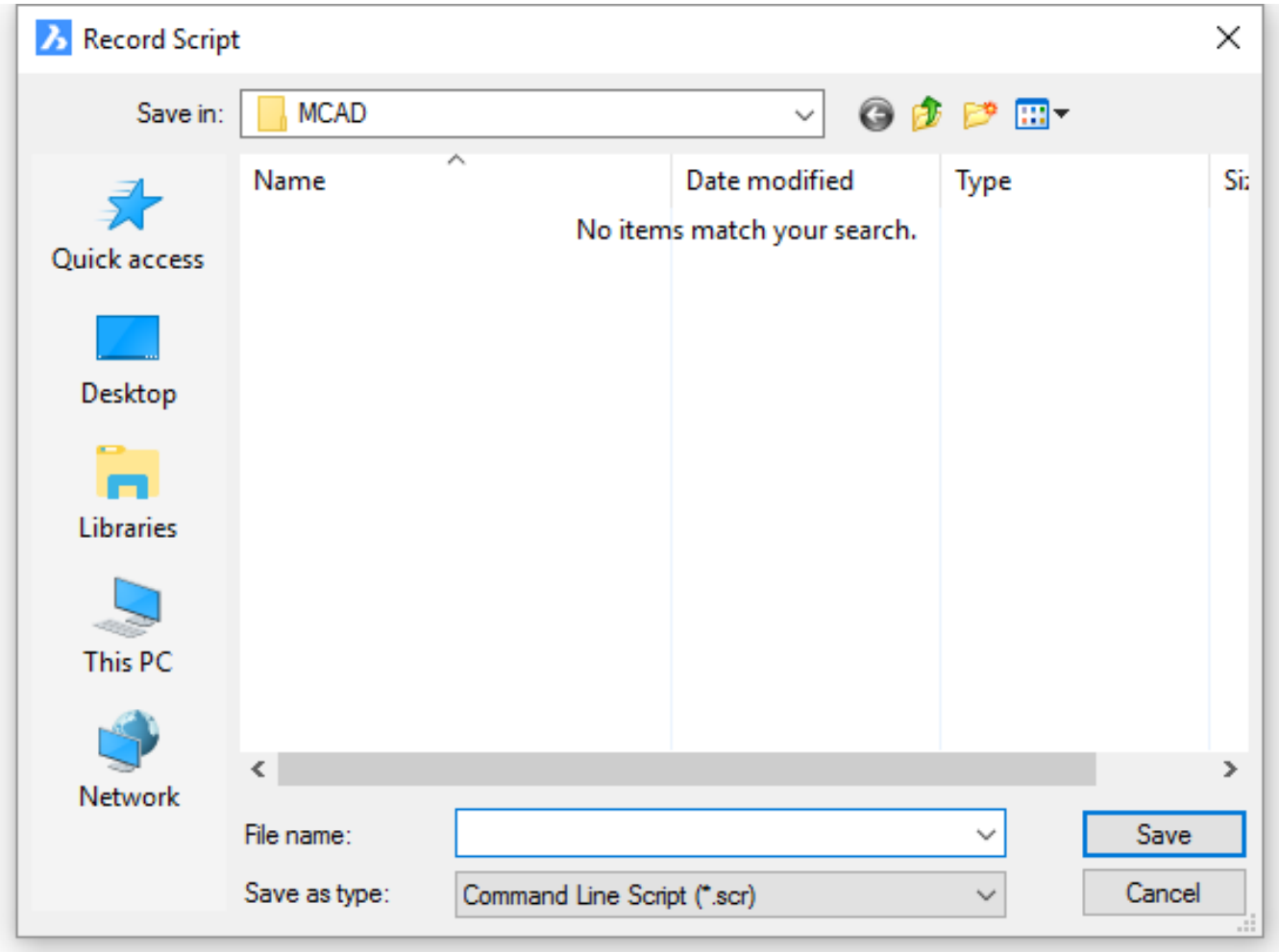
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Records keystrokes to an SCR file for playback with the Script (6.27.9) command (short for ”record script”).

Accessing the Command


command bar: **recscript** menu bar: **Tools | Record Script** ribbon:**Tools | Action Recorder | Record Script** toolbar: **Tools** |  : **RecScript** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/

1. Specify a file name, and then click **Save**.
2. Enter drawing and editing commands, and pick points in the drawing.
3. When done, enter the StopScript (6.27.48) command.
4. To playback scripts, use the Script (6.27.9) command.

Figure 6.1858



Command Options

Option	Description
File name	Specifies the name of the SCR script file.

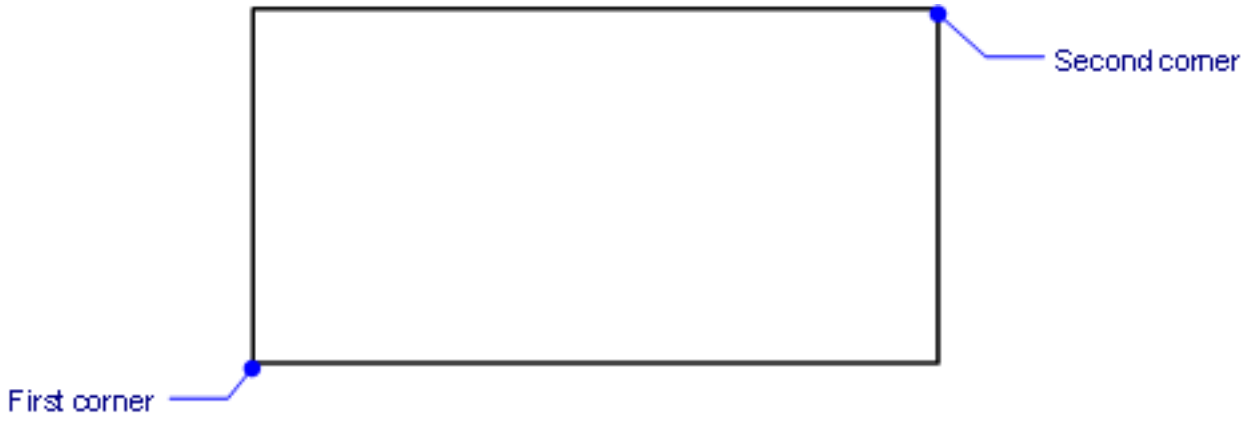
Related Commands

StopScript (6.27.48) - stops recording the script, and saves it to an SCR file. Script (6.27.9) - loads and runs a SCR script file. RScript (6.26.36) - reruns the script. Delay (6.10.23) - pauses the execution of the script for the specified number of milliseconds. Resume (6.26.28) - resumes a stopped script.

Figure 6.1859



Figure 6.1860



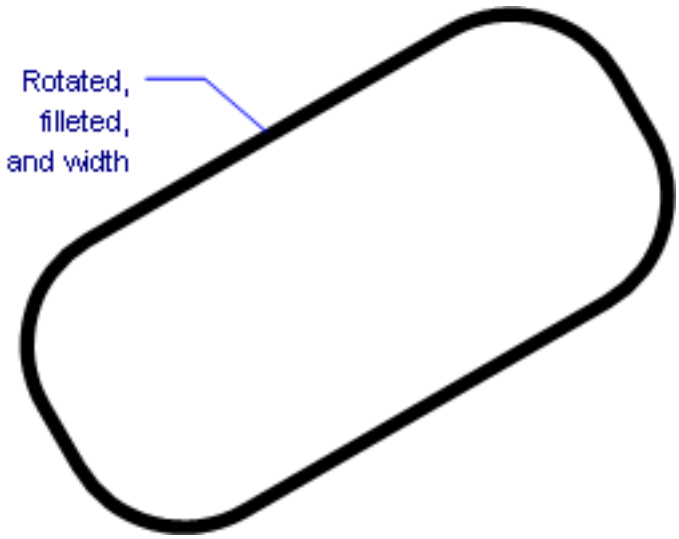
6.26.6 Rectang

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws rectangles and squares in a variety of styles from polylines (short for "rectangle"). (\vref {fig:C:/Dev/Task-2448/images/staticb

Accessing the Command

 command bar: **rectang** aliases: **rec rect rectangle** menu bar: **Draw | Rectangle** ribbon:**Draw | Draw | Polygons | Rectangle** toolbar: **Draw | Polygon |**  : **Rectang Prompts you in the command bar:** Select first corner of rectangle or [Chamfer/ Fillet/ Rotated/ Square/ Elevation/ Thickness/ Width of line/Area/Dimensions]: **(Pick a point, or enter an option)** Other corner of rectangle: **(Pick another point)** Rectangles are created as closed polylines, and so you can edit them later with the PEdit (6.24.20) command. During the Rectang command, you can combine options, such as creating a rotated rectangle with width and filleted corners: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image481.png})

Figure 6.1861



Command Options

Option	Description
Select first corner of rectangle	Specifies the first corner of the rectangle.
Other corner of rectangle	Specifies the opposite corner of the rectangle.
Chamfer	Applies a chamfer to all four corners of the rectangle; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **First chamfer distance to use for all rectangles or [use Default setting] <0.00>** - **(specify a distance or press Enter for the default.)** 2. **Second chamfer distance to use for all rectangles <1.00>** - **(specify another distance or press Enter for the default.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i The same chamfer distances are applied to all four corners.You can specify chamfer distances only; you cannot specify a chamfer angle.You can edit the chamfers laterwith the Chamfer (6.9.9) command.This command does not use the ChamferA and ChamferB system variables. When a chamfer distance is different from 0, the prompt changes to: **First chamfer distance to use for rectangles or [turn chamfer Off/use Default setting] <1.00>** - **(press O to reset the distance to 0.)**
Fillet	Applies a the same fillet radius to all four corners; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Fillet distance to use for all rectangles** - **(specify the fillet radius with a positive number.)** The same size of fillet is applied to all four corners.When the fillet radius is larger than the rectangle, an invisible rectangle is drawn. When fillet radius different from 0, prompts you: **Off/Default/Fillet distance to use for rectangles** - **(press O to reset the distance to 0.)**
Elevation	Specifies the elevation of the rectangle; prompts you: **Elevation to use for all rectangles** - **(specify the elevation as either a positive or negative number.)**The elevation is the height above or below the x,y-plane.
Rotated	Rotates the rectangle; delays the prompt until after the 'Other corner of rectangle' prompt: **Rotation angle for rectangle** - **(specify the rotation angle; pick a point or enter an angle.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Positive angles are measured counter-clockwise from the positive x axis.
Square	Draws a square; delays the prompt until after the 'Select first corner of rectangle' prompt: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Second corner of square** - **(specify the opposite corner; pick a point, or enter x,y coordinates.)**

Thickness	Specifies the thickness of the rectangle; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Rectangle thickness** - **(specify the thickness; enter 0 or a positive number.)** The thickness is the height of the rectangle in the z direction; it is measured upwards from the elevation.When thickness is different from 0, prompts you: **Rectangle thickness or [use Default] <2.00>** - **(press Enter to use the default thickness, or else enter a new one.)**
Width	Specifies the width of the rectangle's lines; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Width to use for all rectangles** - **(enter 0 or a positive number.)** The width is applied equally to all four segments making up the rectangle.To give the rectangle tapered widths, edit the rectangle with the PEdit (6.24.20) command.
Area	Specifies the size of the rectangle by its area; prompts you: 1. **Enter Area of rectangle <current value>** - **(enter a non-zero value or press Enter to accept the current value.)** 2. **Calculate rectangle dimensions based on [Length/Width]<Length>**- **(enter L or W.)** 3. **Enter rectangle length <current value>**- **(enter a non-zero value for the length of the rectangle)**; the program calculates the width **Enter rectangle width <current value>**- **(enter a non-zero valuefor the width of the rectangle)**; the program calculates the length 4. **Select first corner of rectangle** - **(pick the position of one corner of the rectangle, or enter x,y coordinates.)** 5. **Other corner of rectangle** - **(as you move the cursor, the program ghosts the four possible locations of the rectangle, relative to the first corner point.)**

Dimensions	Specifies the length and width of the rectangle; prompts you: 1. **Length to use for rectangles <current value>** - **(enter a non-zero value, press Enter to accept the current value, or specify the length by clicking two points.)** 2. **Width to use for rectangles <current value>** - **(enter a non-zero value or press enter to accept the current value or specify the width of the rectangle by clicking two points.)** 3. **Select first corner of rectangle** - **(pick the position of one corner of the rectangle, or enter x,y coordinates.)** 4. **Other corner of rectangle** - **(as you move the cursor, the program ghosts the four possible locations of the rectangle, relative to the first corner point.)**

Grips Editing

Rectangles can be edited directly through grips: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdR
Select the rectangle. Notice that it has a square grip at each vertex (1) and a triangular grip at the midpoint of each side (2)*. * If GRIPS = 1, midpoint grips do not display. Press and hold the Shift key to select multiple vertices. If **Dynamic Dimensions** (DYN) and **Polar Tracking** (POLAR) are active you can enter the distance(s) in the dynamic entry fields.

1. Drag a vertex grip (1) to stretch the a vertex. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cmd

2. Drag a midpoint grip (2) to stretch the side segment. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V

Figure 6.1862

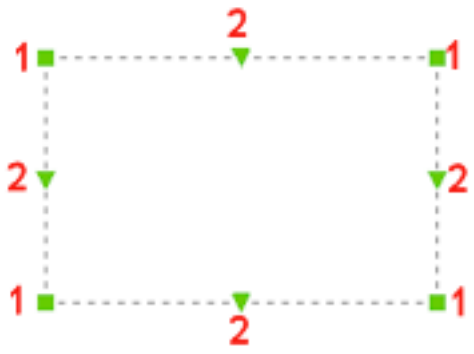


Figure 6.1863

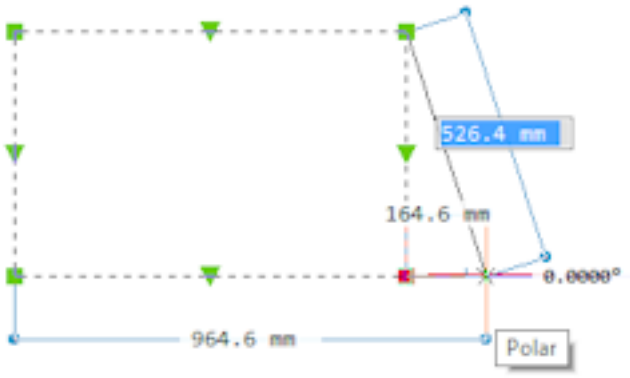
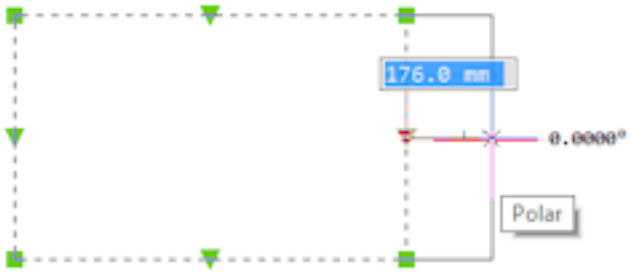


Figure 6.1864



Procedures

- Drawing rectangles (1.17.19)

Related Commands

PLine (6.24.27) - draws polyline segments and arcs. 3dPoly (6.3.10) - draws polylines with x,y,z coordinates. Polygon (6.24.43) - draws regular polygons out of polylines. Donut (6.10.35) - draws wide circles from polylines. PEdit (6.24.20) - edits polylines.

Figure 6.1865

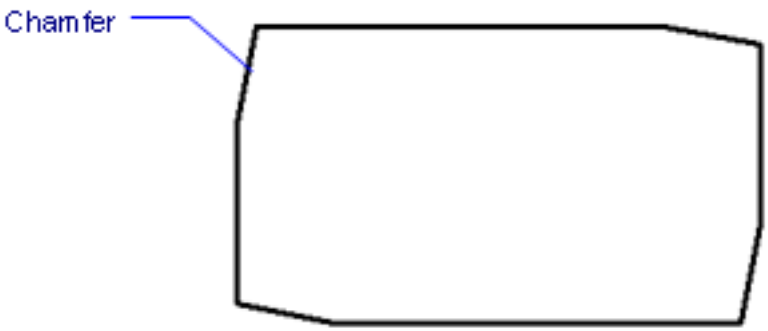


Figure 6.1866

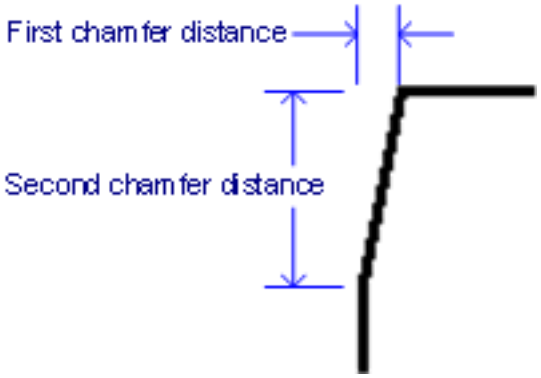


Figure 6.1867

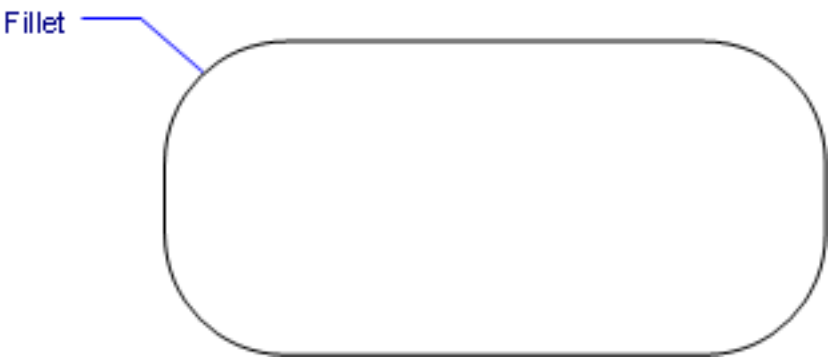


Figure 6.1868

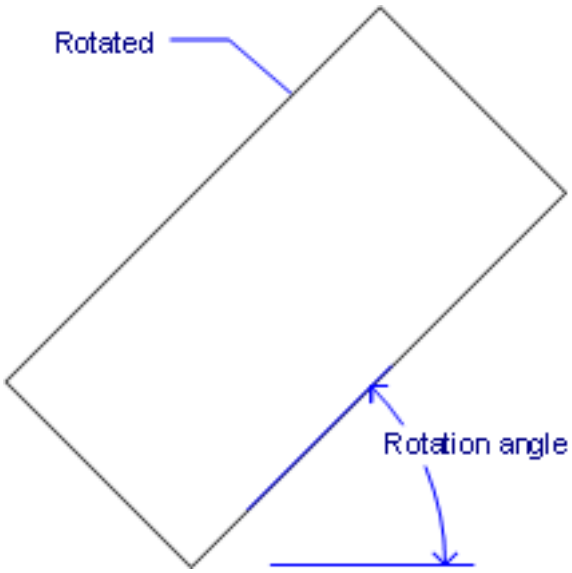


Figure 6.1869

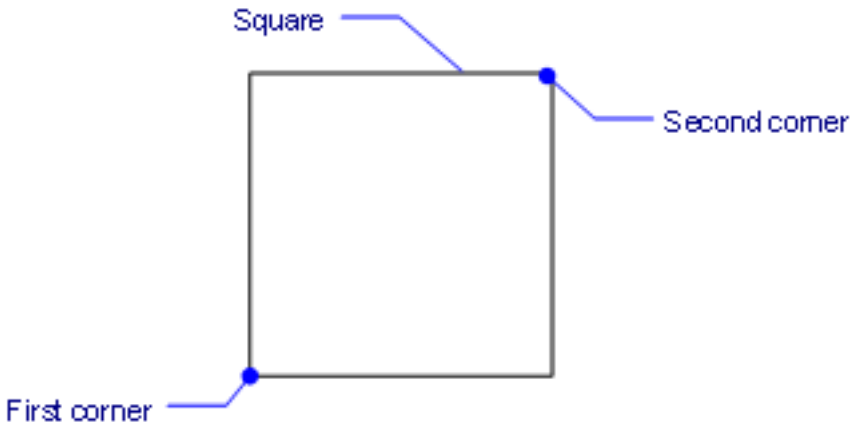


Figure 6.1870

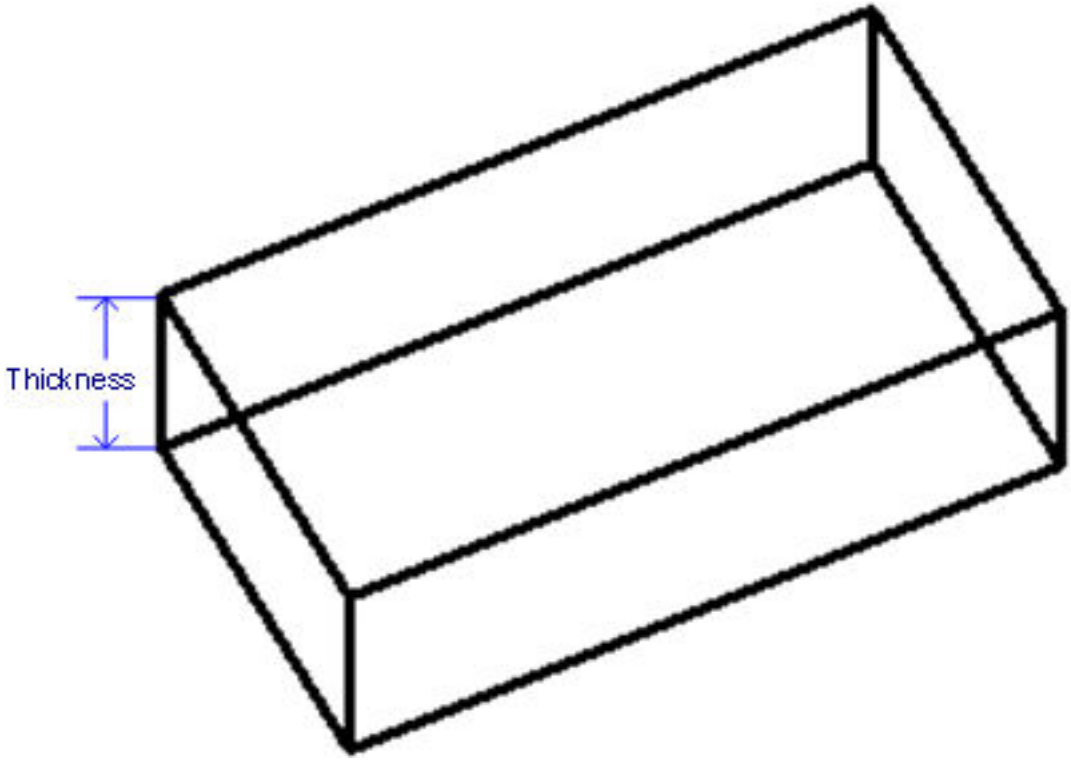


Figure 6.1871

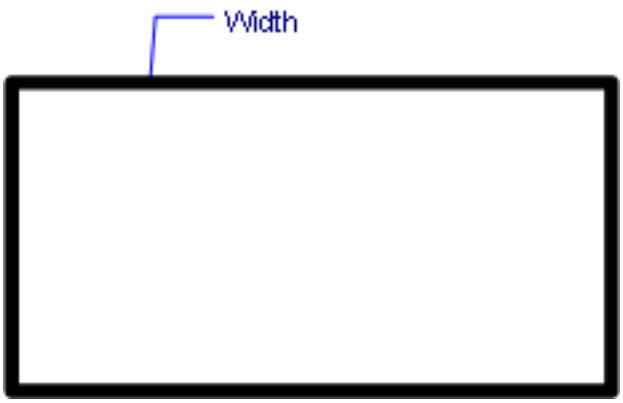


Figure 6.1872



6.26.7 Redefine

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Activates commands that were deactivated with the Undefine (6.30.4) command.
Note: You can temporarily reactivate a command by prefixing its name with a period, such as .line.

Accessing the Command

command bar: **redefine** : **Redefine Prompts you in the command bar:** Command to redefine: **(Enter name of an undefined command)**

Command Options

Option	Description
Command to redefine	Specifies the name of the command to redefine.

Related Commands

Commands (6.9.37) - lists the names of commands. Customize (6.9.58) - creates new commands through macros and LISP routines. Undefine (6.30.4) - removes commands from BricsCAD, temporarily.

Figure 6.1873



6.26.8 Redo

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Reverses the action of the last U (6.30.1) or Undo (6.30.5) command.

Accessing the Command

 command bar: **redo** transparent: **'redo** shortcut: **Ctrl+Y** menu bar: **Edit | Redo** ribbon: **Home | Edit | Redo** toolbar: **Standard | ===NO VALID URL FOR IMAGE=== br>** : **Redo Reports in the command bar:** Redo: **(commandName)** This command works only when the U (6.30.1) or Undo (6.30.5) commands were used at least once in the current editing session.When the U or Undo commands have not been used since the drawing was opened, BricsCAD responds: There is nothing to redo.

Command Options

No options.

Related Commands

U (6.30.1) - undoes the last command. Undo (6.30.5) - undoes one or more commands. Oops (6.23.7) - unerases the last erased object, including newly created blocks. Zoom (6.35.7) - utilizes the Previous option to show the previous view.

Figure 6.1874



6.26.9 Redraw

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Redraws the current viewport to clean it up from drawing detritus, such as blip or drag marks.

Accessing the Command

 command bar: **redraw** transparent: **'redraw** alias: **r** menu bar: **View | Redraw** ribbon: **View | View | Redraw** toolbar: **View | Redraw/Regen |**  : **Redraw Displays no prompt in the command bar.** This command redraws the current viewport; use the RedrawAll (6.26.10) command to redraw all viewports.

Command Options

No options.

Related Commands

Regen (6.26.15) - regenerates the current viewport. RegenAll (6.26.16) - regenerates all viewports. RedrawAll (6.26.10) - redraws all viewports to clean them up.

Figure 6.1875



6.26.10 RedrawAll

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Redraws all viewports to clean up drawing detritus, such as blip and drag marks.

Accessing the Command


command bar: **redrawall** transparent: **'redrawall** alias: **ra** ribbon: **View | View | Redraw | Redraw All** toolbar: **View | Redraw/Regen |**  : **RedrawAll** Displays no prompt in the command bar. This command redraws all viewports; use the Redraw (6.26.9) command to redraw the current viewport.

Command Options

No options.

Related Commands

Regen (6.26.15) - regenerates the current viewport. RegenAll (6.26.16) - regenerates all viewports. Redraw (6.26.9) - redraws the current viewport to clean it up.

Figure 6.1876



6.26.11 RefClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes the reference drawing editor, used to edit externally-referenced files in the current drawing session. **Note:** This command can be used only after the RedEdit (6.26.12) command has commenced.

Accessing the Command

 command bar: **refclose** menu: **Tools | External Reference Editing | Reclose and Save Tools | External Reference Editing | Refclose and Discard** ribbon: **Insert | External References | Refclose and Save** toolbar: **Ref Edit |**  or  : **RefClose Prompts you in the command bar:** Enter option [Save/Discard reference changes] <Save>: (Type S or D.) When you use this command on its own, BricsCAD complains: ** Command not allowed unless a reference is checked out with REFEDIT command **

Command Options

Option	Description
 Save	Saves the changes back to the block reference.
 Discard	Does not save the changes; the block is unchanged.

Figure 6.1877



6.26.12 RefEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits block references and externally-referenced drawings through a dialog box (short for ”reference editor”).

Accessing the Command

 command bar: **refedit** transparent: **'refedit** menu: **Tools | External Reference Editing | Refedit** ribbon: **Insert | External References | Refedit** toolbar: **Ref Edit |**  : **RefEdit Prompts you in the command bar:** Select reference: **(Choose a block or an xref)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
Choose options, and then click **OK**. **Prompts you in the command bar:** Use REFCLOSE or the Refedit toolbar to end reference editing session. Make changes to the reference, and then use the RefClose (6.26.11) command to save the changes. The XFADECTL (1.13.11) system variable sets the fading the rest of the drawing when editing an external reference or block. Values between 0 (no fading) and 90 are accepted. If the XEDIT system variable of the selected external reference is 0 or OFF a dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V

Figure 6.1878

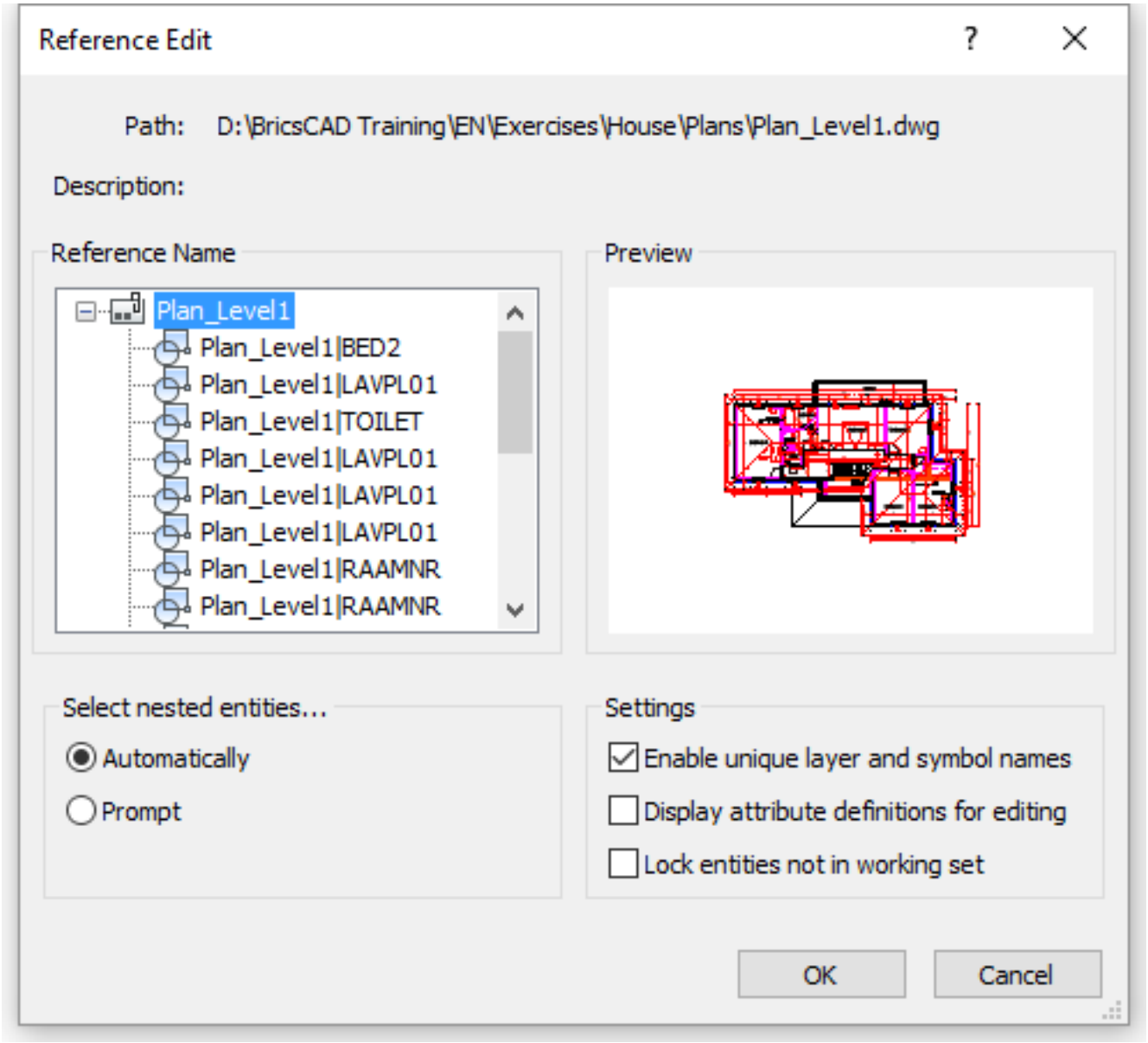
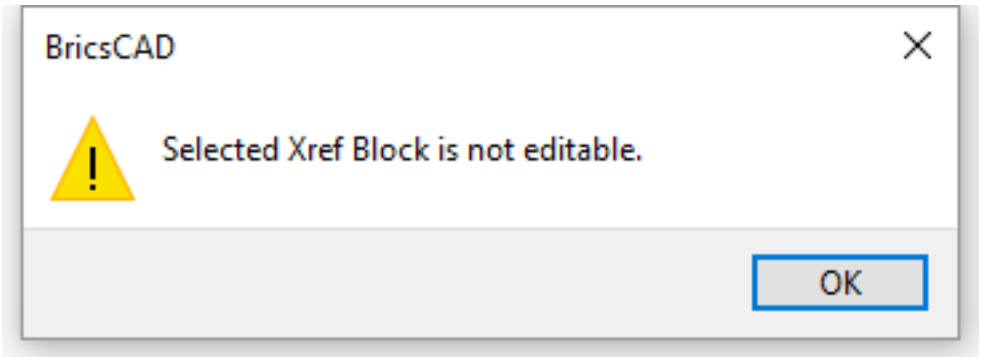


Figure 6.1879



Command Line Options

Option	Description
Select reference	Chooses the reference to edit.Pick a single block or externally-referenced drawing.Only one reference can be edited at a time.

Dialog Box Options

This dialog box appears automatically after you select a block or xref for editing:

Option	Description
Reference Name	Lists the name of the block and nested blocks, if any.
Select nested entities	Determines how nested blocks/xrefs are handled: • **Automatically** - nested entities are included automatically • **Prompt** - nested entities are selected manually after you click **OK** to close this dialog box; prompts you: 1. **Select nested objects** - **(select one or more nested entities.)** 2. **Select nested objects** - **(select more nested entities, or else press Enter to continue.)**
Settings	Determines how entities are treated during editing: **Enable unique layer and symbol names** - toggles the use of 0 in layer and other names: • **Yes** - prefixes names with **filename0** • **No** - does not prefix names **Display attribute definitions for editing** - toggles the inclusion of attributes: • **Yes** - attribute definitions can be edited; attribute values are invisible • **No** - attributes are unavailable for editing Modified attribute definitions do not affect existing insertions; they come into effect with insertions made from now on. **Lock objects not in working set** - toggles locking of entities not being edited: • **Yes** - locks entities outside the reference being edited • **No** - does not lock other entities Locked entities are seen but cannot be edited.

Procedures

- Editing blocks and external references (1.22.19)

Related Commands

Bedit (6.6.7) -Edits the blocks' entities in a temporary drawing session, using the local coordinate system of the block. -RefEdit (6.26.13) - edits references in-place through the command bar Refclose (6.26.11) - closes the reference editor Refset (6.26.14) - adds and removes entities from the reference being edited XOpen (6.34.6) - opens an externally-referenced drawing (Xref) for editing in a separate window AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttDisp (6.4.35) - toggles the display of attribute text. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. DdAtte (6.10.13) - edits the values of attributes through a dialog box. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces blocks to constituent entities and attributes to tags. Insert (6.17.13) - inserts blocks and prompts for attribute values through a dialog box. -Insert (6.17.14) - insert blocks and prompts for attribute values through the command bar. Properties (6.24.49) - edits block and attribute values and properties.

Figure 6.1880



6.26.13 -RefEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits block references and externally-referenced drawings, prompting for options in the command bar (short for ”reference editor”).

Accessing the Command

command bar: **-refedit** : **-RefEdit Prompts you in the command bar:** Select reference: **(Pick a block or an externally-reference drawing)** Select nesting level [Ok/Next] <Next>: **(Press O or N)** Enter entity selection method [All/Nested] <All>: **(Press A or N)** Display attribute definitions [Yes/No] <No>: **(Press Y or N)** Use REFCLOSE or the Refedit toolbar to end reference editing session. The XFADECTL (1.13.11) system variable sets the fading the rest of the drawing when editing an external reference or block. Values between 0 (no fading) and 90 are accepted.

Command Options

Option	Description
Select reference	Chooses the reference to edit.Pick a single block or externally-referenced drawing.Only one reference can be edited at a time.
Select nesting level	Choose the nested reference to edit: • **Ok** - edits the entire block. • **Next** - the program highlights the next nested level within the reference; the command repeats the ’Select nesting level’ prompt until you press**O**.
Enter entity selection method	Determines how nested entities are selected: • **All** - selects all nested entities automatically • **Nested** - prompts you to selects nested entities: 1. **Select nested objects** - **(select one or more nested entities.)** 2. **Select nested objects** - **(select more nested entities, or press Enter to continue.)**
Display attribute definitions	Toggles the inclusion of attributes: • **Yes** - attribute definitions can be edited; attribute values are invisible • **No** - attributes are unavailable for editing Modified attribute definitions do not affect existing insertions; they come into effect with insertions made from now on.

Related Commands

RefEdit (6.26.12) - edits references in-place through a dialog box. Refclose (6.26.11) - closes the reference editor Refset (6.26.14) - adds and removes entities from the reference being edited XOpen (6.34.6) - opens an externally-referenced drawing (Xref) for editing in a separate window AttDef (6.4.27) - defines attributes through a dialog box. -AttDef (6.4.34) - defines attributes through the command bar. AttDisp (6.4.35) - toggles the display of attribute text. AttRedef (6.4.30) - redefines a block and updates associated attributes. AttSync (6.4.31) - synchronizes attribute definitions in all block references of a specified block definition. BattMan (6.6.2) - manages the attributes of a block definition. EAttEdit (6.10.13) - edits the values of attributes through a dialog box. Block (6.6.11) and -Block (6.6.12) - create blocks from attribute definitions and entities. EAttEdit (6.13.1) - edits attribute values and most properties. Explode (6.13.13) - reduces blocks to constituent entities and attributes to tags. Insert (6.17.13) - inserts blocks

Figure 6.1881



6.26.14 RefSet

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Adds and removes entities from the reference (block or xref) being edited **Note:** This command can be used only after the RedEdit (6.26.12) command has commenced.

Accessing the Command

 command bar: **refset** menu:**Tools | External Reference Editing | Add to Refedit Tools | External Reference Editing | Remove from Refedit** ribbon: **Insert | External References | Add to Refedit** toolbar: **Ref Edit |**  : **RefSet Prompts you in the command bar:** Transfer objects between the RefEdit working set and host drawing... Enter an option [Add/Remove] <Add>: **(Press A or R)** When you use this command on its own, BricsCAD complains: ** Command not allowed unless a reference is checked out with REFEDIT command **

Command Options

Option	Description
 Add	Adds entities from the drawing to the reference being edited.
 Remove	Removes entities from the reference being edited; removed entities are placed in the drawing.Entities not in the reference are shown in gray.

Figure 6.1882



6.26.15 Regen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Regenerates the current viewport by reading and reloading the DWG file (short for ”regenerate”).

Accessing the Command

 command bar: **regen** transparent: **'redrawall** alias: **re menu** : **View | Regen** ribbon:**View | View | Regen** toolbar: **View | Redraw/Regen |**  : **Regen** Displays no prompt in the command bar.This command regenerates the current viewport. Use the RegenAll (6.26.16) command to regenerate all viewports. This command reads the DWG file and reconstructs the entire drawing on the screen.

Command Options

No options.

Related Commands

RegenAll (6.26.16) - regenerates all viewports. Redraw (6.26.9) - redraws the current viewport to clean it up. RedrawAll (6.26.10) - redraws all viewports to clean them up.

Figure 6.1883



6.26.16 RegenAll

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Regenerates all viewports by reading and reloading the DWG file (short for ”regenerate all”).

Accessing the Command

 command bar: **regenall** alias: **rea** ribbon:**View | View | Regen | Regen All** toolbar: **View | Redraw/Regen |**  : **RegenAll** Displays no prompt in the command bar.This command reads the DWG file and reconstructs the drawing in all viewports.

Command Options

No options.

Related Commands

Regen (6.26.15) - regenerates the current viewport. Redraw (6.26.9) - redraws the current viewport to clean it up. RedrawAll (6.26.10) - redraws all viewports to clean them up.

Figure 6.1884



6.26.17 RegenAuto

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Controls when BricsCAD regenerates the drawing automatically.

Accessing the Command

command bar: **regenauto** transparent: **'regenauto** : **RegenAuto Prompts you in the command bar:** REGENMODE is currently on: OFF/Toggle/<On>: **(Enter an option)**

Command Options

Option	Description
OFF	Automatic regeneration is turned off.If a regeneration is required, BricsCAD does not execute it; prompts you: **Regen queued.** The regeneration takes place when you enter the Regen (6.26.15) or RegenAll (6.26.16) commands.When the regeneration is required immediately, prompts you: **About to regen--okay? -(Answer yes or no.)**
Toggle	Toggles the state of the RegenMode system variable: • If on, turns it off. • If off, turns it on.
On	Turns on automatic regeneration.BricsCAD regenerates the drawing whenever it is required.

Figure 6.1885



6.26.18 Region

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Creates regions from closed 2D entities. **Note:**Regions are ACIS objects, and can be used with solid editing commands, such as MassProp (6.21.4) and Subtract (6.27.56).

Accessing the Command

 command bar: **region** alias: **reg** menu bar: **Draw | Region** ribbon: **Surface | Create | Region** toolbar: **Draw** |  : **Region Prompts you in the command bar:** Select entities: (Choose one or more closed entities.) Entities in set: 1 Select entities: (Choose more closed entities, or press Enter to continue.) 1 regions have been created. The source entities are erased if the DelObj variable is set to 2.

Command Options

Option	Description
Select entities	Specifies the entities to convert to regions.Choose one or more closed entities.Each entity is converted into a separate region.Closed entities include circles, closed polylines, and ellipses.

Related Commands

Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Intersect (6.17.19) - removes areas not in common between two or more regions. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Revolve (6.26.30) - revolves 2D entities into 3D solid models. Section (6.27.11) - creates 2D regions from 3D models. Subtract (6.27.56) - removes one region from a second. Union (6.30.6) - joins two or more regions into a single entity.

Figure 6.1886



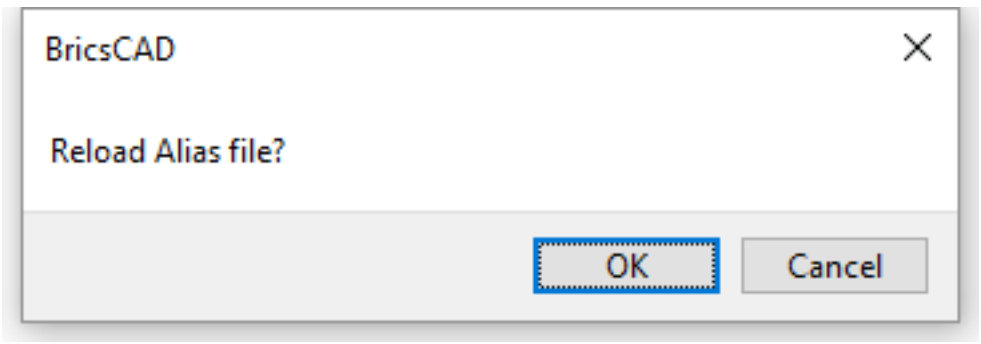
6.26.19 ReInit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Reloads the PGP file after it has been changed (short for ”reinitialize”). **Note:**The PGP file (short for ”program parameters”) file holds command alias definitions and shell commands.

Accessing the Command

command bar: **reinit** alias: **ri** : **ReInit** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
Click **OK** or **Cancel**. This command is used to load the PGP file after it has been edited externally.The PGP file defines command aliases. See the ”Customizing BricsCAD” ebook for details on aliases.

Figure 6.1887



Command Options

Option	Description
OK	Reloads the PGP file into BricsCAD.
Cancel	Does not reload the file.

Related Command

Customize (6.9.58) - creates and edits aliases.

Figure 6.1888



6.26.20 Rename

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Renames named entities through the Drawing Explorer.

Accessing the Command

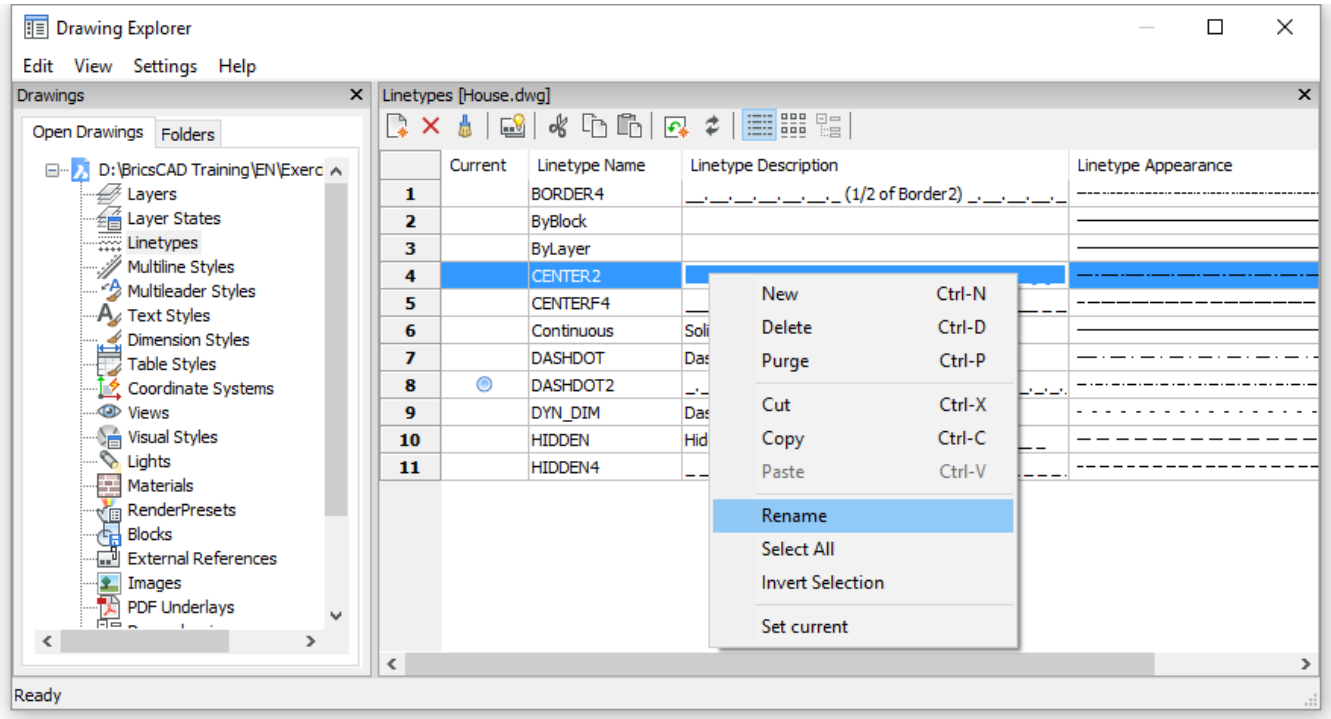
command bar: **rename** aliases: **ren** **ddrename** : **Rename** Displays the Drawing Explorer: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/RenameDialog.png})

1. Choose a section, such as Layers or Dimension Styles.
2. To rename an item, click twice on its name (do not double-click), then enter the new name. Alternatively, right-click the item, then choose Rename in the shortcut menu.

You can change the names of most layers, layer states, most linetypes, UCSs, views, blocks, most text styles, and most dimension styles.To change names at the command line, use the -Rename (6.26.21) command. You cannot change the names of attached files, such as images and external references, nor thefollowing names:

- Layer ”0”
- Linetype ”ByBlock”, ”ByLayer”, or ”Continuous”
- Text style ”Standard”
- Dimension style ”Standard”
- Multiline style”Standard”
- Visual style ”2DWireframe”
- Material ”Global”
- Page layouts that have * as a prefix and suffix, such as *Model*
- View detail style ”Metric50”
- View section style ”Metric 50”

Figure 6.1889



Command Options

No options.

Related Command

-Rename (6.26.21) - renames commands through the command bar.

6.26.21 -Rename

   Renames named entities in the command bar.

Accessing the Command

command bar: **-rename** alias: **-ren** : **-Rename Prompts you in the command bar:** Rename [Block/Dimension style/LAyer/LineType/text Style/Table style/ Ucs/ VIew/ ViewPort]: **(Enter an option)** To rename items such as mleader and view detail styles, use theRename (6.26.20) command.

Command Options

Option	Description
Block	Renames blocks; prompts you: 1. **Block to be renamed** - **(enter the name of an existing block.)** 2. **New name for the Block name** - **(enter the new name.)** If you enter the name of an existing block, then the program prompts you: **A Block with that name already exists. New name for the Block name** - **(enter a different name.)** See theBlock (6.6.11) command.
Dimension style	Renames dimension styles; prompts you: 1. **Dimension style to be renamed** - **(enter the name of an existing dimension style.)** 2. **New name for the Dimension style name** - **(enter the new name.)** If you enter the name of an existing dimension style, then the program prompts you: **A Dimension style with that name already exists. New name for the Dimension style name** - **(enter a different name.)** See DimStyle (6.11.20) command.
LAyer	Renames layers; prompts you: 1. **Layer to be renamed** - **(enter the name of an existing layer.)** 2. **New name for the Layer name** - **(enter the new name.)** If you enter the name of an existing layer, then the program prompts you: **A Layer with that name already exists. New name for the Layer name** - **(enter a different name.)** If you attempt to rename layer 0, then the program prompts you: **That name is reserved by BricsCAD, and cannot be renamed.** See Layer (6.20.2) command.

LineType	Renames linetypes; prompts you: 1. **Linetype to be renamed** - **(enter the name of an existing linetype.)** 2. **New name for the Linetype name** - **(enter the new name.)** If you enter the name of an existing linetype, then the program prompts you: **A Linetype with that name already exists. New name for the Linetype name** - **(enter a different name.)** If you attempt to rename linetypes ByLayer, ByBlock, or Continuous, then the program prompts you: **That name is reserved by BricsCAD, and cannot be renamed.** See Linetype (6.20.33) command.
text Style	Renames text styles; prompts you: 1. **Text style to be renamed** - **(enter the name of an existing style.)** 2. **New name for the Text style name** - **(enter the new name.)** If you enter the name of an existing style, then the program prompts you: **A Text style with that name already exists. New name for the Text style name** - **(enter a different name.)** See Style (6.27.53) command.
Tablestyle	Renames table styles; prompts you: 1. **Table style to be renamed** - **(enter the name of an existing style.)** 2. **New name for the Table style name** - **(enter the new name.)** If you enter the name of an existing style, then the program prompts you: **A Table style with that name already exists. New name for the Table style name** - **(enter a different name.)**
Ucs	Renames user-defined coordinate systems; prompts you: 1. **UCS to be renamed** - **(enter the name of an existing UCS.)** 2. **New name for the UCS name** - **(enter the new name.)** If you enter the name of an existing UCS, then the program prompts you: **A UCS with that name already exists. New name for the UCS name** - **(enter a different name.)** See UCS (6.30.2) command.
VIew	Renames views; prompts you: 1. **View to be renamed** - **(enter the name of an existing view.)** 2. **New name for the View name** - **(enter the new name.)** If you enter the name of an existing view, then the program prompts you: **A View with that name already exists. New name for the View name** - **(enter a different name.)** See View (6.31.1) command.

VPort	Renames viewport configurations; prompts you: 1. **Viewport to be renamed** - **(enter the name of an existing viewport configuration.)** 2. **New name for the Viewport name** - **(enter the new name.)** If you enter the name of an existing viewport, then the program prompts you: **A Viewport with that name already exists. New name for the Viewport name** - **(enter a different name.)** See VPorts (6.31.20) command.

Related Command

Rename (6.26.20) - renames commands through the Drawing Explorer.

Figure 6.1890



6.26.22 Render

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Generates a photo-realistic rendering using materials and lights, through a dialog box. **Note:** When a drawing contains only 2D entities, the result is a blank rendering.

Accessing the Command

 command bar: **render** alias: **rr** menu bar: **View|Rendering - Render** ribbon:**View | Render Modes | Render** toolbar: **Render|**  : **Render** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/C
Reports the current render mode: Current render preset: 'Medium" When the rendering appears in a window, the red square shows which part is currently being rendered: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/C
window.png})

Figure 6.1891

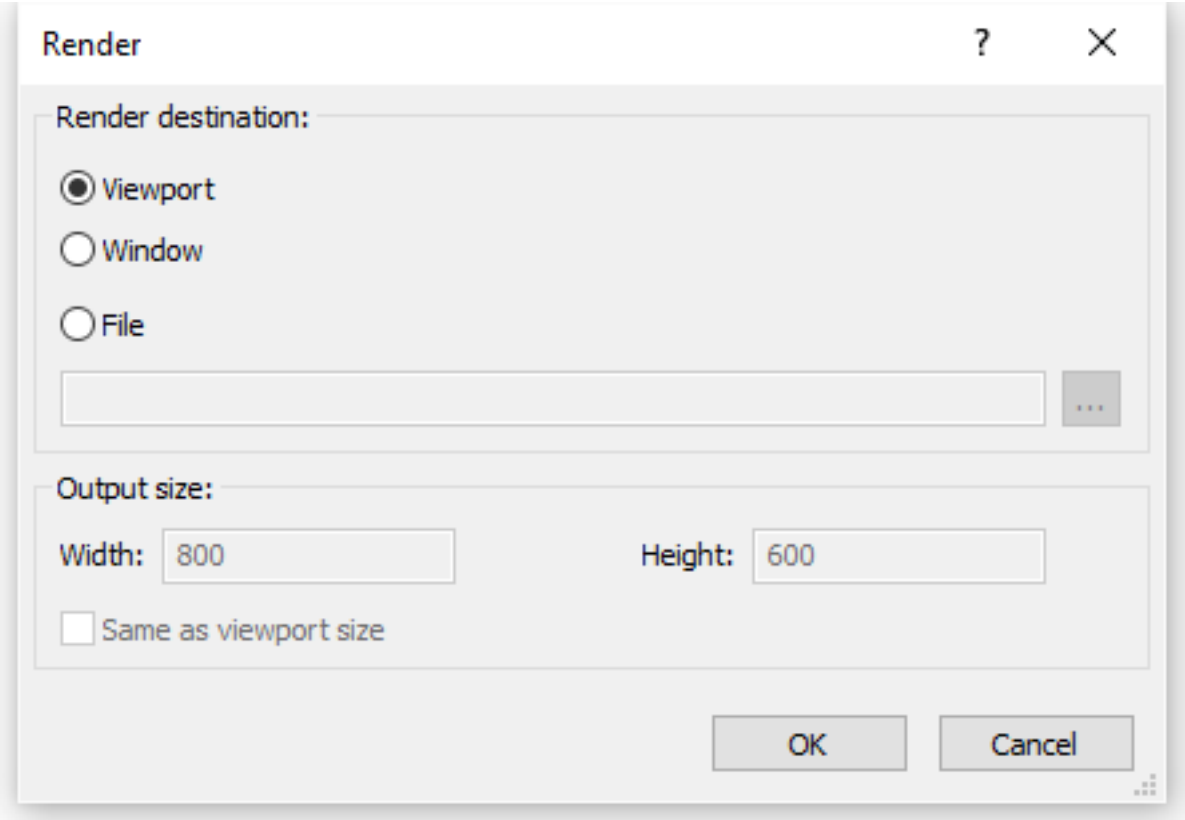
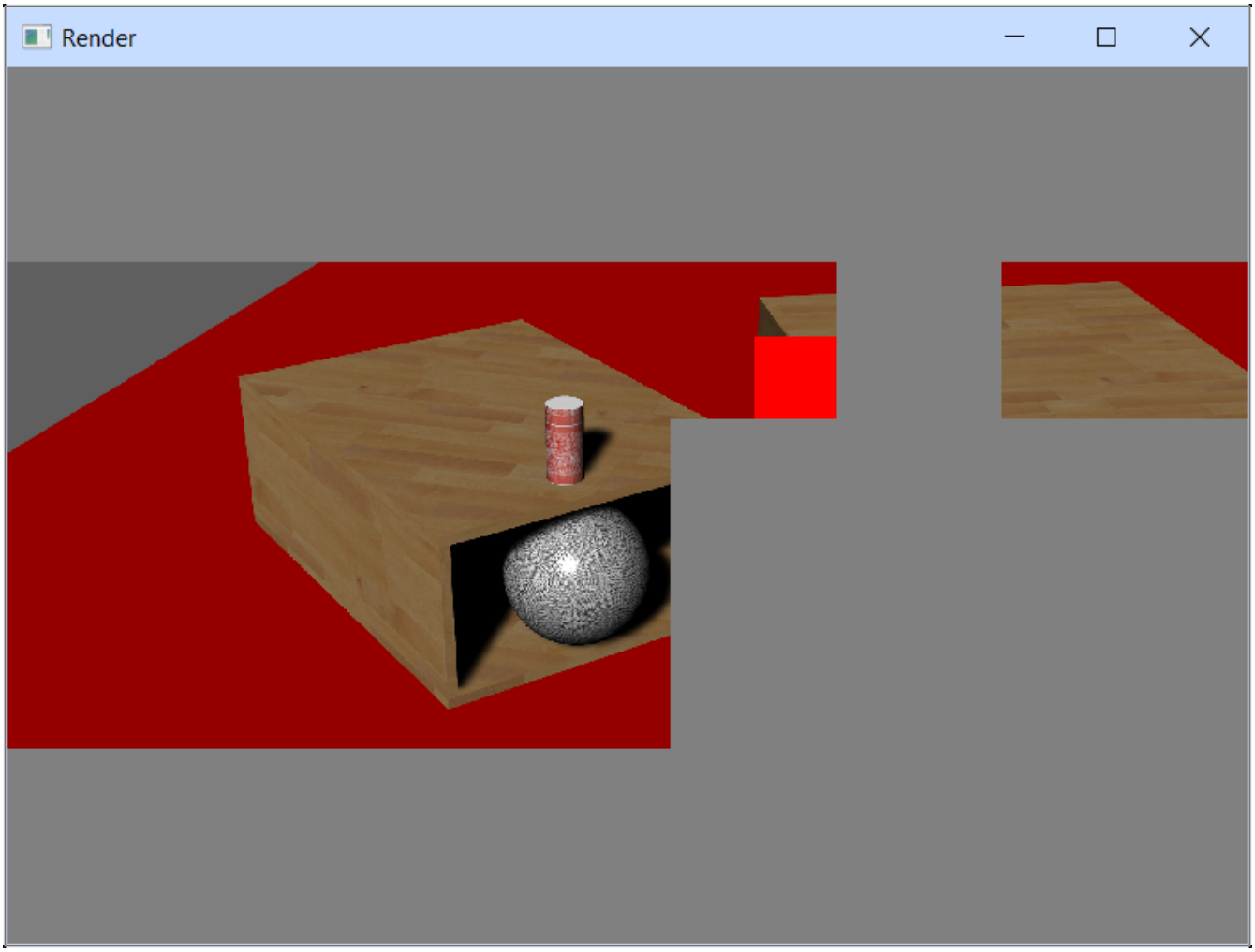


Figure 6.1892



Dialog Box Options

Option	Description
Render destination	Specifies where the rendering will appear: • **Viewport** - the rendering is generated in the current viewport • **Window** - the program opens a new windows, and then generates the rendering in it • **File** - the rendering is saved to a file: 1. Click the **Browse** button to open the Render output file dialog. 2. (Optional) Select an output folder in the **Save In** droplist. 3. Type a name in the **File name** field. 4. Select an image file type from the **Save As Type** droplist: **BMP** - Windows bitmap format; largest file size as it is uncompressed **JPEG** - highly compress raster format; smallest file size but suffers from artifacts **PNG** - compressed raster format; medium file size with no artifact problem 5. Click **Save**.
Output Size	Specifies the size of the rendering, in pixels. Larger sizes provide more detail but take longer to render: • **Width** - specifies thewidth of the rendering, in pixels • **Height** - specifies the height of the rendering, in pixels • **Same as viewport size**- matches the ouput size or the size of the output window to the current viewport size.
OK	Starts calculating the rendering according to the settings established by the RenderPresets (6.26.24) command. The ANTIALIASRENDER (1.13.11)variable controls the amount of anti-aliasing (edge smoothing) applied during the rendering generation. The default value is 1, maximum value is 5.Note: high anti-alias values incur a high calculation cost.

Procedures

- To render a viewport (1.26.2)
- To save a rendering to a file (1.26.2)

Related Commands

AniPath (6.4.8) - records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file. -Render (6.26.23) - generates a photorealistic rendering using materials and lights through the command bar. RenderPresets (6.26.24) - allows to create and edit render presets, and to set the current render preset through a dialog box. Background (6.6.1) - sets the model space background for the current viewport and saved views. Distantlight (6.10.32) - creates a distant light. Hide (6.16.10) - removes hidden lines from 3D entities. Light (6.20.28) - creates a new light in the drawing. Lightlist (6.20.29) - displays the Drawing Explorer - Lights dialog window. Materials (6.21.11) - creates materials and edits their properties through the Drawing Explorer. Pointlight (6.24.42) - creates a point light. RedSdkInfo (6.26.40) - displays rendering related hardware and driver specifications in the command bar. Shade (6.27.27) - generates shaded images of 3D drawings. ShadeMode (6.27.29) - sets the shading style for 3D entities. Spotlight (6.27.42) - creates a spot light. Sunproperties (6.27.57) - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog. VisualStyles (6.31.13) - creates and controls visual style definitions in the Drawing Explorer. Weblight (6.33.6) - creates a web light.

Figure 6.1893



6.26.23 -Render

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Generates a photo-realistic rendering using materials and lights, through the command bar.

Accessing the Command

command bar: **-render** : **-Render Prompts you in the command bar:** Specify render preset [Draft/Low/Medium/High/Presentat <Medium>: **(Choose a render preset, or press Enter to accept the current preset)** Specify render destination [Viewport/File/Render window] <Viewport>: **(Choose an option, or press Enter to accept the default.)**

Command Options

Option	Description
Specify render preset	Sets the settings to be used for the rendering. See the RenderPresets (6.26.24) command.Do one of the following: • Select one of the hard-coded presets (Draft, Low, High, Presentation) • Choose **Other** , to choose a user-defined preset; prompts you: **Enter custom render preset name or [?]**- **(type the name of a preset or type ? to list the available custom presets.)**
Specify render destination	Determines where the rendering ends up; choose one: • **Viewport** - the rendering is displayed in the current viewport • **File** - exports the rendering to an file; prompts you: **Specify output file name** - **(type a file name.)** • **Render view** - Displays the rendering in a pop-up window; prompts you: **Enter output width or [Viewport size] <800>** - **(type a value or enterVto set the size of the pop-up window equal to the current viewport size.) Enter output height <600>** - **(type a value.) Save also to file?[Yes/No] <No>** - if Yes , you are prompted: **Specify output file name** - **(type a file name.)**

The ANTIALIASRENDER (1.13.11) system variable controls the amount of anti-aliasing (edge smoothing) applied during the rendering generation. The default value is 1, maximum value is 5.

Related Commands

AniPath (6.4.8) - records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file. Render (6.26.22) - generates a photorealistic rendering using materials and lights through a dialog box. RenderPresets (6.26.24) - allows to create and edit render presets, and to set the current render preset through a dialog box. Background (6.6.1) - sets the model space background for the current viewport and saved views. Distantlight (6.10.32) - creates a distant light. Hide (6.16.10) - removes hidden lines from 3D entities. Light (6.20.28) - creates a new light in the drawing. Lightlist (6.20.29) - displays the Drawing Explorer - Lights dialog window. Materials (6.21.11) - creates materials and edits their properties through the Drawing Explorer. Pointlight (6.24.42) - creates a point light. RedSdkInfo (6.26.40) - displays rendering related hardware and driver specifications in the command bar. Shade (6.27.27) - generates shaded images of 3D drawings. ShadeMode (6.27.29) - sets the shading style for 3D entities. Spotlight (6.27.42) - creates a spot light. Sunproperties (6.27.57) - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog. VisualStyles (6.31.13) - creates and controls visual style definitions in the Drawing Explorer. Weblight (6.33.6) - creates a web light.

Figure 6.1894



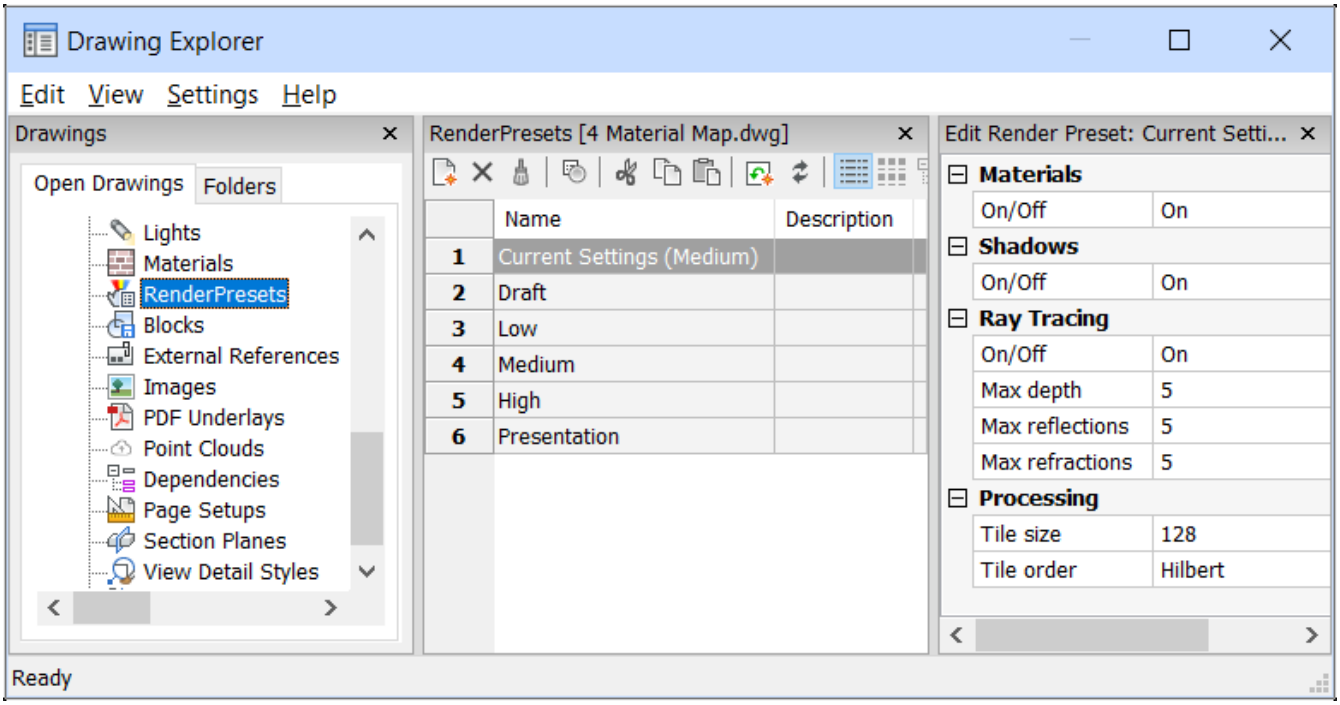
6.26.24 RenderPresets

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and edits properties for renderings, through a dialog box.

Accessing the Command

 command bar: **renderpresets** alias: **roptions** menu bar: **Tools | Drawing Explorer | Render Presets** ribbon:**Home | Settings | Drawing Explorer | Render Presets View | Render Modes | Render Presets** : **RenderPresets**
Displays the Render Presets section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom dlg.png})

Figure 6.1895



Command Options

Toolbar Option	Description
New	Creates new render preset definitions; follows these steps: 1. Click**New**.The New Render Preset dialog box displays. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so 2. Type a name in the **Name** field. 3. Select an existing render preset to use as a template for the new one. 4. Click **Create**.The Drawing Explorer - Render Presets dialog box reopens. 5. (Optional) Type a description in the **Description** field.
Delete	Deletes the currently selected user defined render preset(s).You cannot delete predefined render presets: Draft, Low, Medium, High, and Presentation.
Set Current	Sets the selected render preset current. The current render preset is used by the Render (6.26.22) command or when printing using the shade plot type Rendered as defined in **Shade Plot** setting in the Page Setup (6.24.1) and the Print (6.25.3)commands.The name of the current render preset displays between brackets: e.g. Current Settings (Medium). You can override the properties of a preset in the Edit Render pane. When overrides exist, an asterisk displays in front of the current render preset name: e.g.Current Settings (*Medium)

Edit Render Preset Options	Description
Materials	Toggles the use of materials in the rendering: • **On** - uses material definitions for rendering, at the cost of a slower rendering • **Off** - does not use material, at the cost of a less realistic rendering Materials are applied to layers and to entities. See Materials (6.21.11) commands.
Shadows	Toggles the use of shadows in the rendering: • **On** - uses shadows for rendering, at the cost of a slower rendering • **Off** - does not use shadows, at the cost of a less realistic rendering Shadows are cast by entities in the drawing onto other entities.
Ray Tracing	Ray tracing tracks each ray of light from each light source as it travels through the scene and bounces off (or travels through) entities. **On/Off** toggles the use of ray tracing in the rendering: • **On** - uses ray tracing, at the cost of a slower rendering • **Off** - does not use ray tracing, at the cost of a less realistic rendering **Max. depth** - specifies the maximum total number of times light bounces and refracts **Max reflections**- specifies the maximum times that light beams bounce of of entities **Max refractions** - specifies the maximum times that light beams refract through transparent entities

Processing	Specifies how the rendered image is processed; applies to renderings sent to a separate window. In the figure below, the bright red square is a ”tile.” (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/R/ren tile-eg.png}) **Tile size** - sets the size of the tile, representing the area of the rendered image being processed. **Tile order** - defines the order in which the tiles are generated: • **Hilbert** - renders with a Hilbert curve, a continuous fractal space-filling; see http://en.wikipedia.org/wiki/Hilbert_curve for a description. • **Spiral** - renders in a counterclockwise spiral from the center • **Left to right** - renders vertical columns, starting from the bottom left corner • **Right to left** - renders vertical columns, starting from the bottom right corner • **Top to bottom** - renders horizontal rows, starting at the top left corner • **Bottom to top**- renders i horizontal rows, starting at the bottom left corner

Related Commands

Render (6.26.22) - generates a photorealistic rendering using materials and lights through a dialog box. -Render (6.26.23) - generates a photorealistic rendering using materials and lights through the command bar. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.1896

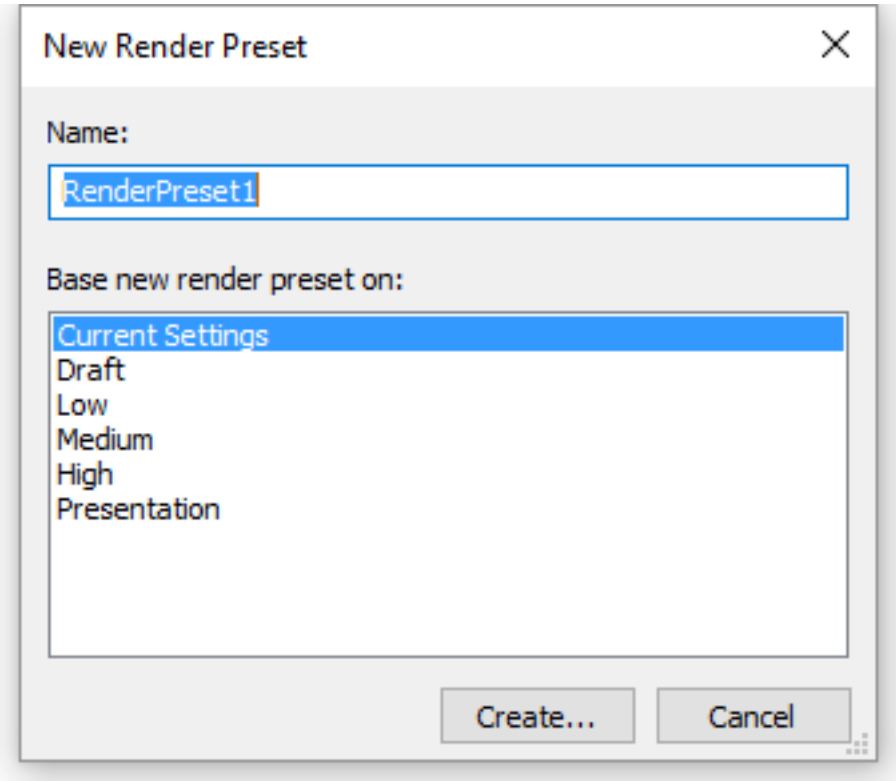


Figure 6.1897

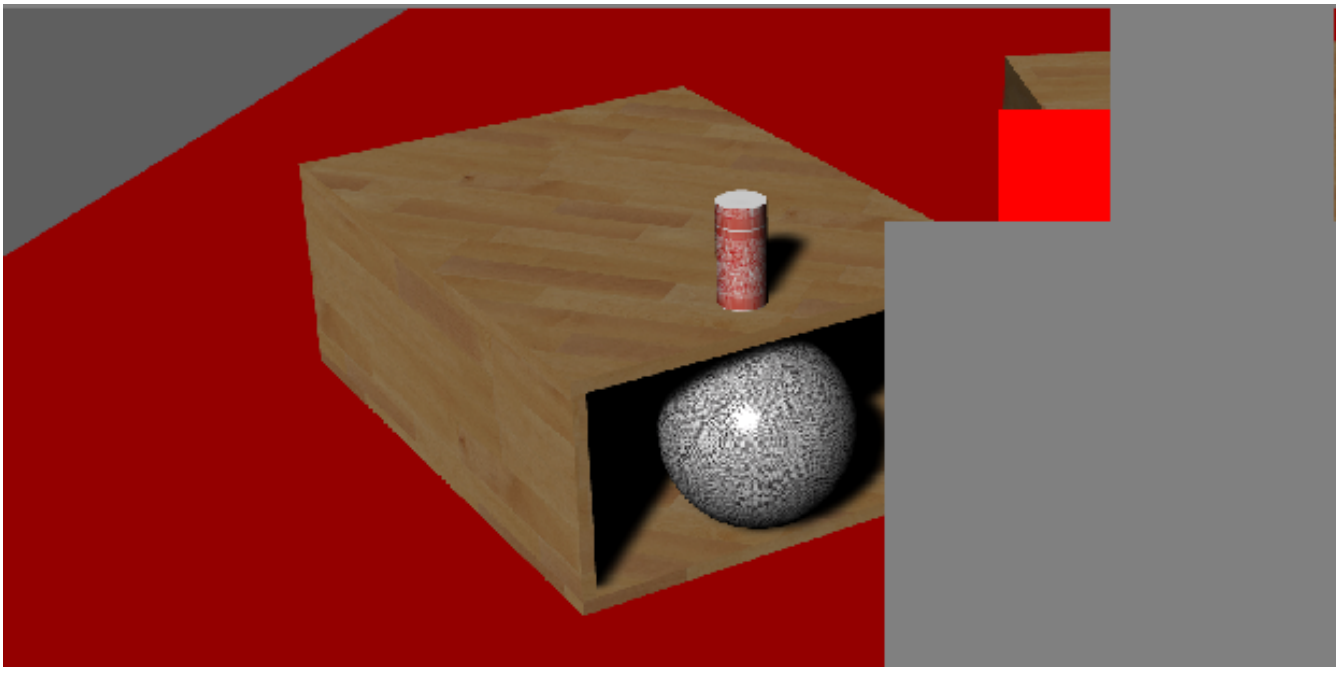


Figure 6.1898



6.26.25 ResetAssocViews

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Removes the associativity from 2D drawings generated from 3D models. **Note:**The ViewBase (6.31.3), ViewSection (6.31.10), and BimSection (6.7.27) commands generate 2D drawings associated with the source 3D model so that drawings are updated automatically as the 3D model is modified (when the GENERATEASSOCVIEWS variable is on). The ResetAssocViews command removes associativity from between 2D drawings and 3D models, including associative dimensions, which is useful for archival purposes.

Accessing the Command

command bar: **resetassocview** transparent: **resetassocview** : **ResetAssocView Prompts you in the command bar:** Select several entities/subentities: **(Select one or more entities)** Entities in set: 42 Select several entities/subentities:**(Select more entities, or else press Enter to finish the command)** Associative attributes were removed from selected entities.

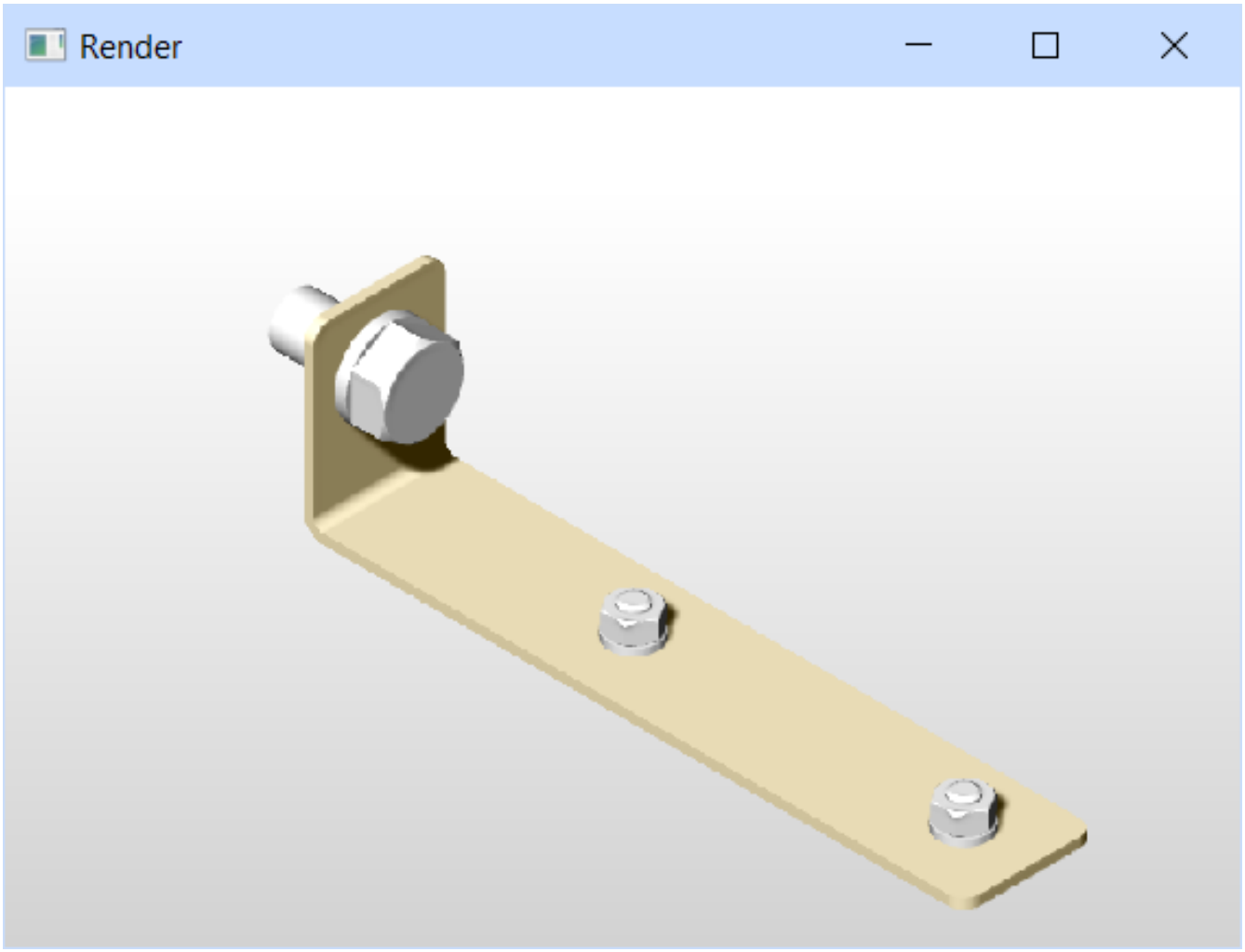
Command Options

Option	Description
Select several entities/subentities	Selects the entities whose associative dimensions should be removed: • Select one or more entities • Enter **all** to select all entities in the drawing • Press **Enter** to finish the command

Figure 6.1899



Figure 6.1900



6.26.26 RenderWinClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Closes the Render window, after it displays a rendering. **Note:** To open the Render window, select the Window option of the Render (6.26.22)command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/R/renderwin

Accessing the Command

command bar: **renderwinclose** transparent: '**renderwinclose** : **RenderWinClose** Does not prompt you; closes the Render window.

Command Options

No options.

Related Command

Render (6.26.22) - generates a photorealistic rendering using materials and lights through a dialog box.

Figure 6.1901



6.26.27 ResetBlock

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Resets dynamic blocks to their default values.

Accessing the Command

command bar: **resetblock** : **ResetBlock Prompts you in the command bar:** Select entities: **(Select one or more dynamic blocks)** Entities in set: 1 Select entities: **(Select more dynamic blocks, or else press Enter)**

Command Options

Option	Description
Select entities	Selects the dynamic blocks whose properties are to be reset: • Choose one or more dynamic blocks • Enter **all** to select all of them in the drawing • Press **Enter** to end the command

Related Commands

-Insert (6.17.14) - inserts blocks, by prompting in the command bar and prompts for attribute values through the command bar or a dialog box. Insert (6.17.13) - inserts blocks, through a dialog box and prompts for attribute values through the command bar or a dialog box.

Figure 6.1902



6.26.28 Resume

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Resumes paused scripts. **Note:** Start a script with the Script (6.27.9) command.

Accessing the Command

command bar: **resume** transparent: **'resume** : **Resume** Displays no prompts in the command bar.This command resumes scripts after they have been stopped with Esc.

Command Options

No options.

Related Commands

RecScript (6.26.5) - records scripts to SCR files. StopScript (6.27.48) - stops recording the script, and saves it to an SCR file. Script (6.27.9) - loads and runs a SCR script file. RScript (6.26.36) - reruns the script. Delay (6.10.23) - pauses the execution of the script for the specified number of milliseconds.

Figure 6.1903



6.26.29 RevCloud

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws revision clouds commonly used for redlining drawings (short for "revision cloud").

Accessing the Command

 command bar: **revcloud** menu bar: **Draw | Revision Cloud** ribbon:**Draw | Markup | Revision Cloud** toolbar:

Draw | Polygon |  : **RevCloud Prompts you in the command bar:** Specify first corner point or [Arc length/Entity/Rectangular/P <Entity>: **(Pick a point, or enter an option)** Guide crosshairs along cloud path... **(Move the cursor to create the free form cloud shape)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image294.jpg}) Revision cloud finished. **(When the cursor reaches the starting point, the cloud closes automatically)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image296.jpg}) Reverse direction [Yes/No <No>: **(Press Y or N)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image2

Figure 6.1904

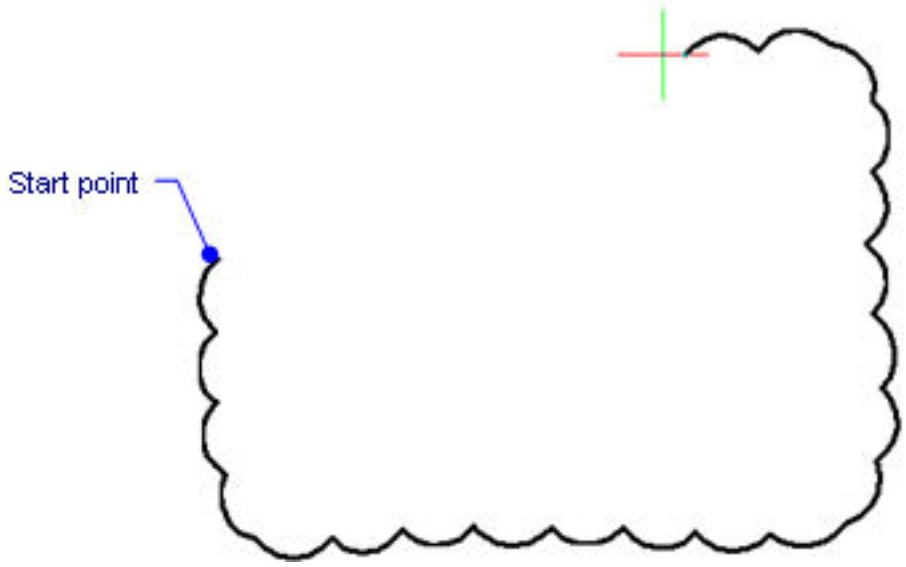


Figure 6.1905

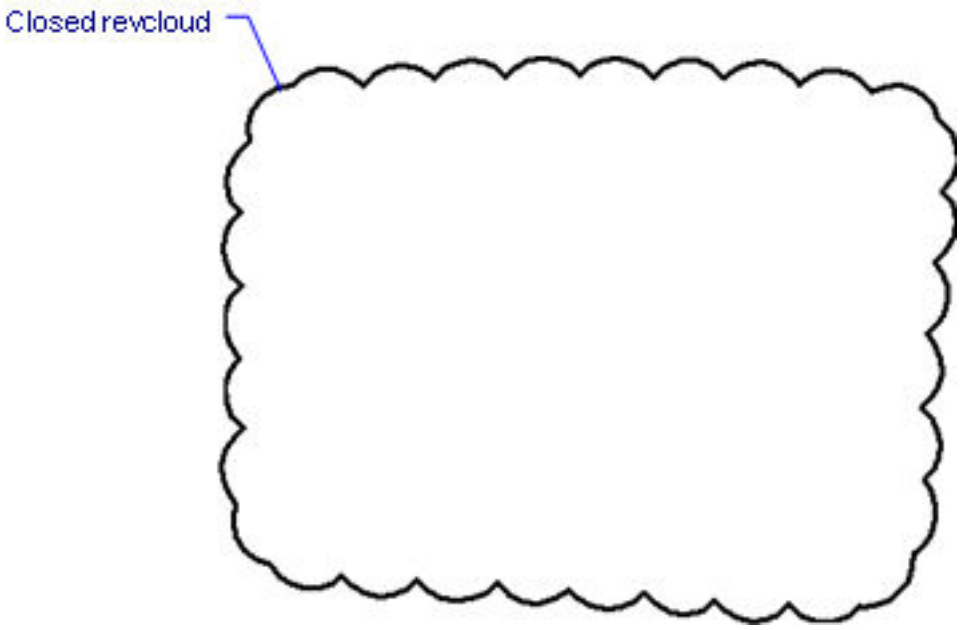
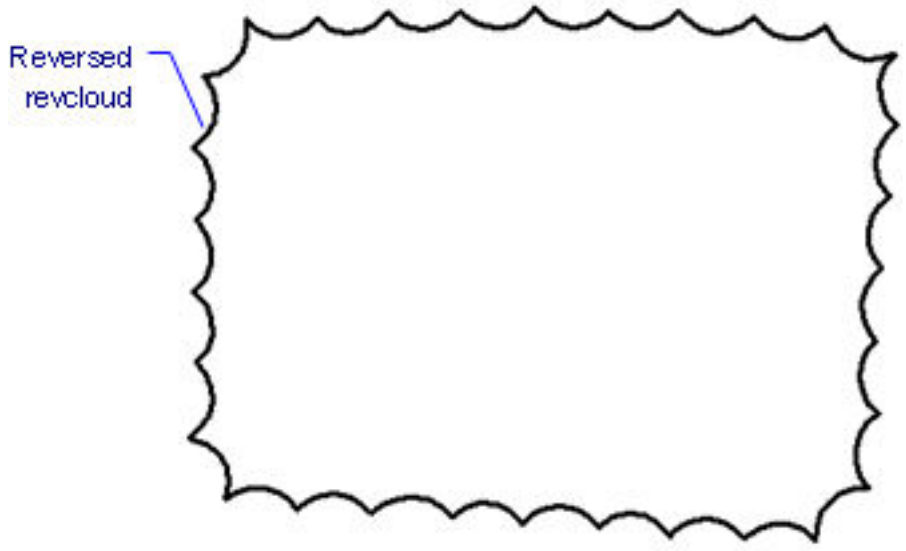


Figure 6.1906



Command Options

Option	Description
Specify start point	Click a point to start drawing; when the cursor returns to the start point, the revision cloud closes automatically.The revision cloud is drawn as a closed polyline composed of arcs. For red revision clouds, set the layer color to red.
Arc length	Specifies the shortest length for the arcs that form the cloud; prompts you: **Specify minimum length of arc** - **(enter a value.)**
Entity	Converts entities into a revision cloud; prompts you: **Select entity** - **(choose the entity to be converted into a revision cloud.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Entities can be open or closed; the original entity is erased.
Rectangular	Draws a cloud shape that is rectangular; prompts you: 1. **Specify first corner point or [Arc length/Entity/Rectangular/ Polygonal/Freehand/Style] <Entity>** - **(pick a point at one corner of the rectangle.)** 2. **Specify opposite corner point** - **(pick the opposite corner for the rectangle shape.)**
Polygonal	Draws a cloud shape defined by several pick points; prompts you: 1. **Specify first point or [Arc length/Entity/Rectangular/Polygonal/ Freehand/Style] <Entity>** - **(pick a point to start the polygon.)** 2. **Specify next point**- **(pick the next point of the polygon.)** 3. **Specify next point or [Undo]**- **(pick the next point of the polygon, or else enter U to undo the last point, or press Enter to finish the cloud shape.)**
Freehand	Draws a cloud shape for as long as you keep the mouse button pressed down; when you let go, the program automatically closes the cloud shape. 1. **Specify start point or [Arc length/Entity/Rectangular/Polygonal/ Freehand/Style] <Entity>** - **(pick a point to start the cloud.)** 2. **Guide crosshairs along cloud path...** - **(move mouse to create the shape.)** 3. **Revision cloud finished.**

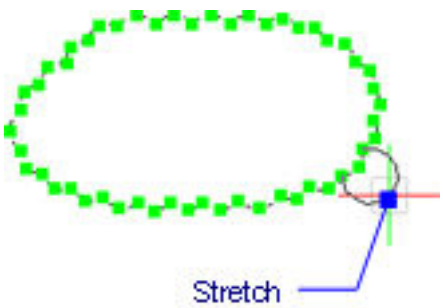
Style	Toggles between normal and calligraphic drawing style; prompts you: **Select arc style [Normal/Calligraphy] <Normal>** - **(type N or C)**: • **Normal** - polyline making up the cloud has uniform width • **Calligraphy** - polyline has varying width (tapered polyarcs) to simulate a calligraphic look (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Grips Editing Revision clouds can be edited directly through grips:

1. Select the revcloud. Notice that it has many grips.
2. Drag a grip to stretch the cloud.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image478.jpg})

Figure 6.1907



Procedures

- Drawing revision clouds (1.17.24)

Related Commands

PEdit (6.24.20) - edits polylines. Sketch (6.27.34) - draws freehand sketches.

Figure 6.1908

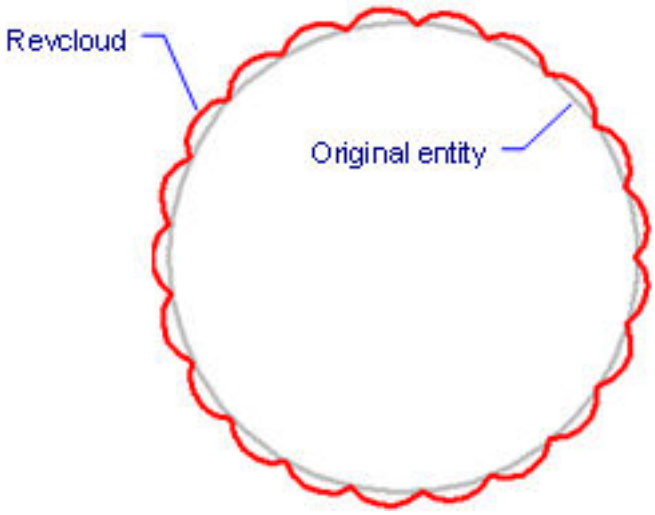


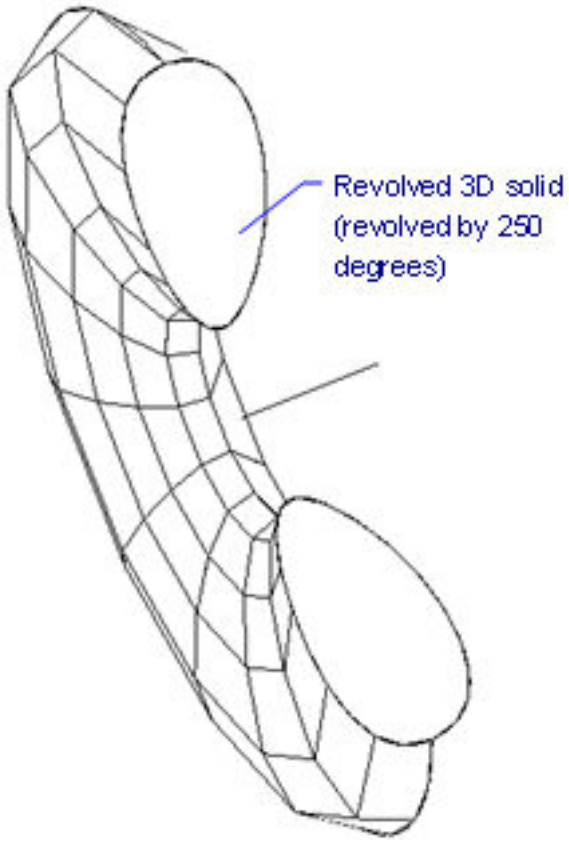
Figure 6.1909



Figure 6.1910



Figure 6.1911



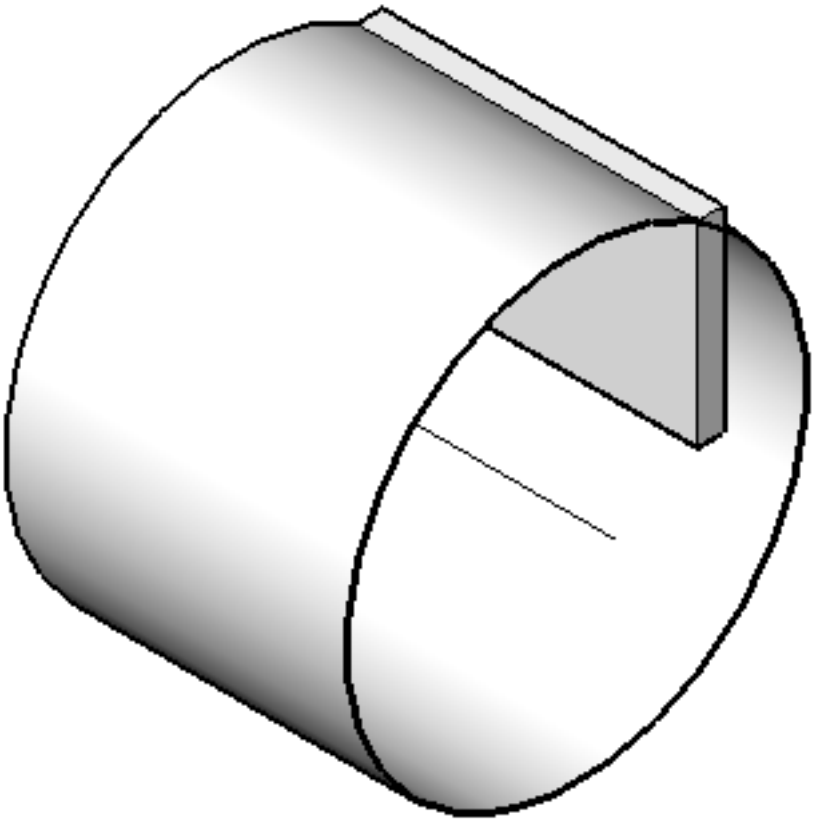
6.26.30 Revolve

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Revolves open or closed 2D entities, solid edges, and faces into 3D solids or 3D surfaces. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image426.jpg})

Accessing the Command

command bar: **revolve** alias: **rev** menu bar: **Model | 3D Solids | Revolve** ribbon:**Model | Solids | Revolve** toolbar: **3D Solids** | : **Revolve Prompts you in the command bar:** Select entities/subentities to revolve or set [MOde]: **(Choose one or more closed 2D entities, or press M)** Entities in set: 1 Select entities/subentities to revolve or set [MOde]: **(Choose more closed 2D entities, or else press** Enter **to continue.)** Specify start point for axis of revolution or define axis by [Object/Xaxis/Yaxis]: **(Pick a point, or enter an option)** Specify endpoint of axis: **(Pick a point)** Specify angle of revolution <360>: **(Press Enter to accept a 360° revolution, or else type a value)** In the figure below, the edge of a 3D solid was revolved about the line. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Cor solidedge-eg.png}) TheDELOBJ (1.18.1)variable determines whether the source entities are retained or deleted following this command.

Figure 6.1912



Command Options

Option	Description
Select entities/ subentities to revolve	Selects the entities and/or sub-entities to be revolved.Choose one or more of the following entities: • **Open 2D entities** - constructs 3D surfaces • **Closed 2D entities** -constructs 3D solids or 3D surfaces, depending on the setting of Mode • **Edges on 3D solids** -constructs 3D surfaces • **Faces of DD solids** -constructs 3D solids or 3D surfaces, depending on the setting of Mode To more easily choose a face or edge on a 3D solid, hold down the **Ctrl** key. You can use almost any kind of 2D and 3D entity as the source, as long as the resulting 3D entity does not intersect itself: • Lines, and open and closed polylines • Circles, arcs, open and closed polyarcs, open and closed splines, elllipses, elliptical arcs, and helices • Edges and faces of 3D objects

MOde	Sets the 3D entity type this command will create; prompts you: **Choose type of created entity [SOlid/SUrface] <SOlid>** - **(enter an option)**: • **Solid** - constructs 3D solids • **Surface** - constructs 3D surfaces This option applies only to closed source entities; open entities always construct 3D surfaces.
Specify start point for axis of revolution	Specifies the first point of the axis, about which the entity is revolved: • Pick a point in the drawing • Enter x,y,z coordinates Prompts you: **Specify endpoint of axis** - **(pick the other point of the axis)**; as you move the cursor, the program displays a yellow, dashed drag line from the start point to the end point. When the result would bes a self-intersecting entity, the program complains: **Modeling operation error: The sweep results in self-intersecting surface**
Object	Specifies the axis of rotation by an object.Pick a line or a polyline segment. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i The axis entity must be outside of the source entity being revolved.
Xaxis	Uses the x axis of the current coordinate system as the axis of rotation.
Yaxis	Uses the y of the current coordinate system axis as the axis of rotation.
Specify angle of revolution	Specifies how far around the source entity is rotated about the axis: • Type a value larger than 0 and up to 360 for degrees • Press **Enter** to accept the default value

Related Commands

dmRevolve (6.12.22) - creates 3D solids or surfaces by revolution of open or closed 2D entities or regions about an axis. Extrude (6.13.21) - extrudes 2D entities into 3D solids or surfaces. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Subtract (6.27.56) - removes one solid model from a second. Sweep (6.27.60) - creates 3D solids or 3D surfaces by sweeping profiles (open or closed 2D entities or 3D subentities) along an open or closed path. Union (6.30.6) - joins two or more solid models into a single entity.

Figure 6.1913

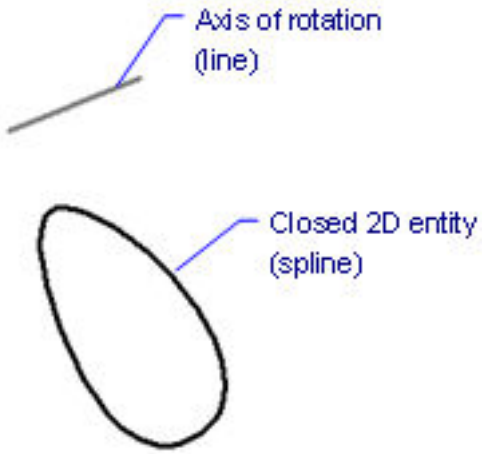
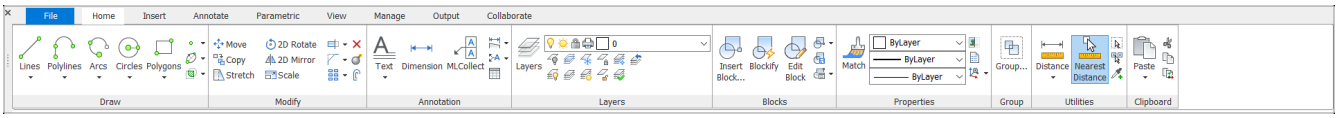


Figure 6.1914



Figure 6.1915



6.26.31 Ribbon

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the ribbon. **Note:** The content of the ribbon depends on the current workspace. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/R/ribbon-eg.png})

Accessing the Command

command bar: **ribbon** shortcut: **Ctrl+F2** menu:**View | Ribbon Bar** context menu: right-click any toolbar, choose **Ribbon** : **Ribbon** Displays the ribbon.

Command Options

No options.

Procedures

- Exploring the ribbon (1.13.6)
- Creating a ribbon tab (1.13.6)
- Editing a ribbon tab (1.13.6)
- Creating a ribbon panel (1.13.6)
- Editing a ribbon panel (1.13.6)
- Editing a panel row (1.13.6)

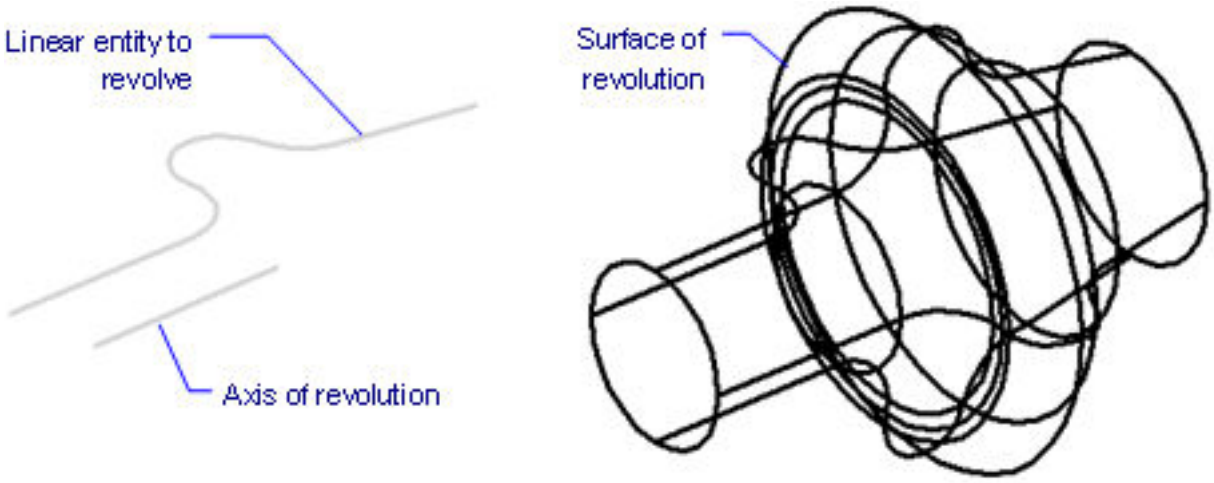
Related Commands

CleanScreenOff (6.9.21)- displays user interface elements that were hidden by teCleanScreenOn (6.9.22)command. CleanScreenOn (6.9.22) - hides elements of the user interface. RibbonClose (6.26.33) - hides the ribbon. ToolPalettes (6.29.19) - opens the Tool Palettes bar. Customize (6.9.58) - customizes the user interface of BricsCAD: menus, toolbars, keyboard shortcuts, aliases, and shell commands.

Figure 6.1916



Figure 6.1917



6.26.32 RevSurf

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a 3D mesh surface by revolving a linear entity around a line (short for "revolved surface"). **Note:** To create 3D surfaces and 3D solids that are revolved, use the Revolve (6.26.30) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image20.jpg})

Accessing the Command

 command bar: **revsurf** menu bar: **Model | Meshes | Revolved Surface** ribbon:**Model | Meshes | Revolved Surface** toolbar: **Meshes | 3D Surfaces |**  : **RevSurf Prompts you in the command bar:** Select a linear entity to revolve: **(Pick a single 2D linear or curved entity)** Select the axis of revolution: **(Pick a line or polyline segment)** Angle to begin surface of revolution <0>: **(Specify a starting angle, or press Enter to accept the default)** Degrees to revolve entity (+ for ccw, - for cw <360>): **(Specify an angle through which to revolve, or press Enter to accept the defaul.)** The SURFTAB1 (1.13.11)variable controls the number of segments of the revolution surface.The SURFTAB2 (1.13.11)variable controls the number of segments of each arc segment in the revolved entity.

Command Options

Option	Description
Select a linear entity to revolve	Specifies a single entity to be revolved into a 3D surface mesh.You may choose a line, circle, arc, open or closed polyline, or open or closed spline; 3D objects do not work.
Select the axis of revolution	Specifies the entity about which the surface is revolved; you may choose a line or polyline; curved objects do not work.
Angle to begin surface of revolution	Specifies the starting angle of the revolution; the default is 0 degrees.

Degrees to revolve entity	Specifies the extent of the revolution; to construct a closed object, start with 0 and revolve by 360 degrees.The revolved surface is drawn counterclockwise; prefix the angle with a negative sign to draw the revolution clockwise. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Note that this is not the ending angle, but the number of degrees by which the entity is revolved.

Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. Revolve (6.26.30) - revolves open or closed 2D entities and solid edges and faces to turn them into 3D solids or surfaces. Sweep (6.27.60) - creates 3D solids or 3D surfaces by sweeping profiles (open or closed 2D entities or 3D subentities) along an open or closed path. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces.

Figure 6.1918

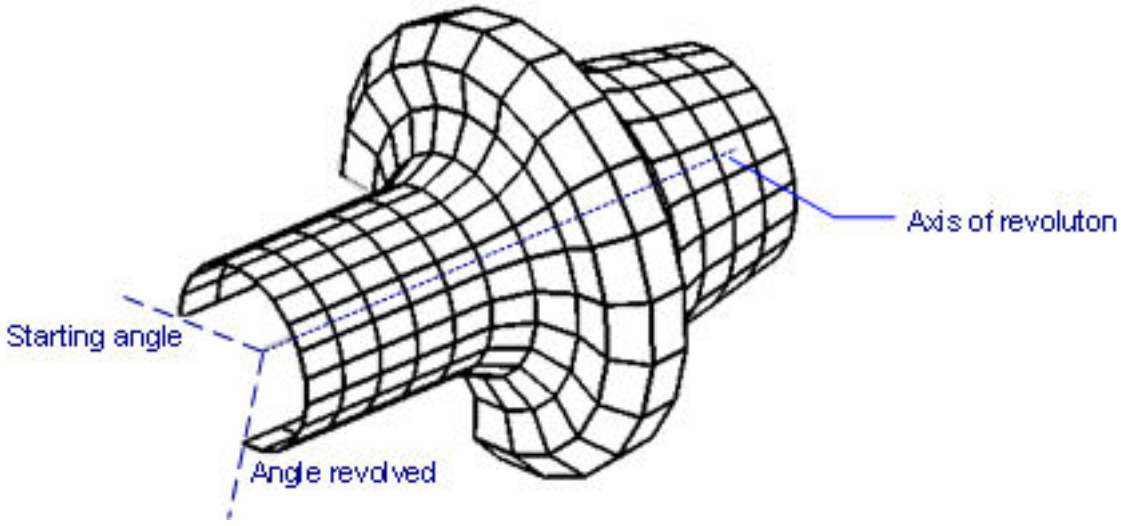


Figure 6.1919



6.26.33 RibbonClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Hides the ribbon.

Accessing the Command

command bar: **ribbon** shortcut: **Ctrl+F2** menu:**View | Ribbon Bar** context menu: right-click any toolbar, choose **Ribbon** : **RibbonClose** Closes the ribbon.

Command Options

No options.

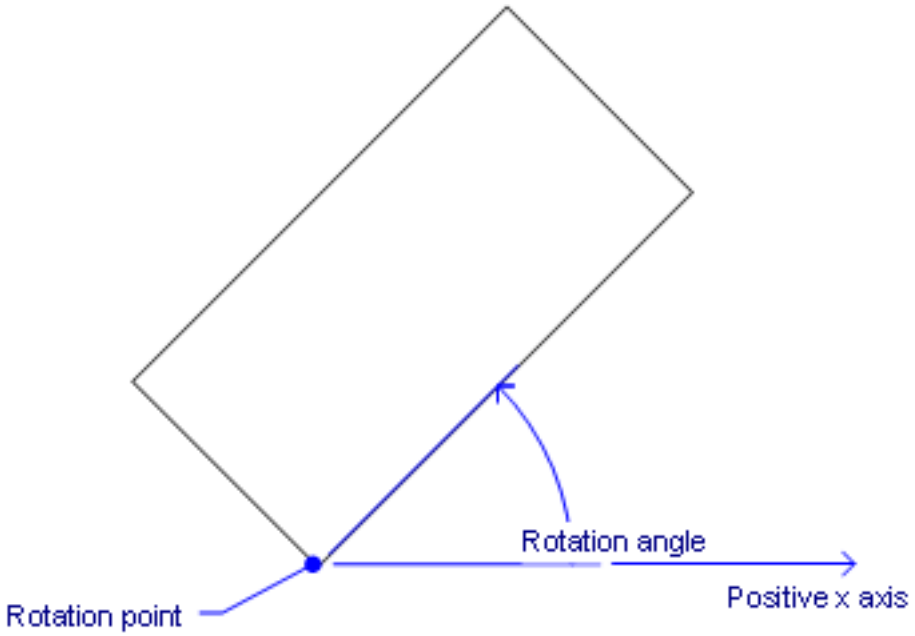
Related Commands

CleanScreenOff (6.9.21)- displays user interface elements that were hidden by teCleanScreenOn (6.9.22)command. CleanScreenOn (6.9.22) - hides elements of the user interface. Ribbon (6.26.31) - displays the ribbon.

Figure 6.1920



Figure 6.1921



6.26.34 Rotate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Rotates entities in 2D. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image492.png})

Accessing the Command

 command bar: **rotate** alias: **ro** menu bar: **Modify | 2D Rotate** ribbon: **Change | Arrange | 2D Rotate** toolbar: **Modify |**  : **Rotate Prompts you in the command bar:** Select entities to rotate: **(Select one or more entities)** Entities in set: 1 Select entities to rotate: **(Select more entities, or else press Enter to continue)** Rotation point: **(Pick a point, or enter x,y coordinates)** Rotation angle or [Base angle/Copy] <0>: **(Enter an angle, or choose an option.)**

Command Options

Option	Description
Select entities to rotate	Chooses the entities to be rotated.You can choose one or more entities using any entity selection method.Press **Enter** to end entity selection.
Rotation point	Specifies the point about which the entities will rotate: • Pick a point • Enter x,y coordinates Use entity snaps for an accurate rotation point.

Rotation angle	Specifies the angle by which to rotate the entities about the rotation point: • Pick a point to show the angle to the program. • Enter a positivenumber to rotate the entities counter clockwise • Enter a negative number to rotate the entities clockwise
Base angle	The starting angle is 0 degrees, and the rotation angle is measured from the positive x axis. Specifies the a new base angle, which is a starting angle other than 0.This option lets you define a different angle as the starting point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Prompts you: 1. **Base angle** - **(type a value or pick a point)**; you are prompted: **Second point** - **(pick a point.)** 2. **New angle** - specifies the angle by which to rotate the entities, by typing a value or picking another point
Copy	Rotates a copy of the source entity, instead of rotating the source itself.

Procedures

- Rotating a selection set (1.23.12)
- Rotating a selection set in reference to a base angle (1.23.12)
- To rotate entities using grips (1.23.4)
- To rotate the display in a viewport (1.16.14)

Related Commands

Rotate3D (6.26.35) - rotates entities about axes in 3D space. Align (6.4.6) - aligns one entity with another. Snap (6.27.38) - rotates the cursor, ortho angle, and grid. UCS (6.30.2) - rotates the coordinate system.

Figure 6.1922

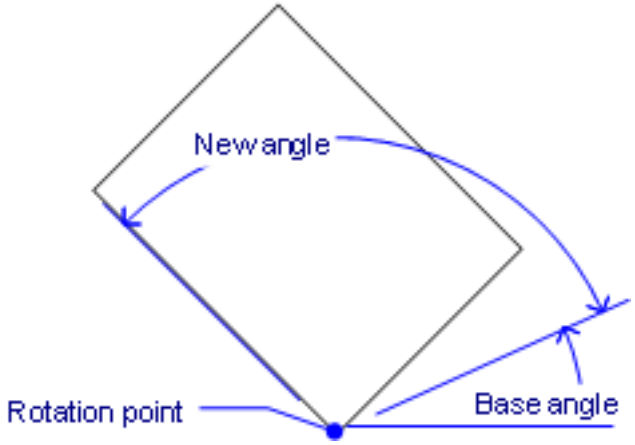


Figure 6.1923



6.26.35 Rotate3D

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Rotates entities about an axis in 3D space.

Accessing the Command

 command bar: **rotate3d** aliases: **3drotate** **3r** menu bar: **Modify | 3D Rotate** toolbar: **Modify |**  : **Rotate3D Prompts you in the command bar:** Select entities to rotate: (Choose one or more entities) Entities in set: 1 Select entities to rotate: **(Choose more entities, or press Enter to continue)** Select axis by [Entity/Last/View/Xaxis/Yaxis/Zaxis, <2points>: **(Pick a point, or enter an option)** Specify the second rotation axis point: **(Pick another point)** Rotation angle or [Base angle/Copy]: **(Type a value, or enter an option.)**

Command Options

Option	Description
Select entities to rotate	Chooses the entities to be rotated: • Choose one or more entities using any entity selection method. • Press **Enter** to end entity selection.
2 points	Defines the rotation axis by two points; prompts you: 1. **Specify the first rotation axis point** - **(pick a point.)** 2. **Specify the second rotation axis point** - **(pick a point.)**
Entity	Uses the geometry of an entity as the axis for rotating the entities; prompts you: **Select an arc, circle, ellipse, line, 2D or 3D polyline segment** - **(specify the entity.)** The program uses the geometry of the entity to determine the axis of rotation: • **Arc, circle, ellipse, 2D polyline arc segment**- axis of rotation is centered verticallyon the arc's center point • **Line,**2D polyline segment,3D polyline segment- the line or segment is the axis of rotation
Last	Uses the last entered axis for rotating the entities.
View	Uses the current view as the axis for rotating the entities; prompts you: **Specify point on view direction axis** - **(locate the axis of rotation.)** The axis of rotation is parallel to the direction of view, passing through the picked point.
Xaxis	The x axis is the axis of rotation; prompts you: **Specify point on the X axis** - **(locate the axis of rotation.)** The axis of rotation is parallel to the x axis, passing through the picked point.

Yaxis	The y axis is the axis of rotation; prompts you: **Specify point on the Y axis** - **(locate the axis of rotation.)** The axis of rotation is parallel to the y axis, passing through the picked point.
Zaxis	The z axis is the axis of rotation; prompts you: **Specify point on the Z axis** - **(locate the axis of rotation.)** The axis of rotation is parallel to the z axis, passing through the picked point.
Rotation angle	Specifies the angle by which to rotate the entities; prompts you: **Rotation angle or [Base angle/Copy]** - **(do one of the** following): • Type a value.Positive values rotate entities counterclockwise from the positive rotation axis • Specify two points **Base angle** - specifies the a new base (starting) angle; prompts you: 1. **Base angle <0>** - **(type a value, or pick two points.)** 2. **New angle** - **(type a value, or pick two points.)** **Copy** - rotates a copy of the selection set.

Procedures

- Rotating in 3D (1.23.12)

Related Commands

Rotate (6.26.34) - rotates entities about a point in 2D planes. Align (6.4.6) - aligns one entity with another. Snap (6.27.38) - rotates the cursor, ortho angle, and grid. UCS (6.30.2) - rotates the coordinate system.

Figure 6.1924



6.26.36 RScript

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Reruns the currently loaded SCR script file (short for "repeat script"). **Note:**You load and run SCR script files with the Script (6.27.9) command.

Accessing the Command

command bar: **rscript** transparent: **'rscript** : **RScript** Displays no prompts in the command bar.This command reruns the currently loaded script file. This command can be placed at the end of a script file to make it loop repeatedly.You create script files with the RecScript (6.26.5) command or through the Notepad text editor.

Command Options

Option	Description
Esc	Press **Esc** to stop the script.

Related Commands

RecScript (6.26.5) - records scripts to SCR files. StopScript (6.27.48) - stops recording the script, and saves it to an SCR file. Script (6.27.9) - loads and runs a SCR script file. Delay (6.10.23) - pauses the execution of the script for the specified number of milliseconds. Resume (6.26.28) - resumes a stopped script.

Figure 6.1925



6.26.37 RtLook

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Looks around in a 3D scene (short for "real time looking").

Accessing the Command

command bar: **rtlook** transparent: **'rtlook** mouse: hold down **Ctrl** and the middle mouse button : **RtLook Prompts you in the command bar:** >>Press ENTER or Esc to complete, or right-click to display context menu **(Hold down the mouse's left button, and then move the mouse to look around in real-time; press Enter to exit command.)** BricsCAD shows the look-around cursor during real-time rotation:

Command Options

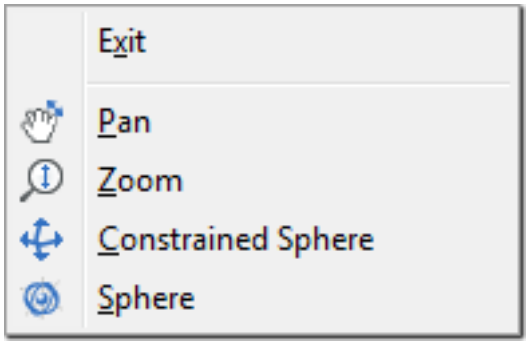
Option	Description
Enter, Esc	Exits the command.

Shortcut Menu Options

Right-click to switch viewing mode through the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en conext-mnu.png})

Option	Description
Exit	Exits the command.
Pan	Pans the drawing in real-time.Starts theRtPan (6.26.39)command.
Zoom	Zooms the drawing in real-time.Starts theRtZoom (6.26.44)command.
Constrained Sphere	Rotates the 3D drawing in real-time. StartsRtRotctr (6.26.47)the command.
Sphere	Rotates the 3D drawing in real-timeStarts theRtRot (6.26.38)command.

Figure 6.1926



Related Commands

RtRotF (6.26.48) - rotates the 3D viewpoint in real time. RtRotCtr (6.26.47) - rotates the 3D viewpoint in real time about a user defined center point. RtRotX (6.26.49) - rotates the 3D viewpoint about the x axis in realtime. RtRotY (6.26.50) - rotates the 3D viewpoint about the y axis in realtime. RtRotZ (6.26.41) - rotates the 3D viewpoint about the z axis in realtime. RtUpDown (6.26.42) - move up / down and left / right in a 3D scene. RtWalk (6.26.43) - walk left / right and forward / backward in a 3D scene. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. RtPan (6.26.39) - performs real-time panning of the drawing. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44) - performs real-time zooming of the drawing.

Figure 6.1927



6.26.38 RtRot

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Rotates 3D drawings in a constrained manner, in real-time (short for "real time rotation"). **Note:** The OrbitAutoTarget variable determines if the viewpoint rotates around your pick point (0) or the center of the objects (1).

Accessing the Command

 command bar: **rtrot** transparent: **'rtrot** shortcut: hold down **Ctrl** and right mouse button menu bar: **View | Real-Time Motion | Real-time Constrained Sphere** ribbon:**View | Navigate | Real-Time Constrained Sphere** toolbar: **Real-Time Motion |**  : **RtRot Prompts you in the command bar:** >>Press ENTER or Esc to complete, or right-click to display context menu**(Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.)** BricsCAD shows the constrained rotation cursor during real-time rotation: 

Command Options

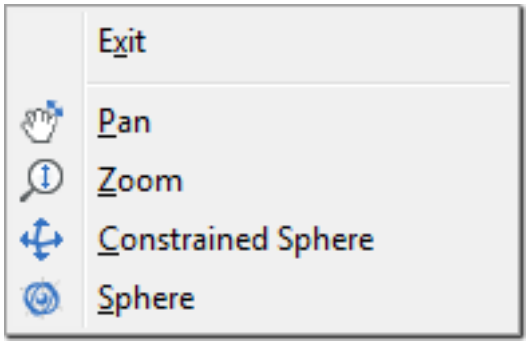
Option	Description
Enter, Esc	Exits the command.

Shortcut Menu Options

Right-click to switch viewing mode through the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en conext-mnu.png})

Option	Description
Exit	Exits the command.
Pan	Pans the drawing in real-time.Starts theRtPan (6.26.39)command.
Zoom	Zooms the drawing in real-time.Starts theRtZoom (6.26.44)command.
Constrained Sphere	Rotates the 3D drawing in real-time. StartsRtRotctr (6.26.47)the command.
Sphere	Rotates the 3D drawing in real-timeStarts theRtRot (6.26.38)command.

Figure 6.1928



Related Commands

Navigate (6.22.1) - allows to walk or fly through models using the well-known approach used in computer games. RtLook (6.26.37) - look around in a 3D scene. RrRotF (6.26.48) - rotates the 3D viewpoint in real time. RtRotCtr (6.26.47) - rotates the 3D viewpoint in real time about a user defined center point. RtRotX (6.26.49) - rotates the 3D viewpoint about the x axis in realtime. RtRotY (6.26.50) - rotates the 3D viewpoint about the y axis in realtime. RtRotZ (6.26.41) - rotates the 3D viewpoint about the z axis in realtime. RtUpDown (6.26.42) - move up / down and left / right in a 3D scene. RtWalk (6.26.43) - walk left / right and forward / backward in a 3D scene. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. RtPan (6.26.39) - performs real-time panning of the drawing. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44) - performs real-time zooming of the drawing.

Figure 6.1929



6.26.39 RtPan

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Pans the drawing in real-time (short for "real time panning").

Accessing the Command

 command bar: **rtpan** transparent: **'rtpan** shortcut: hold down middle mouse button menu bar: **View | Real-Time Motion | Real-time Pan** ribbon:**View | Navigate | Real-Time Pan** toolbar: **Real-Time Motion**| : **RtPan Prompts you in the command bar:** >>Press ENTER or Esc to complete, or right-click to display context menu**(Hold down the mouse's left button, and then move the mouse to pan the drawing in real-time; press Enter to exit command)** BricsCAD shows the pan cursor during realtime panning: 

Command Options

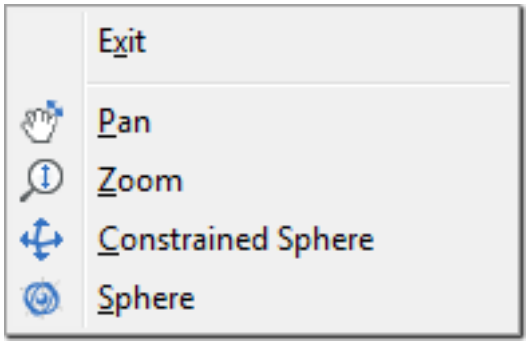
Option	Description
Shift key	Hold down the shift key to restrict panning to the X and Y direction of the viewport.
Enter, Esc	Exits the command.

Shortcut Menu Options

Right-click to switch viewing mode through the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en conext-mnu.png})

Option	Description
Exit	Exits the command.
Pan	Pans the drawing in real-time.Starts theRtPan (6.26.39)command.
Zoom	Zooms the drawing in real-time.Starts theRtZoom (6.26.44)command.
Constrained Sphere	Rotates the 3D drawing in real-time. StartsRtRotctr (6.26.47)the command.
Sphere	Rotates the 3D drawing in real-timeStarts theRtRot (6.26.38)command.

Figure 6.1930



Related Commands

Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. View (6.31.1) and -View (6.31.2) - set up named views of drawings. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. RtLook (6.26.37) - Look around in a 3D scene. RtRot (6.26.38) - rotates the 3D viewpoint in realtime. RtRotCtr (6.26.47) - rotates the 3D viewpoint in realtime about a user defined center point. RtRotF (6.26.48) - rotates the 3D viewpoint in realtime. RtRotX (6.26.49) - rotates the 3D viewpoint about the x axis in realtime. RtRotY (6.26.50) - rotates the 3D viewpoint about the y axis in realtime. RtRotZ (6.26.41) - rotates the 3D viewpoint about the z axis in realtime. RtUpDown (6.26.42) - move up / down and left / right in a 3D scene. RtWalk (6.26.43) - walk left / right and forward / backward in a 3D scene. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44) - performs real-time zooming of the drawing.

Figure 6.1931



6.26.40 RedSdkInfo

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Displays rendering related hardware and driver specifications in the command bar.

Accessing the Command

command bar: **redsdkinfo** : **RedSdkInfo** Prints a report in the command window; the content of the report depends on the graphics capabilities of your cmoputer.Press **F2** to open the Prompt History (6.29.13) window: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/RedSdkInfo.png})

Figure 6.1932

```
: redsdkinfo
RenderUsingHardware: Yes
Operating system: Win_seven64
Desktop composition (Aero) is enabled.
Local GPU:
    GPU information:
        name:     'Intel(R) HD Graphics Family'
        chipset:  'GMAHD'
        class:    'HW_INTEL_GMA_4000'
        asicID:   '278'
        vendorID: '32902'
        recommended driver: 'Intel(r) Graphics Media Accelerator Driver 15.21.10.64.2291'
    Installed driver:
        driver:      ''
        dll version: ''
        validation: 'NO_DRIVER_INSTALLED'
        You could try out this driver: 'http://www.redway3d.com/drivers/Win_seven64_Intel(r)
Graphics Media Accelerator Driver 15.21.10.64.2291'
Local GPU:
    GPU information:
        name:     'AMD Radeon HD 6800M Series'
        chipset:  'Manhattan'
        class:    'HW_ATI_R800'
        asicID:   '26792'
        vendorID: '4098'
        recommended driver: 'ATI Catalyst 11.1'
    Installed driver:
        driver:      ''
        dll version: '6.14.10.11013'
        validation: 'KNOWN_ERRORS'
        You could try out this driver: 'http://www.redway3d.com/drivers/Win_seven64_ATI
Catalyst 11.1'
```

Command Options

No Options

Related Commands

Render (6.26.22) - generates a photorealistic rendering using materials and lights through a dialog box. ShadeMode (6.27.29) - sets the shading style for 3D entities.

Figure 6.1933



6.26.41 RtRotZ

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Rotates 3D drawings in real-time about the z axis (short for "real time rotation z-axis").

Accessing the Command

 command bar: **rtrotz** transparent: **'rtrotz** menu bar: **View | Real Time Motion | Real-time Z** ribbon:**View | Navigate | Real-Time Y** toolbar: **Real-Time Motion |** **: RtRotZ Prompts you in the command bar:** >>Press ENTER or Esc to complete, or right-click to display context menu**(Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.)** BricsCAD shows the rotation cursor during real-time rotation: 

Command Options

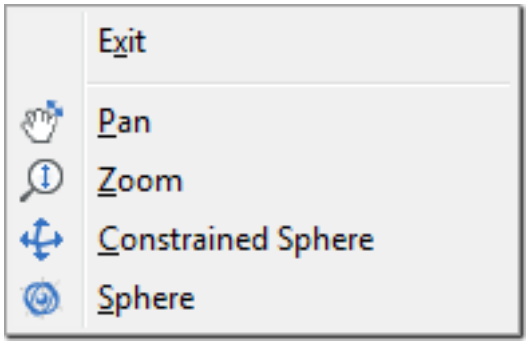
Option	Description
Enter, Esc	Exits the command.

Shortcut Menu Options

Right-click to switch viewing mode through the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en conext-mnu.png})

Option	Description
Exit	Exits the command.
Pan	Pans the drawing in real-time.Starts theRtPan (6.26.39)command.
Zoom	Zooms the drawing in real-time.Starts theRtZoom (6.26.44)command.
Constrained Sphere	Rotates the 3D drawing in real-time. StartsRtRotctr (6.26.47)the command.
Sphere	Rotates the 3D drawing in real-timeStarts theRtRot (6.26.38)command.

Figure 6.1934



Related Commands

RtLook (6.26.37) - Look around in a 3D scene. RtRot (6.26.38) - rotates the 3D viewpoint in realtime. RtRotF (6.26.48) - rotates the 3D viewpoint in real time. RtRotCtr (6.26.47) - rotates the 3D viewpoint in real time about a user defined center point. RtRotX (6.26.49) - rotates the 3D viewpoint about the x axis in realtime. RtRotY (6.26.50) - rotates the 3D viewpoint about the y axis in realtime. RtUpDown (6.26.42) - move up / down and left / right in a 3D scene. RtWalk (6.26.43) - walk left / right and forward / backward in a 3D scene. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. RtPan (6.26.39) - performs real-time panning of the drawing. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44) - performs real-time zooming of the drawing.

Figure 6.1935



6.26.42 RtUpDown

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Moves the viewpoint up and down, and left and right in a 3D scene, in real time (short for "real time up down").

Accessing the Command

command bar: **RtUpDown : RtUpDown Prompts you in the command bar:** >>Press ENTER or Esc to complete, or right-click to display context menu... **(Hold down the mouse's left button, and then move the mouse to move up-down and left-right in real-time; press Enter to exit command.)** BricsCAD shows the up-down cursor during real-time movement: 

Command Options

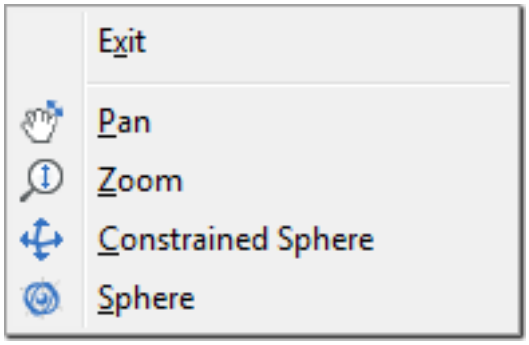
Option	Description
Enter, Esc	Exits the command.

Shortcut Menu Options

Right-click to switch viewing mode through the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en conext-mnu.png})

Option	Description
Exit	Exits the command.
Pan	Pans the drawing in real-time.Starts theRtPan (6.26.39)command.
Zoom	Zooms the drawing in real-time.Starts theRtZoom (6.26.44)command.
Constrained Sphere	Rotates the 3D drawing in real-time. StartsRtRotctr (6.26.47)the command.
Sphere	Rotates the 3D drawing in real-timeStarts theRtRot (6.26.38)command.

Figure 6.1936



Related Commands

RtLook (6.26.37) - Look around in a 3D scene. RtRotF (6.26.48) - rotates the 3D viewpoint in real time. RtRotCtr (6.26.47) - rotates the 3D viewpoint in real time about a user defined center point. RtRotX (6.26.49) - rotates the 3D viewpoint about the x axis in realtime. RtRotY (6.26.50) - rotates the 3D viewpoint about the y axis in realtime. RtRotZ (6.26.41) - rotates the 3D viewpoint about the z axis in realtime. RtWalk (6.26.43) - walk left / right and forward / backward in a 3D scene. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. RtPan (6.26.39) - performs real-time panning of the drawing. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44) - performs real-time zooming of the drawing.

Figure 6.1937



6.26.43 RtWalk

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Walks left-right and forward-backwards through 3D scenes (short for "real time walk through").

Accessing the Command

command bar: **rtwalk** mouse: **Alt** + Left Button **: RtWalk Prompts you in the command bar:** >>Press ENTER or Esc to complete, or right-click to display context menu**(Hold down the mouse's left button, and then move the mouse to walk left-right and forward-backward in real-time; press Enter to exit command.)** BricsCAD shows the walk cursor during real-time movement: 

Command Options

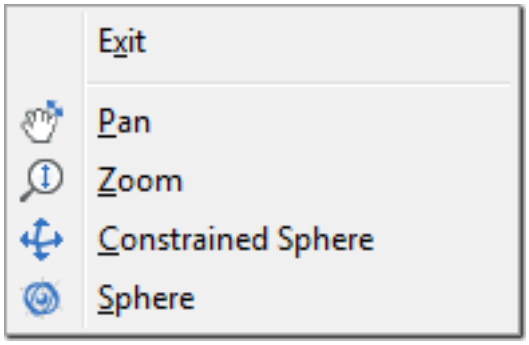
Option	Description
Enter, Esc	Exits the command.

Shortcut Menu Options

Right-click to switch viewing mode through the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en conext-mnu.png})

Option	Description
Exit	Exits the command.
Pan	Pans the drawing in real-time.Starts theRtPan (6.26.39)command.
Zoom	Zooms the drawing in real-time.Starts theRtZoom (6.26.44)command.
Constrained Sphere	Rotates the 3D drawing in real-time. StartsRtRotctr (6.26.47)the command.
Sphere	Rotates the 3D drawing in real-timeStarts theRtRot (6.26.38)command.

Figure 6.1938



Related Commands

RtRotCtr (6.26.47) - rotates the 3D viewpoint in real time about a user defined center point. Navigate (6.22.1) - allows to walk or fly through models using the well-known approach used in computer games. RtLook (6.26.37) - look around in a 3D scene. RtRotF (6.26.48) - rotates the 3D viewpoint in real time. RtRotX (6.26.49) - rotates the 3D viewpoint about the x axis in realtime. RtRotY (6.26.50) - rotates the 3D viewpoint about the y axis in realtime. RtRotZ (6.26.41) - rotates the 3D viewpoint about the z axis in realtime. RtUpDown (6.26.42) - move up / down and left / right in a 3D scene. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. RtPan (6.26.39) - performs real-time panning of the drawing. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44) - performs real-time zooming of the drawing.

Figure 6.1939



6.26.44 RtZoom

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Zooms the viewpoint in and out of the drawing in real-time (short for "real time zooming").

Accessing the Command


command bar: **rtzoom** transparent: **'rtzoom** shortcut: roll mouse wheel menu bar: **View | Real-Time Motion | Real-time Zoom** ribbon: **View | Navigate | Real-Time Zoom** toolbar: **Real-Time Motion |**  **: RtZoom Prompts you in the command bar:** >>Press ENTER or Esc to complete, or right-click to display context menu**(Hold down the mouse's left button, and then move the mouse to zoom the drawing in real-time; press Enter to exit command)**

BricsCAD shows the zoom cursor during realtime zooming: 

Command Options

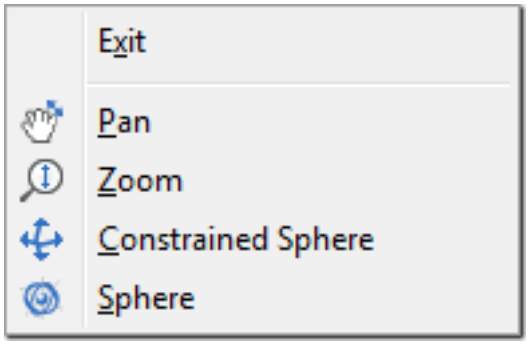
Option	Description
Enter, Esc	Exits the command.

Shortcut Menu Options

Right-click to switch viewing mode through the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en conext-mnu.png})

Option	Description
Exit	Exits the command.
Pan	Pans the drawing in real-time.Starts theRtPan (6.26.39)command.
Zoom	Zooms the drawing in real-time.Starts theRtZoom (6.26.44)command.
Constrained Sphere	Rotates the 3D drawing in real-time. StartsRtRotctr (6.26.47)the command.
Sphere	Rotates the 3D drawing in real-timeStarts theRtRot (6.26.38)command.

Figure 6.1940



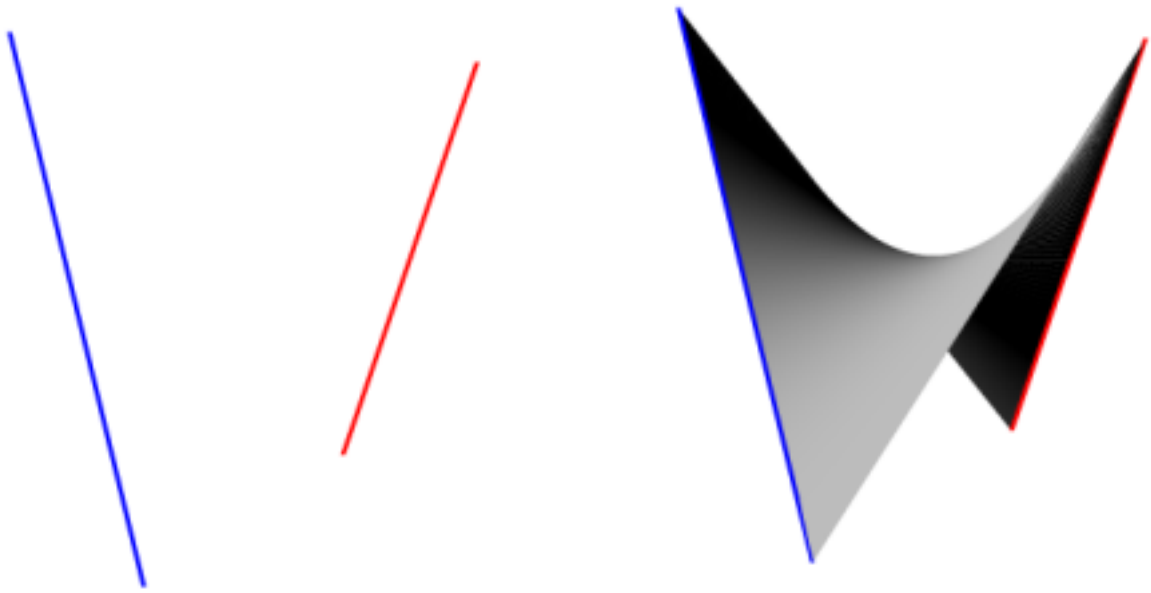
Related Commands

Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. RtPan (6.26.39) - performs real-time panning of the drawing. View (6.31.1) and -View (6.31.2) - set up named views of drawings. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. RtLook (6.26.37) - Look around in a 3D scene. **RtRot (6.26.38)** - rotates the 3D viewpoint in realtime. RtRotCtr (6.26.47) - rotates the 3D viewpoint in realtime about a user defined center point. RtRotF (6.26.48) - rotates the 3D viewpoint in realtime. RtRotX (6.26.49) - rotates the 3D viewpoint about the x axis in realtime. RtRotY (6.26.50) - rotates the 3D viewpoint about the y axis in realtime. RtRotZ (6.26.41) - rotates the 3D viewpoint about the z axis in realtime. RtUpDown (6.26.42) - move up / down and left / right in a 3D scene. RtWalk (6.26.43) - walk left / right and forward / backward in a 3D scene.

Figure 6.1941



Figure 6.1942



6.26.45 RuleSurf

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates 3D mesh surfaces by connecting two separate linear entities (short for "ruled surfaces"). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/RuleSurf.png})

Accessing the Command


command bar: **rulesurf** menu bar: **Model | Meshes | Ruled Surface** ribbon: **Model | Meshes | Ruled Surface** toolbar: **Meshes | 3D Surfaces |**  **: RuleSurf Prompts you in the command bar:** Select entity to define first end of ruled surface: (**Select an open object)** Select entity to define other end: **(Select another open object)**

Command Options

Option	Description
Select entity to define first end of ruled surface	Specifies the first boundary entity; choose any open object, such as line, arc, polyline, or spline. Closed entities and 3D entities are not accepted; prompts you: **Unable to use the selected entity. Please pick again.**
Select entity to define other end	Specifies the second boundary entity.

Grips Editing

Ruled surfaces can be edited directly through grips:

1. Select the rulesurf. Notice that it has one grip at each vertex.
2. Drag the grip to move the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image490.jpg})

Figure 6.1943



Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. TabSurf (6.29.8) - draws tabulated surfaces as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.1944



6.26.46 RunAsLevel

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Allows to run BricsCAD in a lower license level:

- Classic
- Pro
- Platinum
- with BIM add-on
- with Mechanical add-on
- with BIM and Mechanical add-on (Ultimate license level)
- with Communicator add-on

The command sets the value of the RunAsLevel*, UseBim, UseMechanical and UseCommunicator user preferences.

Note	The command name is identical to the user preference name in the English version. As a result you cannot set the value of theRunAsLevel user preference in the command line. When entering 'runaslevel' at the command prompt, the RUNASLEVEL command runs instead.

Accessing the Command

command bar: **runaslevel : RunAsLevel Prompts you in the command bar:** Specify product [BricsCAD/Communicator/Mechar <BricsCAD>: **(Choose an option)** Now restart the program for the new level to take effect.

Command Options

Option	Description
BricsCAD	Prompts you: **Run BricsCAD as [Classic/Pro/PLatinum] <Platinum>** - **(choose an option.)** • **Classic**- runs the program in Classic mode, losing the commands that are included with Pro and Platinum licenses • **Pro**- runs the program in Pro mode, losing the commands that are included with Platinum license • **Platinum**-runs the program in Platinum mode The new level comes into effect after restarting the program.The selected level is saved through the **RunAsLevel** user preference.
Mechanical	Determines the type of Mechanical license; a Platinum license is also required Prompts you: **Run Mechanical with license [No licence/Trial/Full] <Full>** - **(choose an option.)** • **No license**- Mechanical commands are not available • **Trial**- runs Mechanical in trial mode, expiring after 30 days • **Full**- runs the full Mechanical command set The new level comes into effect after restarting the program.The selected level is saved through the **UseMechanical**user preference.

BIM	Determines the type of BIM license; a Platinum license is also required Prompts you: **Run BIM with license [No licence/Trial/Full] <Full>** - **(choose an option.)** • **No license**- BIM commands are not available • **Trial**- runs BIM in trial mode, expiring after 30 days • **Full**- runs the full BIM command set The new level comes into effect after restarting the program. The selected level is saved through the **UseBIM** user preference.
Communicator	Determines the type of Communicator license; a Platinum license is also required. Prompts you: **Run Communicator with license [No licence/Trial/Full] <Trial>** - **(choose an option.)** • **No license**- Communicator import and export formats are not available • **Trial**- runs Communicator in trial mode, expiring after 30 days • **Full**- runs the full Communicator import-export set The new level comes into effect after restarting the program.The selected level is saved through the **UseCommunicator** user preference. Note thatCommunicator is installed through a separate installer.

Related Commands

LicenseManager (6.20.22) - displays the License Manager dialog, summarizing your available licenses (BricsCAD / BIM / Sheet Metal / Communicator).

Figure 6.1945



6.26.47 RtRotCtr

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Rotates 3D drawings about a center point that you pick in the drawing, in real-time (short for "real time rotation centered").

Accessing the Command

command bar: **rtrotctr** transparent: **'rtrotctr : RtRotCtr Prompts you in the command bar:** Specify rotation center point: **(Enter the center point for the rotation by clicking a point in the drawing or typing x,y,z coordinates)** >>Press ENTER or Esc to complete, or right-click to display context menu**(Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.)** BricsCAD shows the constrained rotation cursor during real-time rotation: 

Command Options

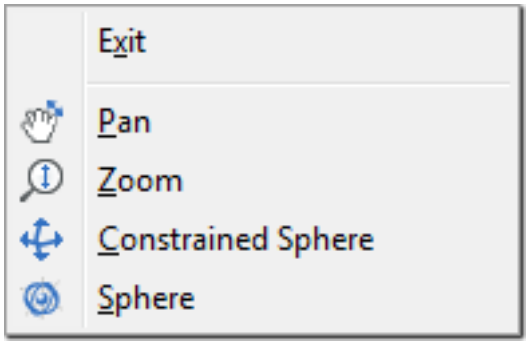
Option	Description
Enter, Esc	Exits the command.

Shortcut Menu Options

Right-click to switch viewing mode through the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en conext-mnu.png})

Option	Description
Exit	Exits the command.
Pan	Pans the drawing in real-time.Starts theRtPan (6.26.39)command.
Zoom	Zooms the drawing in real-time.Starts theRtZoom (6.26.44)command.
Constrained Sphere	Rotates the 3D drawing in real-time. StartsRtRotctr (6.26.47)the command.
Sphere	Rotates the 3D drawing in real-timeStarts theRtRot (6.26.38)command.

Figure 6.1946



Related Commands

RtLook (6.26.37) - Look around in a 3D scene. RtRot (6.26.38) - rotates the 3D viewpoint in real time. RtRotF (6.26.48) - rotates the 3D viewpoint in real time. RtRotX (6.26.49) - rotates the 3D viewpoint about the x axis in realtime. RtRotY (6.26.50) - rotates the 3D viewpoint about the y axis in realtime. RtRotZ (6.26.41) - rotates the 3D viewpoint about the z axis in realtime. RtUpDown (6.26.42) - move up / down and left / right in a 3D scene. RtWalk (6.26.43) - walk left / right and forward / backward in a 3D scene. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. RtPan (6.26.39) - performs real-time panning of the drawing. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44) - performs real-time zooming of the drawing.

Figure 6.1947



6.26.48 RtRotF

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Rotates 3D drawings about a fixed point, in real-time (short for "real time rotation fixed"). **Note:** The OrbitAutoTarget variable determines if the viewpoint rotates around your pick point (0) or the center of the objects (1).

Accessing the Command

 command bar: **rtrotf** transparent: **'rtrotf** menu bar: **View | Real-Time Motion | Real-time Sphere** toolbar: **Real-Time Motion**| :**RtRotF Prompts you in the command bar:** >>Press ENTER or Esc to complete, or right-click to display context menu**(Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.)** BricsCAD shows the rotation cursor during real-time rotation: 

Command Options

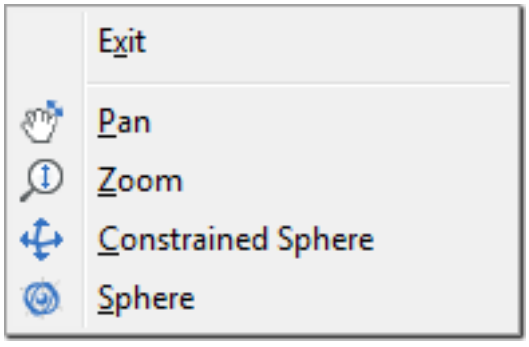
Option	Description
Enter, Esc	Exits the command.

Shortcut Menu Options

Right-click to switch viewing mode through the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en conext-mnu.png})

Option	Description
Exit	Exits the command.
Pan	Pans the drawing in real-time.Starts theRtPan (6.26.39)command.
Zoom	Zooms the drawing in real-time.Starts theRtZoom (6.26.44)command.
Constrained Sphere	Rotates the 3D drawing in real-time. StartsRtRotctr (6.26.47)the command.
Sphere	Rotates the 3D drawing in real-timeStarts theRtRot (6.26.38)command.

Figure 6.1948



Related Commands

RtLook (6.26.37)- Look around in a 3D scene. RtRot (6.26.38)- rotates the 3D viewpoint in real time. RtRotCtr (6.26.47)- rotates the 3D viewpoint in real time about a user defined center point. RtRotX (6.26.49)- rotates the 3D viewpoint about the x axis in realtime. RtRotY (6.26.50)- rotates the 3D viewpoint about the y axis in realtime. RtRotZ (6.26.41)- rotates the 3D viewpoint about the z axis in realtime. RtUpDown (6.26.42)- move up / down and left / right in a 3D scene. RtWalk (6.26.43)- walk left / right and forward / backward in a 3D scene. DdVPoint (6.10.22)- changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19)- changes the 3D viewpoint of the drawing through the command bar. Pan (6.24.2)and-Pan (6.24.3)- move the drawing within the current viewport. RtPan (6.26.39)- performs real-time panning of the drawing. Zoom (6.35.7)- increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44)- performs real-time zooming of the drawing.

Figure 6.1949



6.26.49 RtRotX

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Rotates 3D drawings in real-time about the x axis (short for "real time rotation x-axis").

Accessing the Command

command bar: **rtrotx** transparent: **'rtrotx** menu bar: **View | Real-Time Motion | Real-time X** ribbon:**View | Navigate | Real-Time Y** toolbar: **Real-Time Motion| : RtRotX Prompts you in the command bar:** >>Press ENTER or Esc to complete, or right-click to display context menu **(Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.)** BricsCAD shows the rotation cursor during real-time rotation:

Command Options

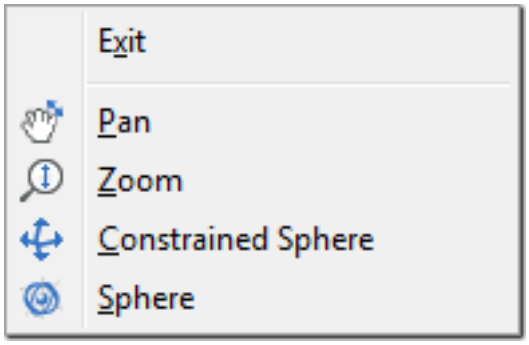
Option	Description
Enter, Esc	Exits the command.

Shortcut Menu Options

Right-click to switch viewing mode through the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en conext-mnu.png})

Option	Description
Exit	Exits the command.
Pan	Pans the drawing in real-time.Starts theRtPan (6.26.39)command.
Zoom	Zooms the drawing in real-time.Starts theRtZoom (6.26.44)command.
Constrained Sphere	Rotates the 3D drawing in real-time. StartsRtRotctr (6.26.47)the command.
Sphere	Rotates the 3D drawing in real-timeStarts theRtRot (6.26.38)command.

Figure 6.1950



Related Commands

RtLook (6.26.37) - Look around in a 3D scene. RtRot (6.26.38) - rotates the 3D viewpoint in realtime. RtRotF (6.26.48) - rotates the 3D viewpoint in real time. RtRotCtr (6.26.47) - rotates the 3D viewpoint in real time about a user defined center point. RtRotY (6.26.50) - rotates the 3D viewpoint about the y axis in realtime. RtRotZ (6.26.41) - rotates the 3D viewpoint about the z axis in realtime. RtUpDown (6.26.42) - move up / down and left / right in a 3D scene. RtWalk (6.26.43) - walk left / right and forward / backward in a 3D scene. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. RtPan (6.26.39) - performs real-time panning of the drawing. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44) - performs real-time zooming of the drawing.

Figure 6.1951



6.26.50 RtRotY

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Rotates 3D drawings in real-time about the y axis (short for "real time rotation y-axis").

Accessing the Command

 command bar: **rtroty** transparent: **'rtroty** menu bar: **View | Real-Time Motion | Real-time Y** ribbon:**View | Navigate | Real-Time Y** toolbar: **Real-Time Motion|**  **: RtRotY Prompts you in the command bar:** >>Press ENTER or Esc to complete, or right-click to display context menu... **(Hold down the mouse's left button, and then move the mouse to rotate the drawing in real-time; press Enter to exit command.) .** BricsCAD shows the rotation cursor during real-time rotation: 

Command Options

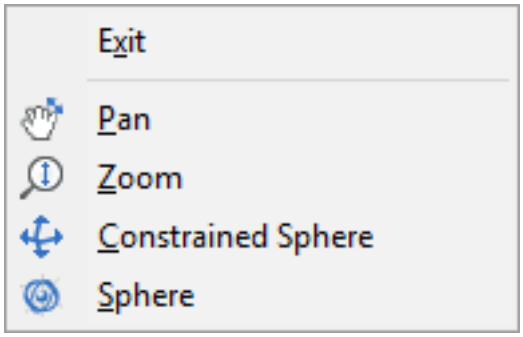
Option	Description
Enter	Exits the command.
Esc	Exits the command.

Shortcut Menu Options

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/RealTimeContextMenu.png})

Option	Description
Exit	Exits the command.
 Pan	Pans the drawing in real-time. Starts the RtPan (6.26.39) command.
 Zoom	Zooms the drawing in real-time. Starts the RtZoom (6.26.44) command.
 Constrained Sphere	Rotates the 3D drawing in real-time Starts RtRotctr (6.26.47) the command.
 Sphere	Rotates the 3D drawing in real-time Starts the RtRot (6.26.38) command.

Figure 6.1952



Related Commands

RtLook (6.26.37) - Look around in a 3D scene. RtRot (6.26.38) - rotates the 3D viewpoint in realtime. RtRotF (6.26.48) - rotates the 3D viewpoint in real time. RtRotCtr (6.26.47) - rotates the 3D viewpoint in real time about a user defined center point. RtRotX (6.26.49) - rotates the 3D viewpoint about the x axis in realtime. RtRotZ (6.26.41) - rotates the 3D viewpoint about the z axis in realtime. RtUpDown (6.26.42) - move up / down and left / right in a 3D scene. RtWalk (6.26.43) - walk left / right and forward / backward in a 3D scene. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. RtPan (6.26.39) - performs real-time panning of the drawing. Zoom (6.35.7) - increases and decreases the size of the drawing within the current viewport. RtZoom (6.26.44) - performs real-time zooming of the drawing.

6.27 S

6.27.1 Save

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Saves drawings in DWG drawing, DWT template, and DXF interchange formats; always displays a dialog box.

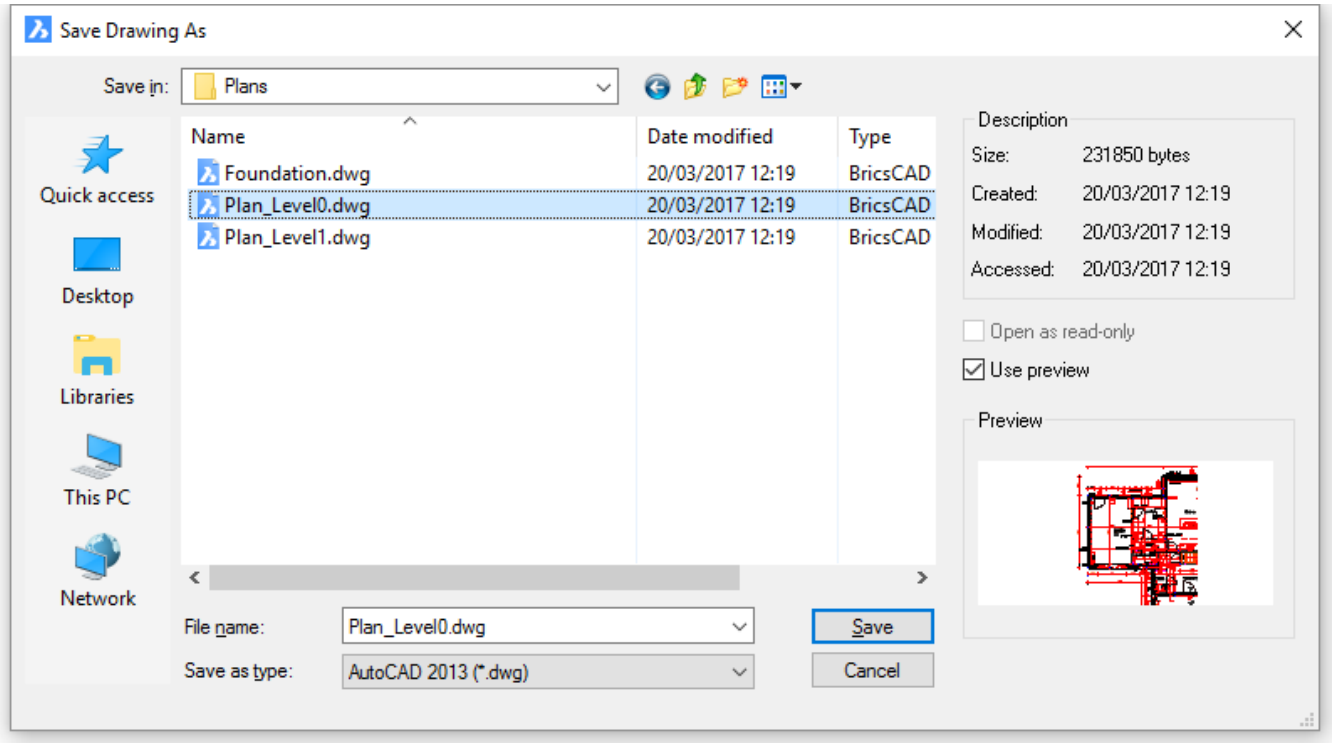
Figure 6.1953



Accessing the Command

 command bar: **save** shortcut: **Ctrl+S** alias: **sa** toobar: **Standard** |  ribbon: **Home | File | Save : Save** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SaveAsDialog.pn
Enter a file name, and then click **Save**. Use the QSave (6.25.4) command to save drawings without seeing this dialog box.

Figure 6.1954



Command Options

Option	Description
Save in	Selects the drive and folder in which to save the drawing. • Click the droplist to access additional locations. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
File name	Specifies the name by which to save the drawing: • Accept the default **drawing.dwg** file name • Enter another name • Click the droplist to access names of previously-saved drawings: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Save as Type	Specifies the format of the saved file. Accept the default (AutoCAD 2013 .dwg) or click the droplist to choose another format.There are three basic formats: • **DWG** - drawing file • **DXF** - interchange file (ASCII or Binary) • **DWG** - template file (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

Use Preview	Determines whether a preview image is included with the drawing file: • **Yes** - includes the bitmap preview image, which is displayed by file managers and other programs • **No** - does not include the preview image The preview is a bitmap (raster) image of the drawing's view at the time it is saved.
Save	Saves the drawing, and then returns to the drawing editor.
Cancel	Cancels the command: does not save the drawing, and returns to the drawing editor.

Procedures

- Working with document tabs (1.12.2)

Related Commands

Export (6.13.15) - exports drawings in a variety of vector and raster formats. QSave (6.25.4) - saves the drawing without displaying the Save dialog box. SaveAll (6.27.2) - saves all open drawings. SaveAs (6.27.3) - saves drawings by another name. SaveAsR12 (6.27.4) - saves drawing in older formats. SaveFileFolder (6.27.5) - opens the SAVEFILEPATH location in the system file browser. SecurityOptions (6.27.15) - sets a password to protect a drawing

Figure 6.1955

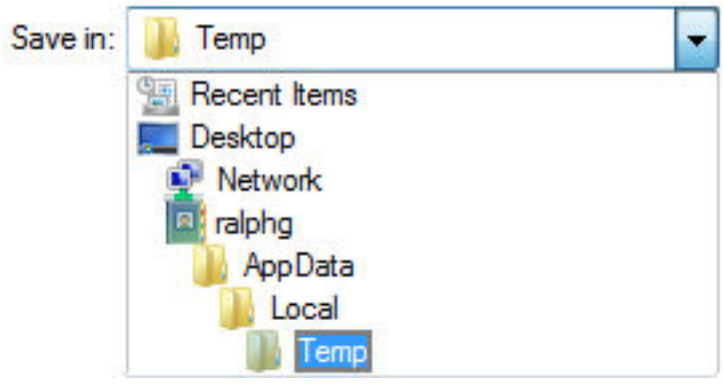


Figure 6.1956

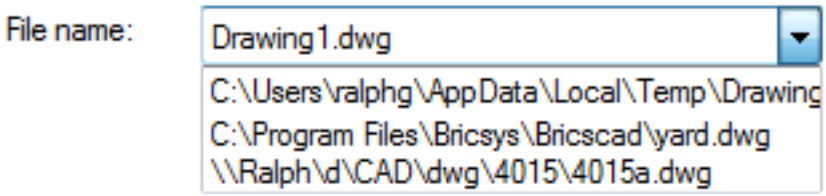


Figure 6.1957

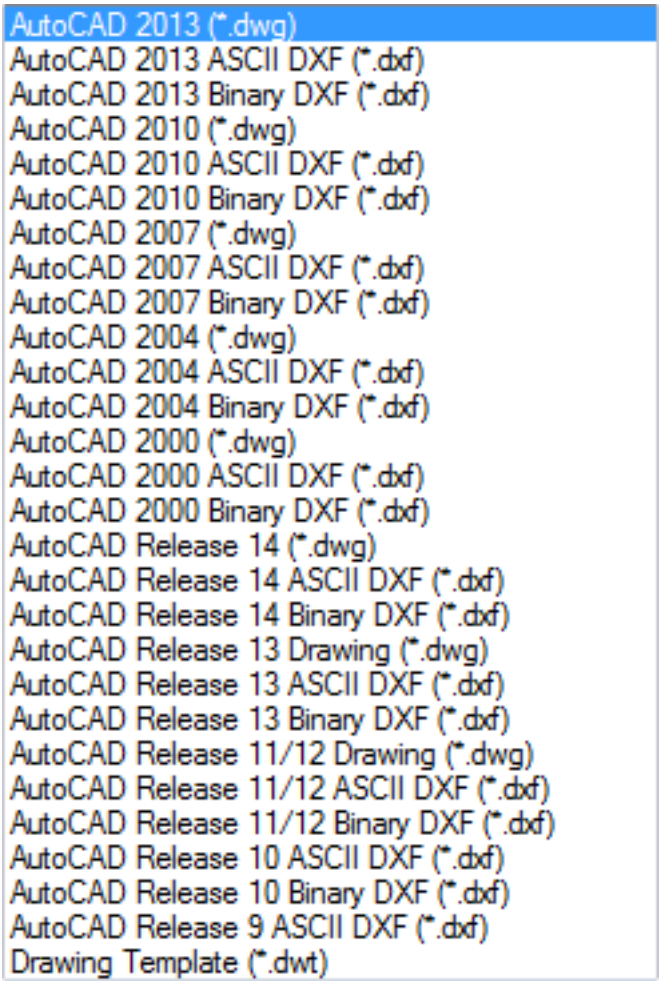


Figure 6.1958



6.27.2 SaveAll

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Saves all open drawings. **Note**: This command saves all drawings with no "Are you sure?" check. BricsCAD may save drawings that you do not intend to be saved, and so could overwrite older versions of the files.

Accessing the Command

command bar: **saveall** document tab: right-click and choose **Save All : SaveAll** Displays no prompt in the command bar. Saves all open drawings. When a drawing has not yet been saved, displays the Save Drawing As dialog box; see SaveAs (6.27.3) command.

Command Options

No options.

Procedures

- Working with document tabs (1.12.2)

Related Commands

Save (6.27.1) - saves the current drawing through a dialog box. QSave (6.25.4) - saves the drawing without displaying the Save dialog box. SaveAs (6.27.3) - saves drawings by another name. SaveAsR12 (6.27.4) - saves drawing in older formats. Export (6.13.15) - exports drawings in a variety of vector and raster formats.

Figure 6.1959



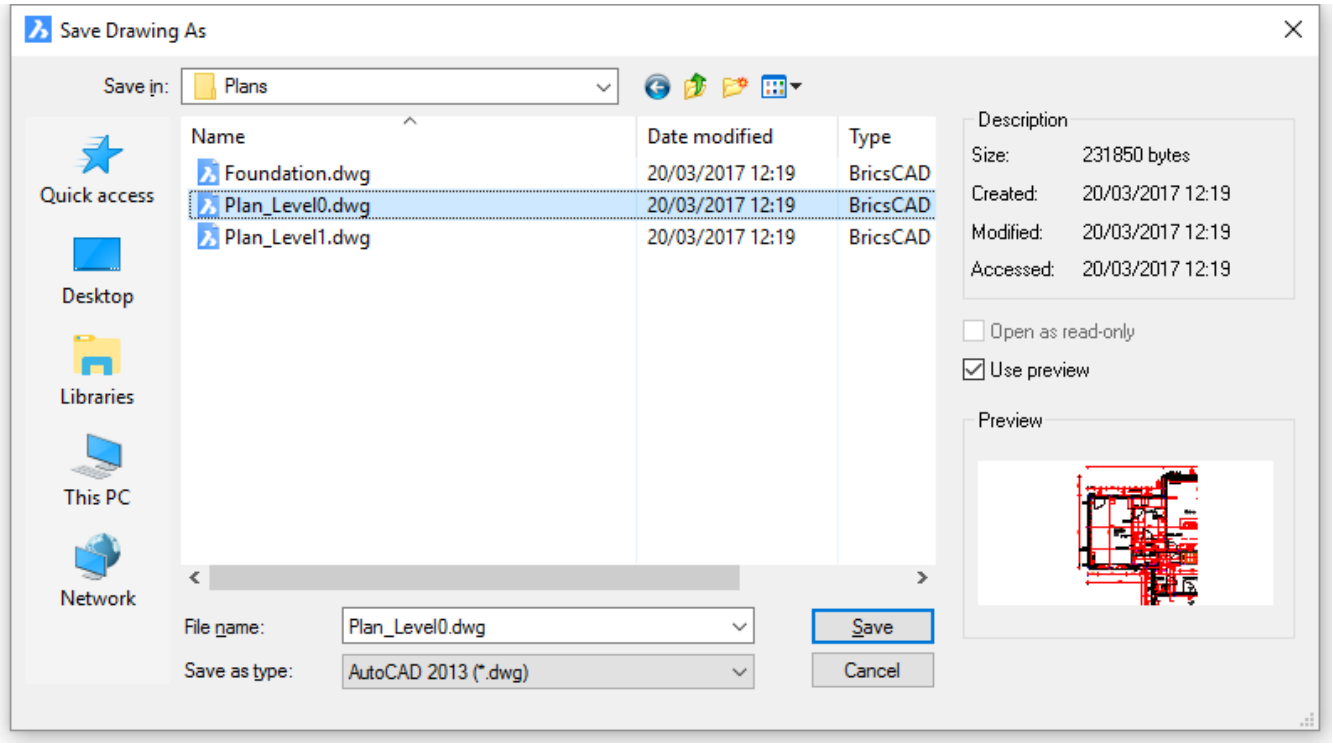
6.27.3 SaveAs

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Saves drawings in DWG drawing, DWT template, and DXF interchange formats; always displays a dialog box. **Note:** This command is intended for saving drawings by different file names or format, such as ones opened in read-only mode.

Accessing the Command

 command bar: **saveas** shortcut: **Ctrl+Shift+S** menu:**File | Save As** ribbon:**Home | File | Save As : SaveAs**
Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SaveAsDialog
Enter a file name, and then click **Save**. Use the QSave (6.25.4) command to save drawings without seeing this dialog box.

Figure 6.1960



Command Options

Option	Description
Save in	Selects the drive and folder in which to save the drawing. • Click the droplist to access additional locations. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
File name	Specifies the name by which to save the drawing: • Accept the default **drawing.dwg** file name • Enter another name • Click the droplist to access names of previously-saved drawings: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Save as Type	Specifies the format of the saved file. Accept the default (AutoCAD 2013 .dwg) or click the droplist to choose another format.There are three basic formats: • **DWG** - drawing file • **DXF** - interchange file (ASCII or Binary) • **DWG** - template file (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

Use Preview	Determines whether a preview image is included with the drawing file: • **Yes** - includes the bitmap preview image, which is displayed by file managers and other programs • **No** - does not include the preview image The preview is a bitmap (raster) image of the drawing's view at the time it is saved.
Save	Saves the drawing, and then returns to the drawing editor.
Cancel	Cancels the command: does not save the drawing, and returns to the drawing editor.

Procedures

- Working with document tabs (1.12.2)

Related Commands

Export (6.13.15) - exports drawings in a variety of vector and raster formats. QSave (6.25.4) - saves the drawing without displaying the Save dialog box. Save (6.27.1) - saves the current drawing through a dialog box. SaveAll (6.27.2) - saves all open drawings. SaveAsR12 (6.27.4) - saves drawing in older formats. SecurityOptions (6.27.15) - sets a password to protect a drawing

Figure 6.1961

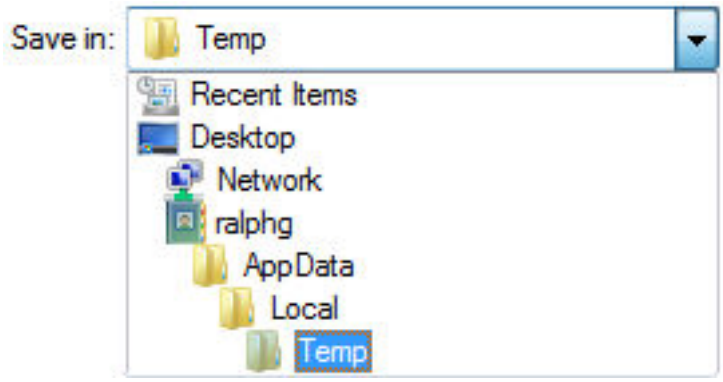


Figure 6.1962

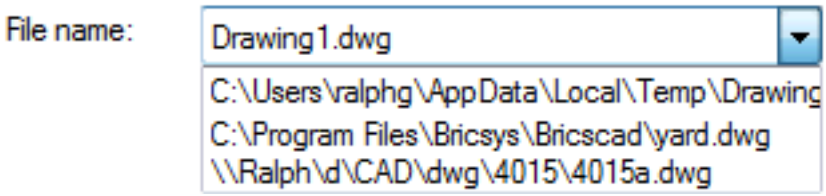


Figure 6.1963

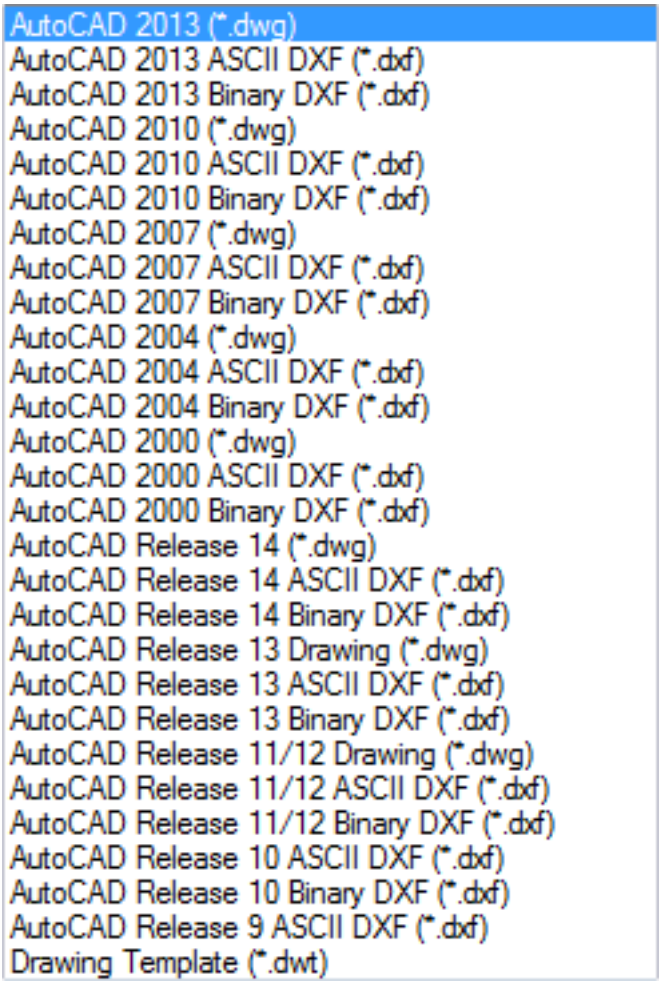


Figure 6.1964



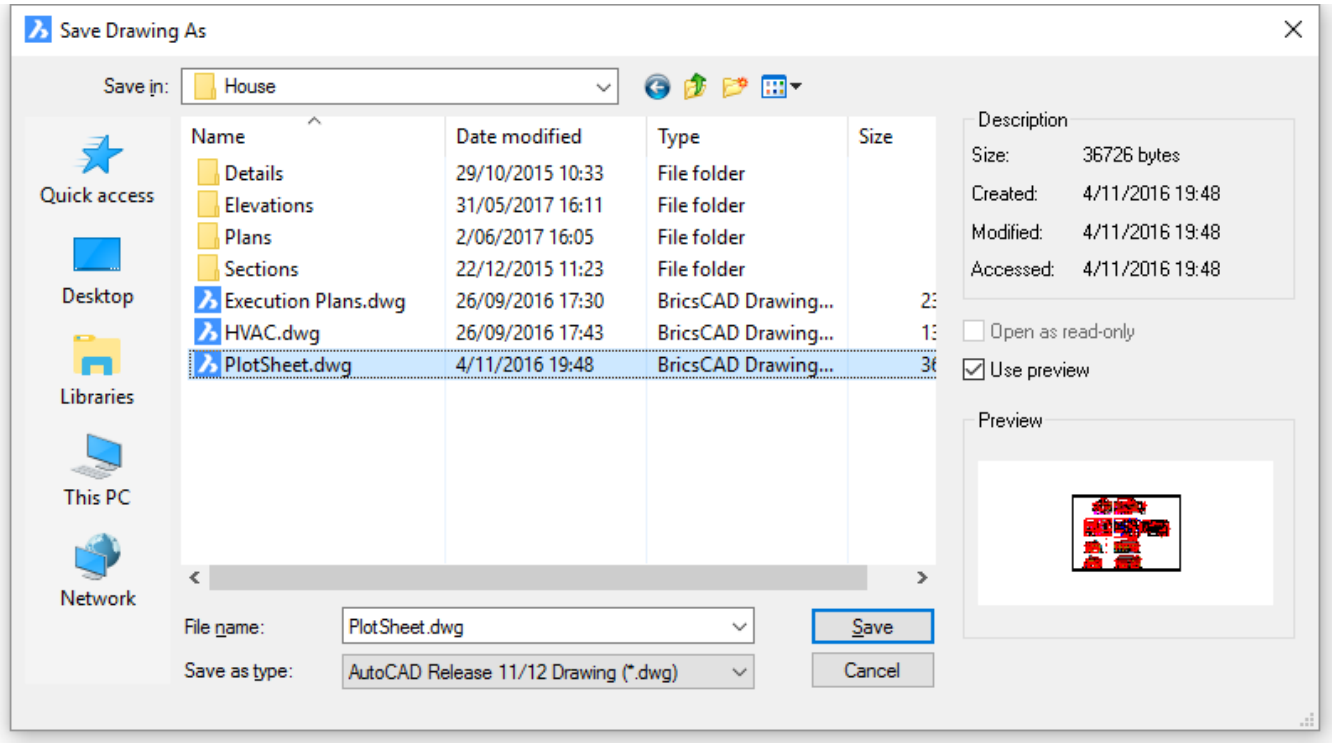
6.27.4 SaveAsR12

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Saves drawings in DWG R12 format, through a dialog box.

Accessing the Command

command bar: **saveas12 : SaveAsR12** Displays a dialog box with Save As Type set to "AutoCAD Release 11/12 Drawing (*.dwg)": (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SaveAsR12Dialog Enter a file name, and then click **Save**. To save the drawing in other formats, click the **Save As Type** droplist.

Figure 6.1965



Command Options

Option	Description
Save in	Selects the drive and folder in which to save the drawing. • Click the droplist to access additional locations. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
File name	Specifies the name by which to save the drawing: • Accept the default **drawing.dwg** file name • Enter another name • Click the droplist to access names of previously-saved drawings: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Save as Type	Specifies the format of the saved file. Accept the default (AutoCAD 2013 .dwg) or click the droplist to choose another format.There are three basic formats: • **DWG** - drawing file • **DXF** - interchange file (ASCII or Binary) • **DWG** - template file (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

Use Preview	Determines whether a preview image is included with the drawing file: • **Yes** - includes the bitmap preview image, which is displayed by file managers and other programs • **No** - does not include the preview image The preview is a bitmap (raster) image of the drawing's view at the time it is saved.
Save	Saves the drawing, and then returns to the drawing editor.
Cancel	Cancels the command: does not save the drawing, and returns to the drawing editor.

Related Commands

Save (6.27.1) - saves the current drawing through a dialog box. QSave (6.25.4) - saves the drawing without displaying the Save dialog box. SaveAll (6.27.2) - saves all open drawings. SaveAs (6.27.3) - saves drawings by another name. Export (6.13.15) - exports drawings in a variety of vector and raster formats.

Figure 6.1966

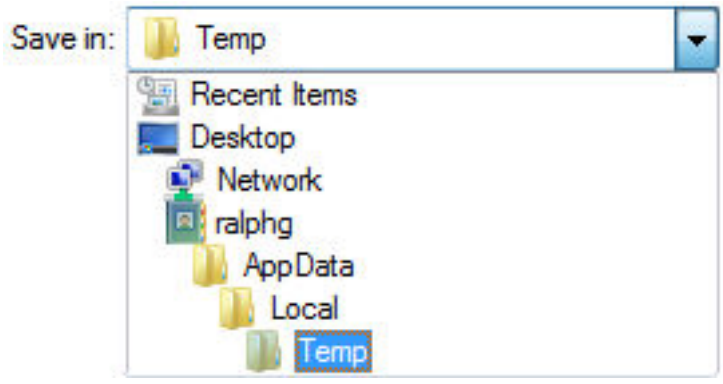


Figure 6.1967

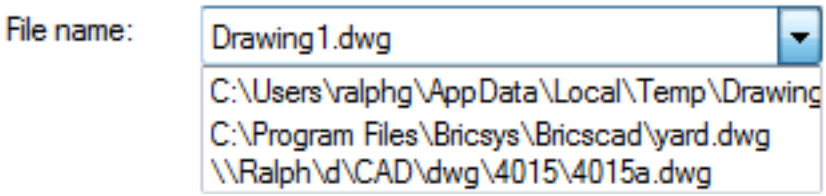


Figure 6.1968

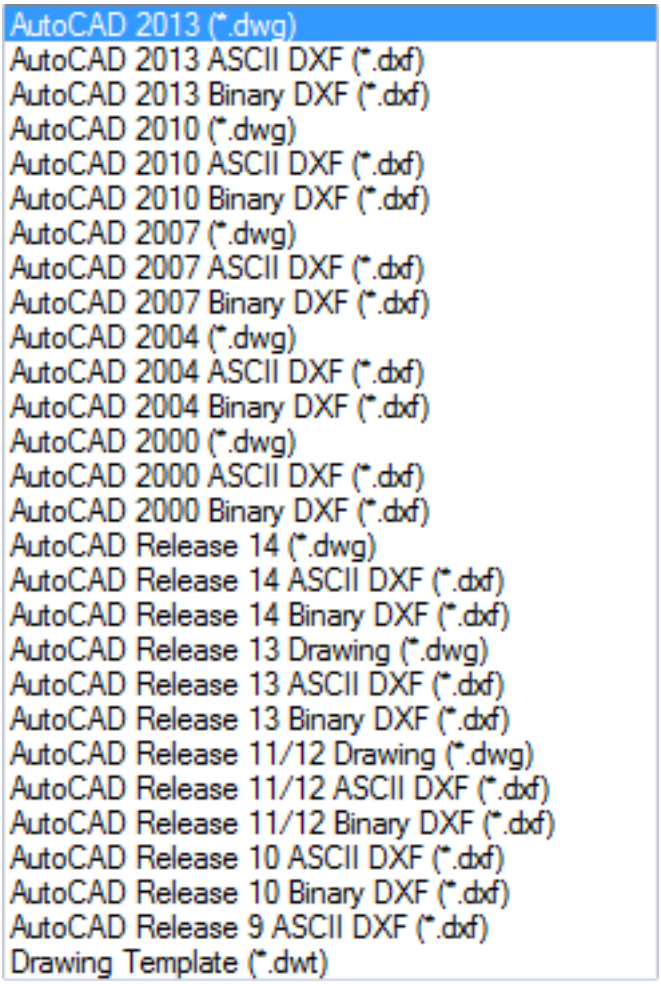


Figure 6.1969



6.27.5 SaveFileFolder

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Opens the folder in which the automatic save and backup files are being saved, usually**C:\Users\login\AppData\Local\Temp** by default. **Note:** Use this command to quickly find AC$ and BAK files, with which you can recover ”lost” files.

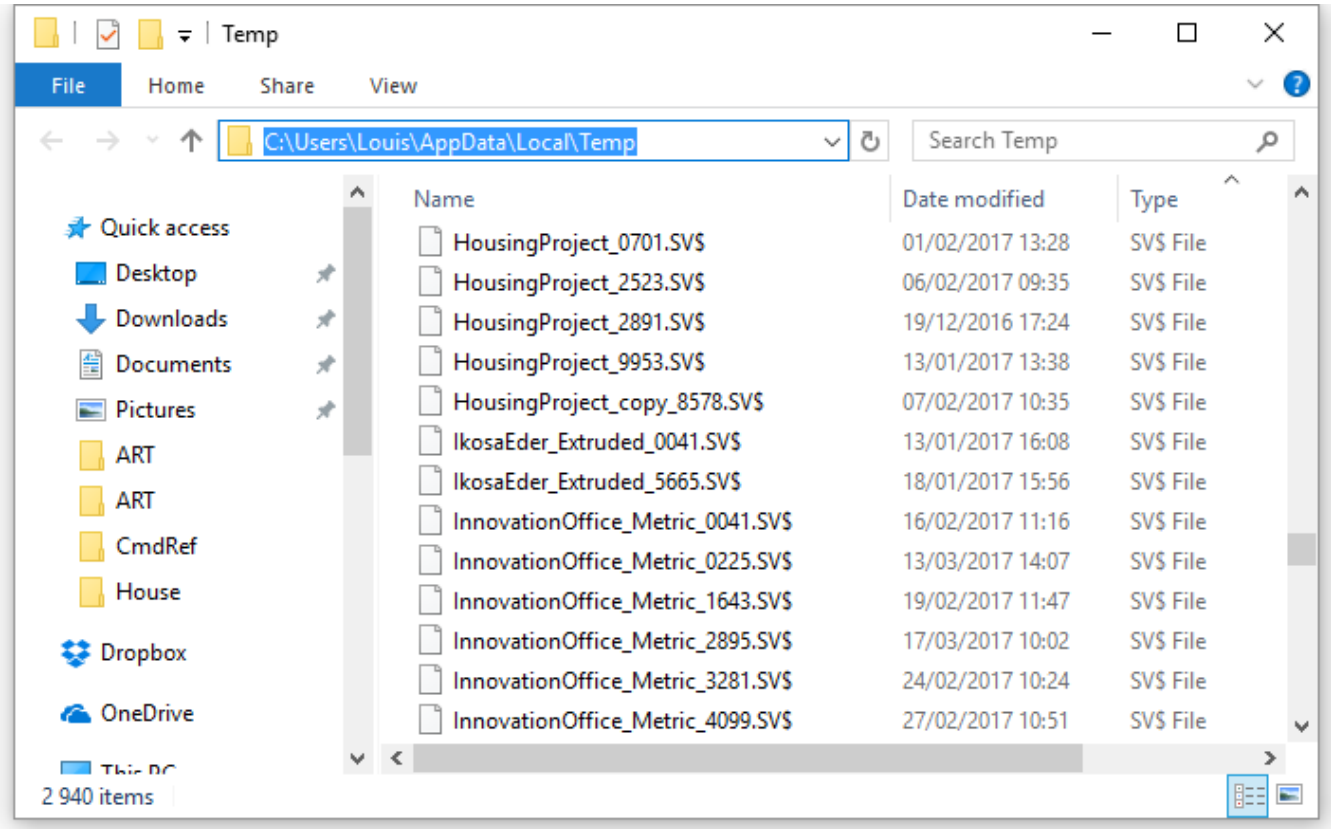
Accessing the Command

command bar: **savefilefolder** transparent: ’**savefilefolder : SaveFileFolder** Opens a file dialog box in the folder defined by theSAVEFILEPATH variable: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ This folder holds temporary files used by many applications. This program stores the following files there:

- **.SV$** - automatic backup files made by the program automatically, at the interval defined by the SAVETIME variable (default = 60 minutes).
- **.BAK** - manual backup files made each time you save the drawing, such as by pressing**Ctrl+S** or entering commands like Save and SaveAll.

You can change the location of the backup folder by editing the value oftheSAVEFILEPATH variable in the Settings (6.27.23) dialog box. You can rename the extension of these files to .DWG and then open them in a CAD program.

Figure 6.1970



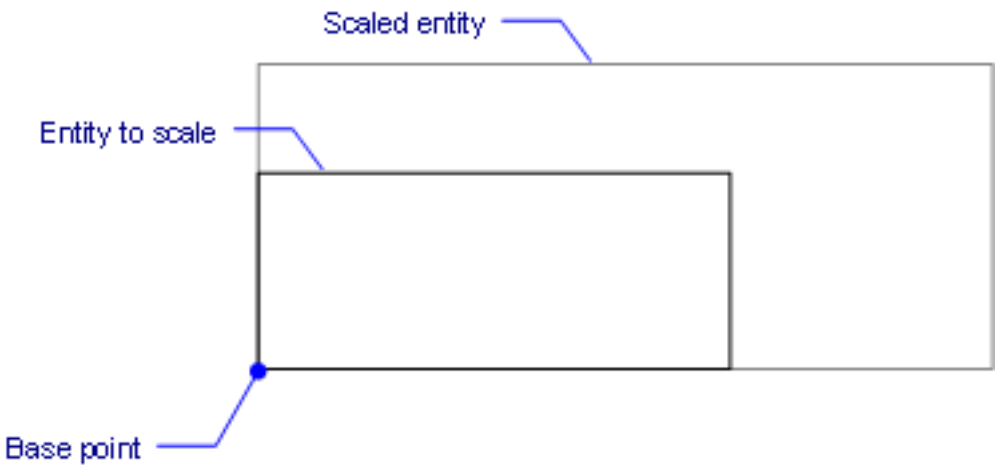
Command Options

No command options.

Related Commands

Save (6.27.1) - saves drawings in DWG drawing, DWT template, and DXF interchange formats.

Figure 6.1971



6.27.6 Scale

Resizes 2D and 3D entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image52

Accessing the Command

 command bar: **scale** alias: **sc** menu bar: **Modify | Scale** ribbon:**Change | Modify | Scale Solid | Edit | Scale** toolbar: **Modify |**  **: Scale Prompts you in the command bar:** Select entities to scale:**(Choose one or more entities)** Entities in set: 1 Select entities to scale:**(Choose more entities, or else press Enter to continue)** Base point:**(Pick a point, or enter x,y,z coordinates)** Scale factor or [Reference/Copy] <1>:**(Enter a number, or enter an option)**

Command Options

Option	Description
Select entities to scale	Chooses the entities to be resized: • Use any selection method • Enter **all** to select all entities in the drawing • Press **Enter** to end entity selection, and carry on
Base point	Specifies the base point, the point from which the entities ”grow” or ”shrink”: • Pick a point in the drawing • Enter x,y,z coordinates Use entity snap to pick a geometric feature accurately.

Scale factor	Specifies the scale factor: • Enter a number smaller than 1 to shrink the entities • Enter 1 to leave the entities the same size (black in the figure below) • Enter a number larger than 1 to make the entities larger (green, in the figure below) • Enter a negative number to scale the entities in the negative direction (red, in the figure) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/sc eg.png})
Copy	Scales a copy of the selection set, leaving the original entities intact.
Reference	Resizes entities using a base or **reference** scale; prompts you: 1. **Reference scale <1>** - **(enter a scale factor, or pick two points.)** 2. **New scale <1>** - **(enter the new scale factor.)** Use this option to scale entities relative to other entities.For example, pick two points on an existing entity; when you pick two points, BricsCAD prompts you: **Second point** - **(pick another point.)** This command normally scales from a base factor of 1.

Procedures

- To scale a selection set (1.23.24)
- Scaling an entity using a base scale (1.23.24)
- To scale entities using grips (1.23.4)

Related Commands

Align (6.4.6) - scales one entity with another. Insert (6.17.13) - scales blocks. Print (6.24.46) - scales drawings upon plotting.

Figure 6.1972

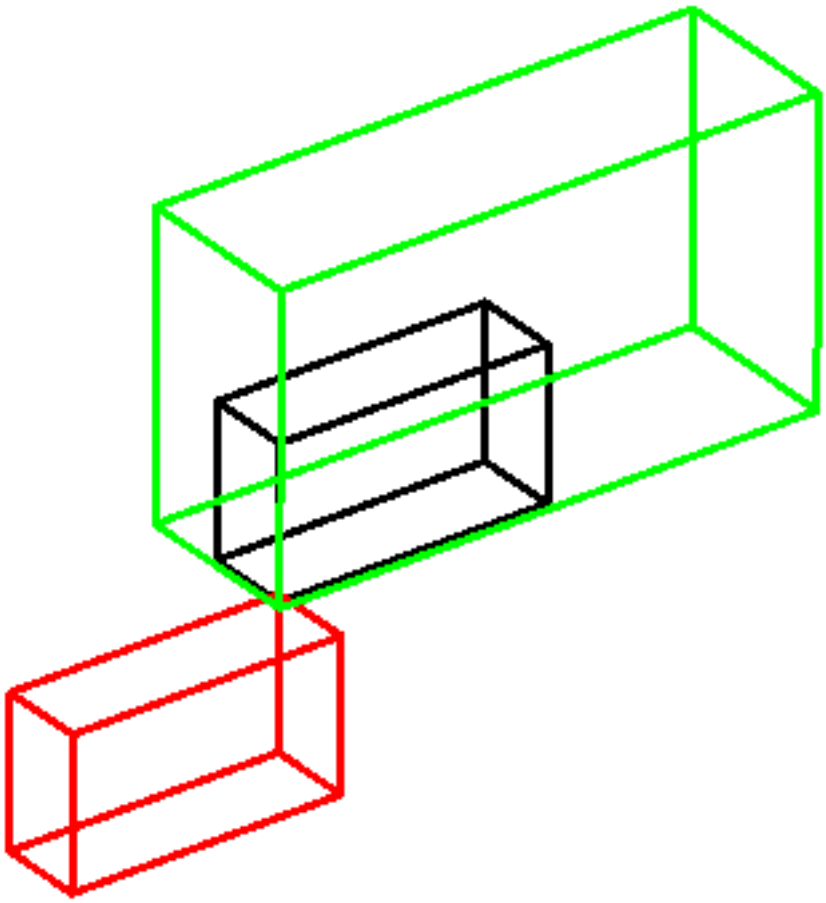


Figure 6.1973



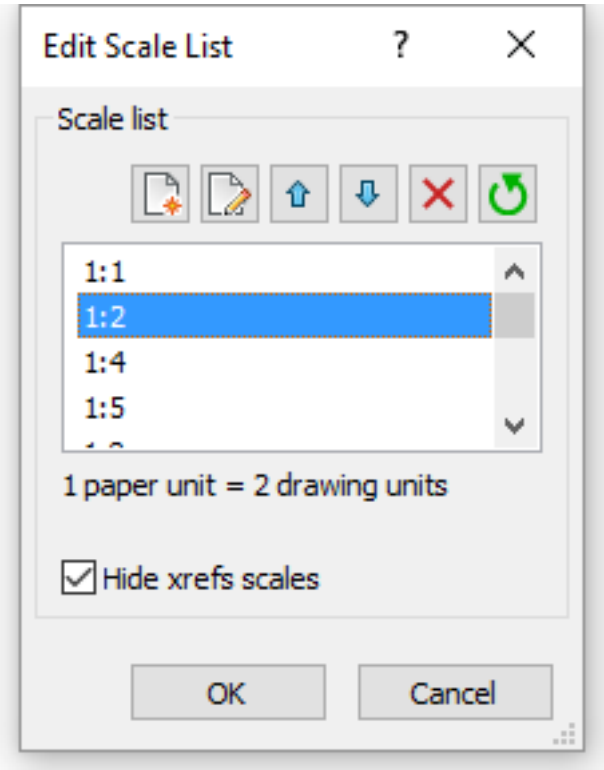
6.27.7 ScaleListEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits the list of scale factors displayed by commands, such asPrint and PageSetup, and by annotative scale factors.

Accessing the Command

command bar: **scalelistedit** status bar: right-click **Annotation Scale** and choose **Custom : ScaleListEdit** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ScaleListEditDialog.png}

Figure 6.1974



Command Options

Option	Description
Scale List	Lists the scale factors that are displayed by other commands. To add and remove scale factors, click the buttons: • **Add a New Scale** - adds a scale factor to the list; see below • **Edit Scale** - edits the factors of an existing scale; see below • **Move Up** - moves the selected factor up the list • **Move Down** - moves the selected factor down the list • **Delete the Selected Scale**- deletes the selected factor(s); hold down the **Ctrl** key to select and delete more than one factor at a time • **Reset** - returns the list to its original pristine condition, adding back in the deleted factors and removing those that were added but are not being used

 Add a New Scale	Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S 1. **Name appearing in scale list** - **(type a name , such as the scale factor itself (1:333) or a descriptive name, like Triple-Treat.)** 2. **Scale properties** - **(specify the scale factor in paper (size when plotted) and drawing units.)** 3. **OK**- adds the scale to the list
 Edit Scale	Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S 1. **Name appearing in scale list** - **(edit the name.)** 2. **Scale properties** - **(change the scale factors in paper (size when plotted) and/or drawing units)** 3. **OK**- save the changes to the list
Hide xrefs scales	Determines if scale factors from xrefs are displayed in this list: • **Yes** - scale factors from externally-referenced drawings are not shown in this list • **No** - scales in the attached xrefs not in the current drawing are listed here. Togglesthe HIDEXREFSCALES variable.
OK	Saves the changes made to the list, and then closes the dialog box.
Cancel	Closes the dialog box without saving the changes you made.

Related Commands

ObjectScale (6.23.1) - adds or removes supported scales for annotative entities through a dialog box. -ObjectScale (6.23.2) - adds or removes supported scales for annotative entities at the command line. Print (6.24.46) - prints drawings to plotters and files through a dialog box. PageSetup (6.24.1) - creates and edits page setups for plotting drawings in the Drawing Explorer. -ScaleListEdit (6.27.8) - edits the scale list of a drawing in the command bar.

Figure 6.1975

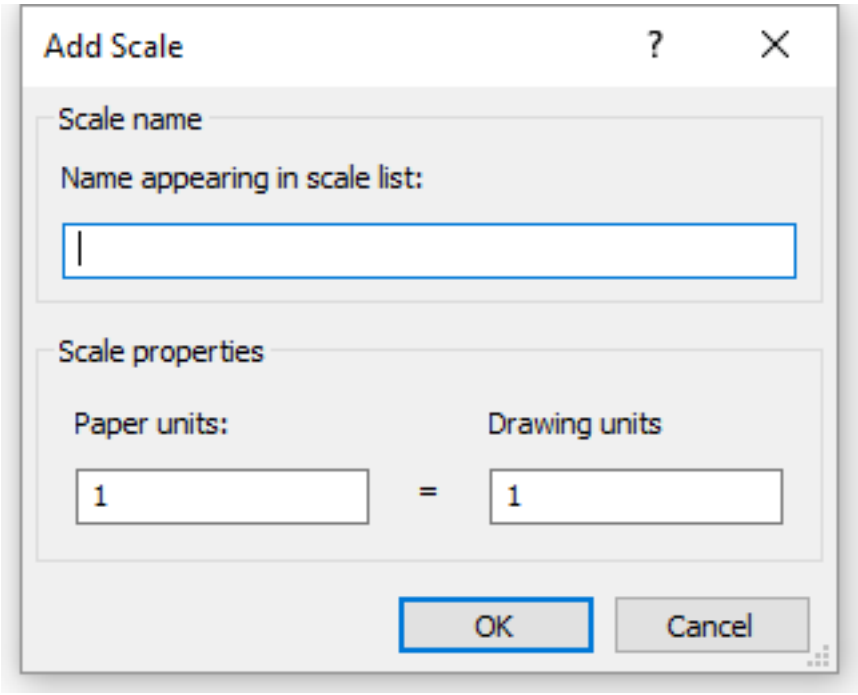


Figure 6.1976

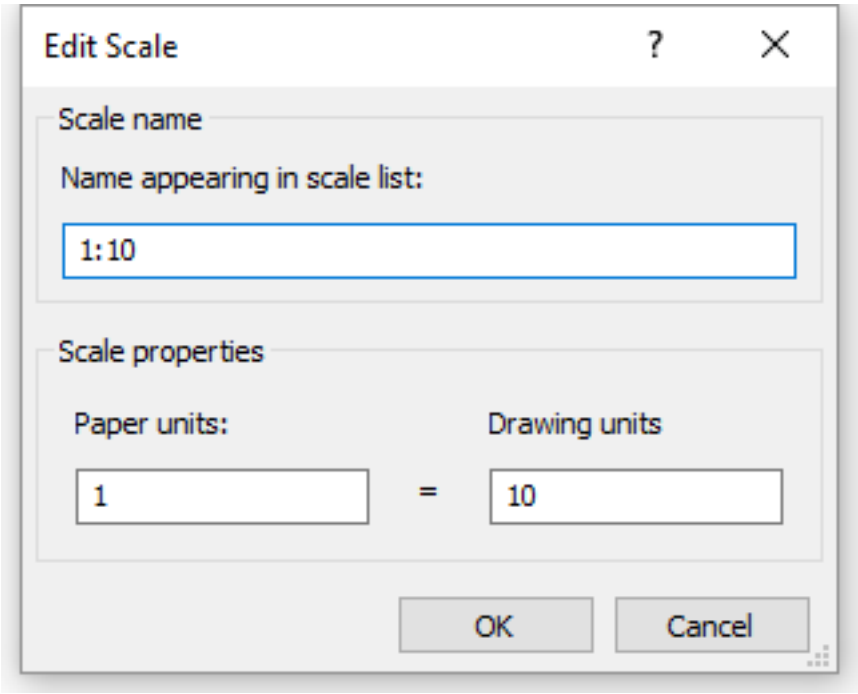


Figure 6.1977



6.27.8 -ScaleListEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits the list of scale factors displayed by commands, such asPrint and PageSetup and by annotative scale factors, at the command line.

Accessing the Command

command bar: **-scalelistedit : -ScaleListEdit Prompts you in the command bar:** Enter option [?/Add/Delete/Reset/Exit] <Add>: **(Enter an option)**

Command Options

Option	Description
?	Displays the current scale list in the command bar.Press the **F2** function key to open the Prompt History window, such as `Scale Name Paper Units Drawing Units Effective Scale`
Add	Creates a new scale; prompts you: 1. **Enter new scale name** - **(type a name for the scale in the command bar.)** 2. **Enter scale ratio** - **(type the scale ration in the command bar.)** The following formats are accepted to define a scale of, say, 1 paper unit = 10 drawing units. • 1:10 • 1=10
Delete	Removes unused scale factors from this list; prompts your: **Enter scale name to delete or * to delete all unused scales** - **(Do one of the following:)** • Type the name of the scale to delete. • Type * to delete all unreferenced scales.
Reset	Reloads hard-coded default scales;Deletes all unreferenced user defined scales.
Exit	Concludes the command.

Related Commands

Print (6.24.46) - prints drawings to plotters and files through a dialog box. PageSetup (6.24.1) - creates and edits page setups for plotting drawings in the Drawing Explorer. ScaleListEdit (6.27.7) - edits the scale list of a drawing in a dialog box.

Figure 6.1978



6.27.9 Script

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Loads and runs SCR script files through a dialog box.

Accessing the Command

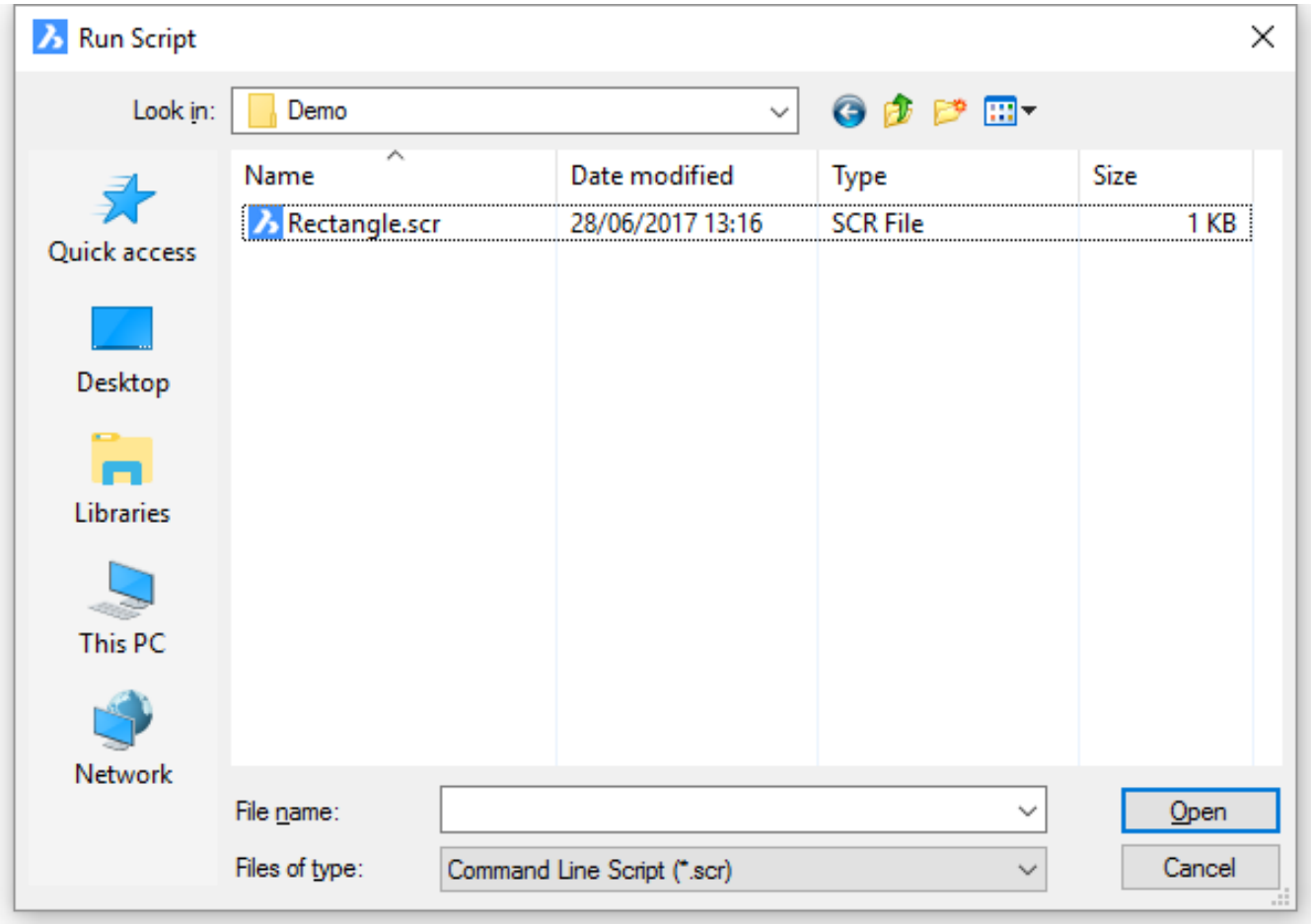
 command bar: **script** transparent: **'script** alias: **scr** menu bar: **Tolls | Run Script** ribbon:**Tools | Action Recorder | Run Script** toolbar: **Tools |** **: Script** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

1. Select an SCR file.
2. Click **Open**. The script begins running immediately.

- To rerun the script, use the RScript (6.26.36) command.
- To stop the script, press **Esc**.

For more information on writing scripts, see **Customizing BricsCAD**.

Figure 6.1979



Command Options

Option	Description
Look in	Chooses the drive and folder name.
File name	Specifies the name of the SCR file.
Open	Opens the SCR file, and then runs the script.

Related Commands

RecScript (6.26.5) - records scripts to SCR files. StopScript (6.27.48) - stops recording the script, and saves it to an SCR file. RScript (6.26.36) - reruns the script. Delay (6.10.23) - pauses the execution of the script for the specified number of milliseconds. Resume (6.26.28) - resumes a stopped script.

Figure 6.1980



6.27.10 ScrollBar

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the horizontal and vertical scroll bars for panning the drawing.

Accessing the Command

command bar: **scrollbar** transparent: **'scrollbar** shortcut:**Shift+F4** menu bar: **View | Scrollbars : ScrollBar Prompts you in the command bar:** WNDLSCRL is currently on [ON/OFf/Toggle] <On>:(**Enter an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/scrollbar-eg.png}) Scroll bars are useful for panning the drawing without using the Pan (6.24.2) command: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/
The only reason to turn off scroll bars is to gain a bit more drawing area.

Figure 6.1981

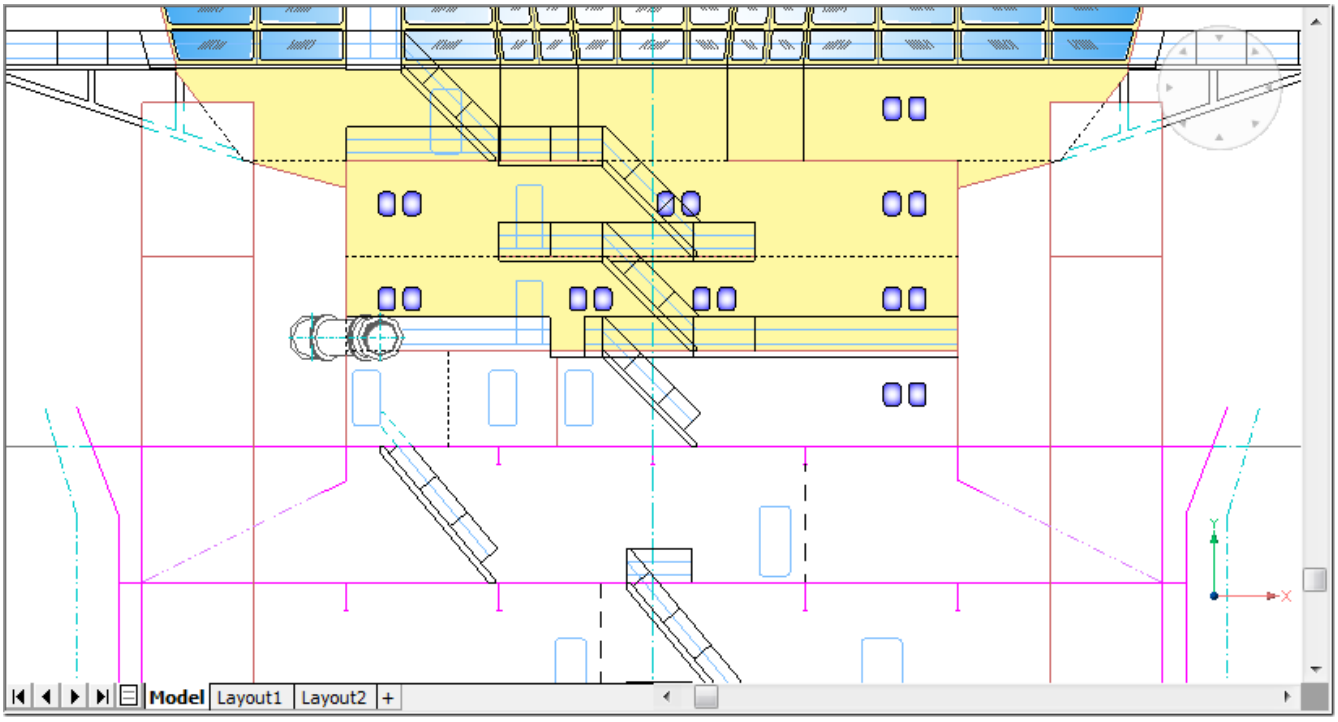


Figure 6.1982



Command Options

Option	Description
OFf	Turns off the scroll bars.
Toggle	Toggles the display of scroll bars: • If on, turns them off. • If off, turns them on.
OnN	Turns on the scroll bars.

Related Commands

CommandLine (6.9.34) - shows the command line. CommandLineHide (6.9.36) - hides the command line. GraphScr (6.15.20) - brings the graphics window to the foreground. StatBar (6.27.45) - toggles the display of the status bar. TextScr (6.29.13) - brings the text window to the foreground. Toolbar (6.29.18) and -Toolbar (6.29.18) - toggle the display of toolbars.

Figure 6.1983



6.27.11 Section

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Creates section planes of 3D solids, surfaces, polyface meshes, and 3D faces; the result is a slice made of region entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image4

Accessing the Command



command bar: **section** alias: **sec** menu bar: **Model | 3D solids| Section** toolbar: **3D Solids | Section : Section Prompts you in the command bar:** Select entities: **(Choose one or more 3D entities)** Entities in set: 1 Select entities: (**Choose more 3D entities, or else press Enter to continue.)** Specify first point on section plane or Object/Zaxis/View/XY/YZ/ZX: **(Enter an option.)** Specify a point on the current view plane <0,0,0>: **(Pick a point, or enter x,y,z coordinates)** Section made in the current view plane (left); section made of a region (right) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image420.jpg}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image730.png})

Figure 6.1984

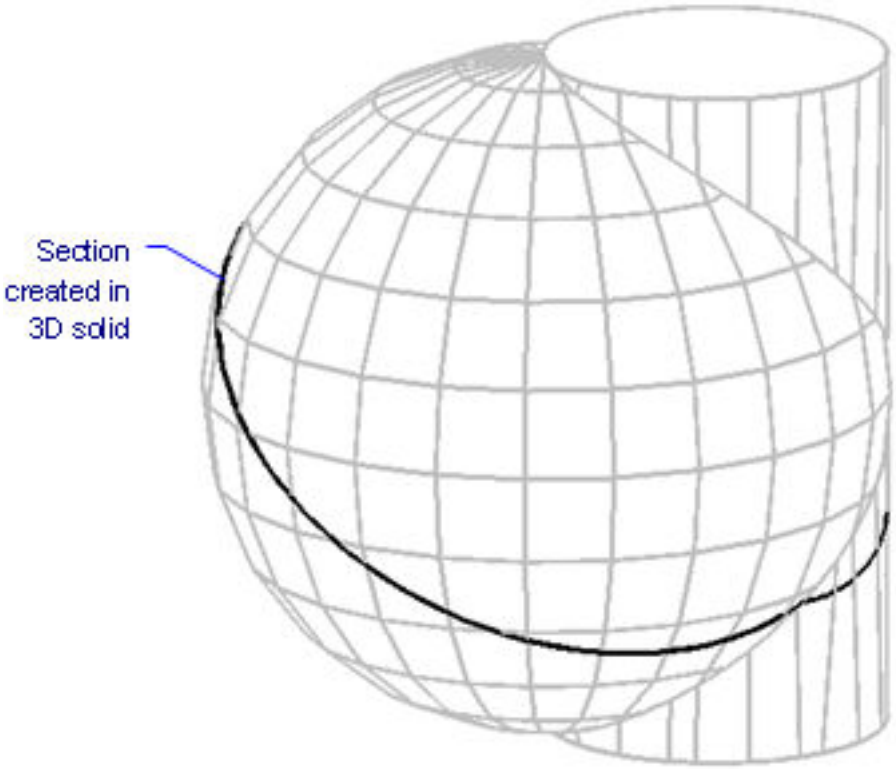
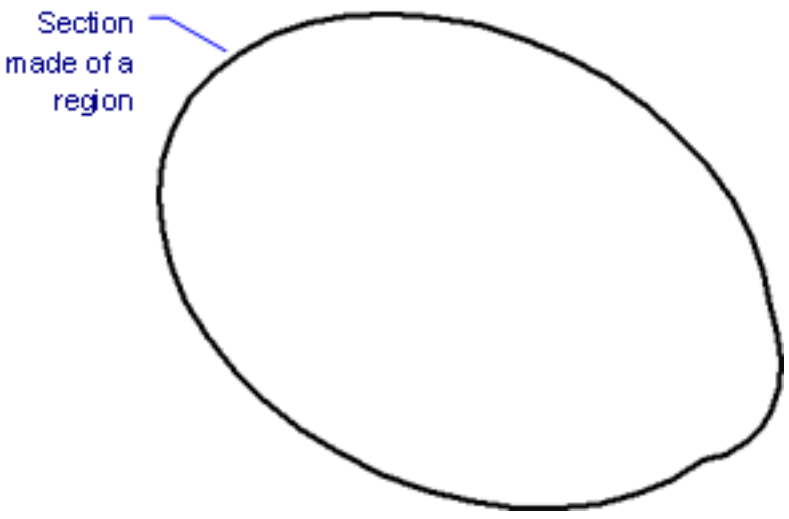


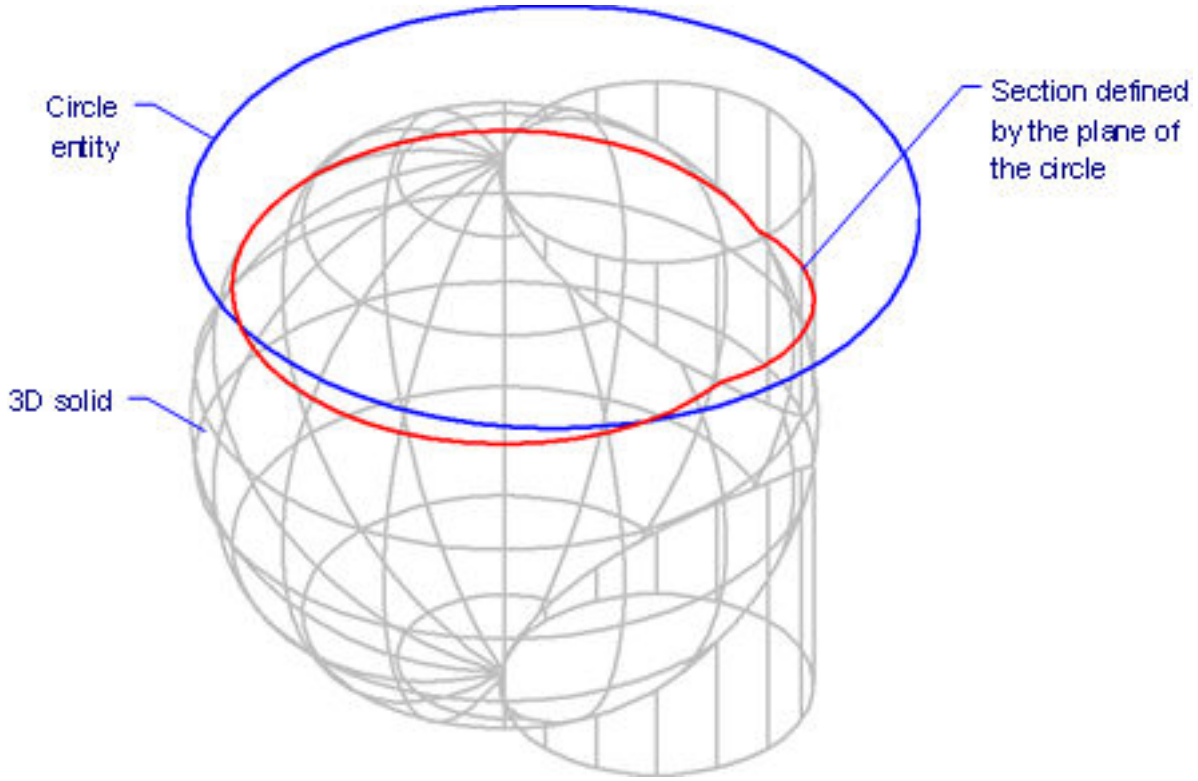
Figure 6.1985



Command Options

Option	Description
Select entities	Specifies one or more 3D entities from which the section is created: • 3D solids • 3D surfaces • 3D polyface meshes • 3D faces
Specify first point on section plane	Pick a point to specify the location of the plane in which the section lies; prompts you: 1. **Specify second point on plane** - **(pick another point.)** 2. **Specify third point on plane** - **(pick a third point, as three points define a plane.)**
Object	Creates the section from entities that intersect the 3D solid; prompts you: **Select a circle, ellipse, arc, 2D-spline, or 2D-polyline** - **(choose an entity; you can also use elliptical arcs.)** The entity defines the plane through which to create the section. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Zaxis	Creates a section in the plane defined by one point on the plane, and a second on the plane's normal (z axis); prompts you: 1. **Specify a point on the section plane** - **(pick a point to define the location of the section plane.)** 2. **Specify a point on the Z-axis (normal) of the plane** - **(pick another point to locate the normal, which determines the angle of the plane.)**
View	Creates a section in the plane of the current 3D viewpoint; prompts you: **Specify a point on the current view plane** - **(pick a point to define the height of the section.)** This is probably the easiest of the options.
XY	Creates a section parallel to the x,y plane; prompts you: **Specify a point on the XY-plane** - **(pick a point to define the height of the section above or below the x,y plane.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
YZ	Creates a section parallel to the y,z plane; prompts you: **Specify a point on the YZ-plane** - **(pick a point to define the height of the section above or below the y,z plane.)**
ZX	Creates a section parallel to the z,x plane; prompts you: **Specify a point on the ZX-plane** - **(pick a point to define the height of the section above or below the z,x plane.)**

Figure 6.1986



Related Commands

Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. Slice (6.27.36) - cuts 3D solids into two parts. Subtract (6.27.56) - removes one solid model from a second. Union (6.30.6) - joins two or more solid models into a single entity. ClipDisplay (6.9.24) - toggles the Clip Display property of a section plane or a BIM section entity. BimSection (6.7.27) - creates a BIM section entity. BimSectionOpen (6.7.28) - opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing. BimSectionUpdate (6.7.29) - exports or updates the result of a BIM section in a drawing. LiveSection (6.20.34) - switches the **Live Section** property of a section plane on or off. SectionPlane (6.27.12) - creates a section entity that creates sections of 3D solids, 3D Faces, and Surfaces. SectionPlaneSettings (6.27.13) - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog. SectionPlaneToBlock (6.27.14) - saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

Figure 6.1987

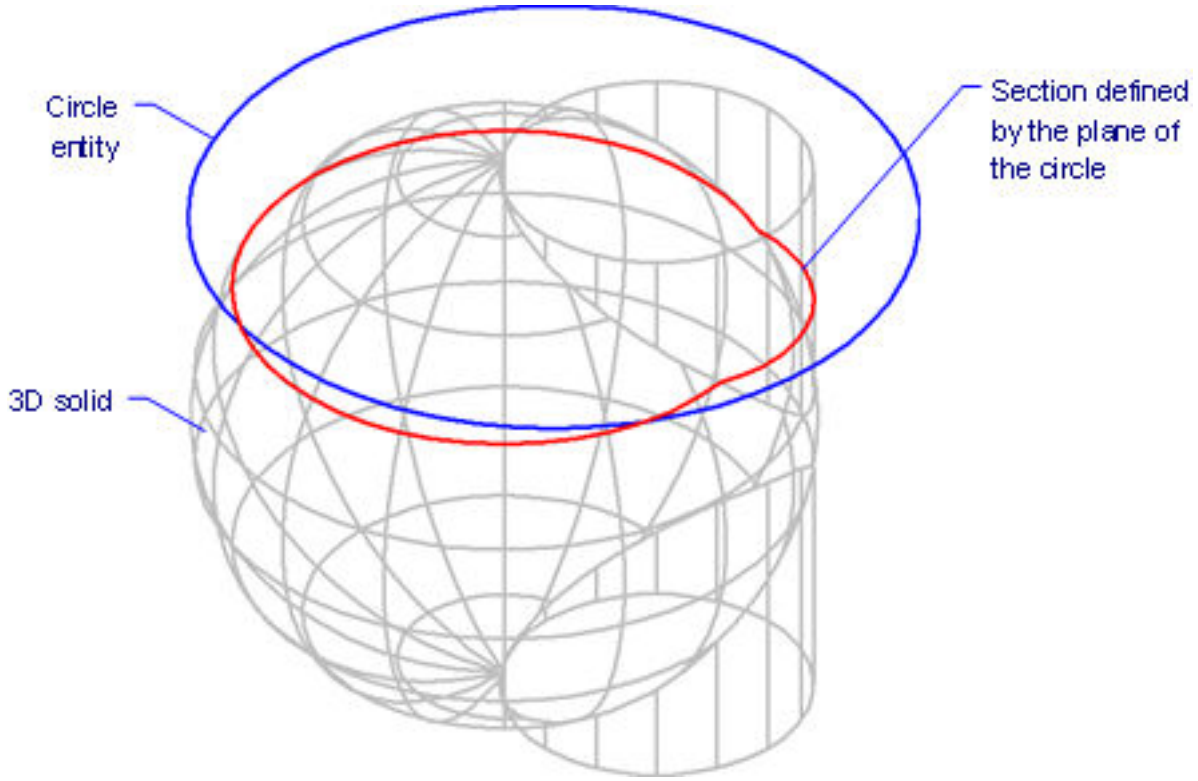


Figure 6.1988

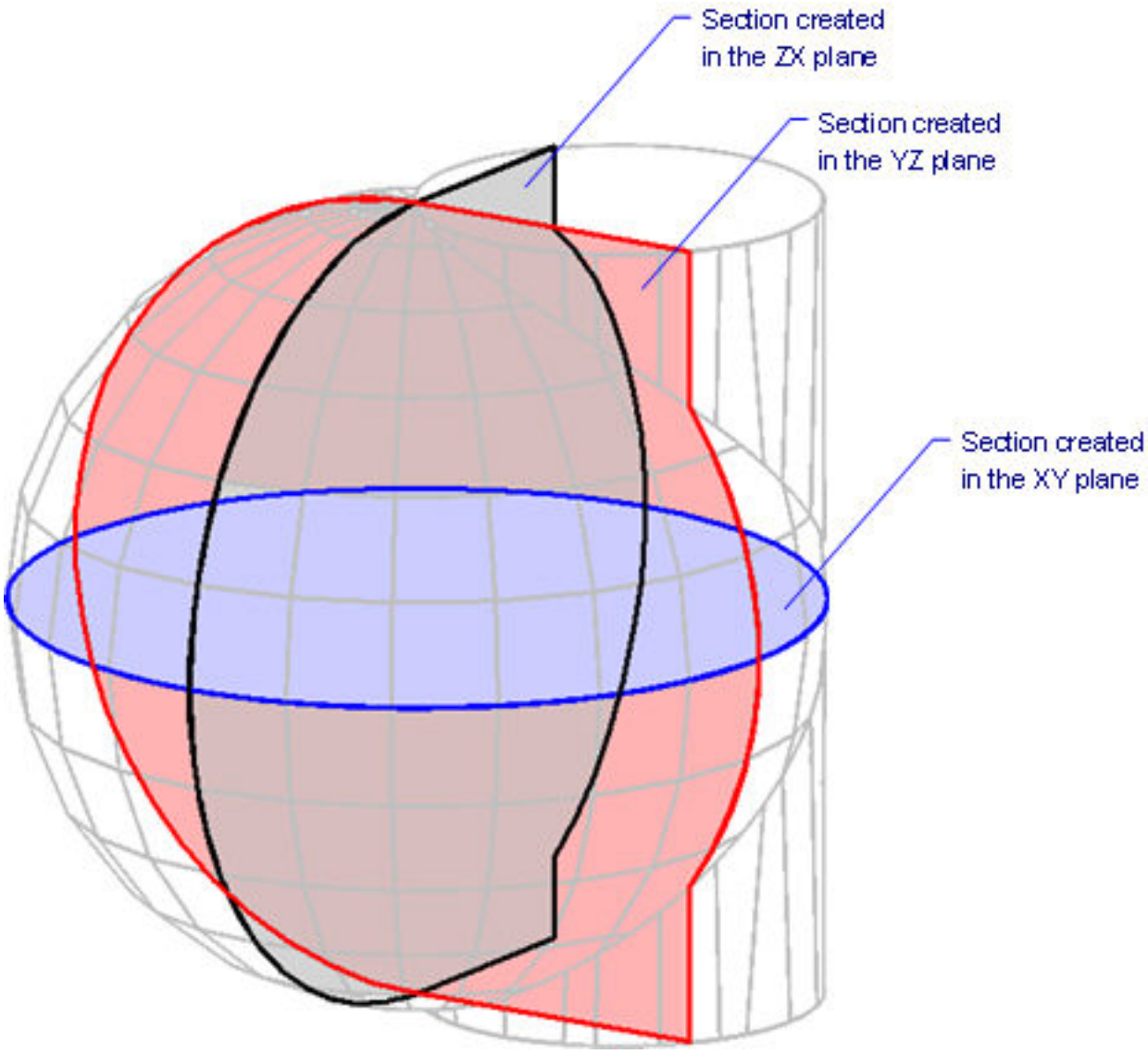
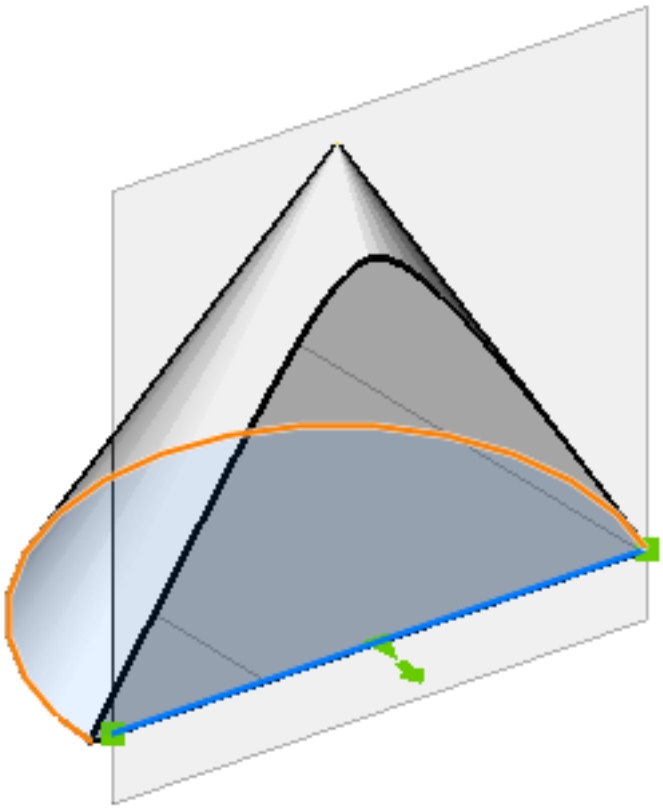


Figure 6.1989



Figure 6.1990



6.27.12 SectionPlane

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates section entities from 3D solids, 3D surfaces, polyface meshes, and 3D faces, so that you can see inside the 3D entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comma eg1.png})

Accessing the Command


command bar: **sectionplane** menu bar: **Model | Sections | Section Plane** ribbon:**Model | Sections | Section Plane** toolbar: **Sections |**  **: SectionPlane Prompts you in the command bar:** Specify start point or [select Face/Draw/Orthographic]: **(Specify the start point of the section plane)** Specify through point: **(Specify a second point)** Newly created section planes have their**Clip Display** status set to on. As sectionplanes are entities, they can be edited and deleted.

Command Options

Option	Description

Specify start point	Specifies the starting point of a 2D section plane; prompts you: **Specify through point** - **(pick a second point.)** As you move the cursor, the program ghosts the section plane: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/se startingpoint-eg.png}) Use entity snaps to make the pick points accurate.
select Face	Select the planar face of a 3D solid; this option does not work with non-planar faces, such as on spheres. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/se facet-eg.png}) Prompts you: **Select entity** - **(Choose a face.)** The section plane coincides with the selected face.The LiveSection (6.20.34) property is turned on automatically. Live sections are sections that can be interactively edited. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/se facet-eg2.png})
Draw	Define the section plane by two or more points; draws L and U-shaped sections planes. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/se draw-eg.png}) Prompts you: 1. **Specify start point** - **(pick the first point or enter x,y,z coordinates; use entity snaps to place the plane accurately.)** 2. **Specify next poin**t - **(pick another point.)** 3. **Specify next point or ENTER to finish** - **(pick another point, or else press Enter to stop specifying points.)** 4. **Specify the direction of the section entity** - **(pick a point to specify the view direction of the section plane; the side of the section plane you pick is the side that is not cut away.)** The section plane is created through the specified points and perpendicular to the current UCS.The Image below shows an L-shape section plane.
Orthographic	Places a section plane entity in one of the six standard drawing orientations, relative to the current UCS is created; prompts you: **Align section to [Front/bAck/Top/Bottom/Left/Right]: <Top>- (enter an option)** The program draws the orthographic section plane through the center of the bounding box that contains all 3D entities in the drawing. The figure shows the result of the Top section plane: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C The Live Section property is turned on automatically.

Procedures

- Align a section plane to a face (1.18.4)
- To create a vertical section plane (1.18.4)
- To create a horizontal section plane (1.18.4)
- To create a vertical jogged section plane (1.18.4)
- To create a horizontal jogged section plane (1.18.4)
- To create a orthographic section plane (1.18.4)

- To edit the properties of a section plane (1.18.4)
- To modify the size and position of a section plane (1.18.4)

Related Commands

ClipDisplay (6.9.24) - toggles the **Clip Display** property of a section plane or a BIM section entity. LiveSection (6.20.34) - switches the **Live Section** property of a section plane on or off. Section (6.27.11) - creates section planes of 3D solids, 3D Faces, and Surfaces; the results are region entities. SectionPlaneSettings (6.27.13) - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog. SectionPlaneToBlock (6.27.14) - saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block. BimSection (6.7.27) - creates a BIM section entity. BimSectionOpen (6.7.28) - opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing. BimSectionUpdate (6.7.29) - exports or updates the result of a BIM section in a drawing.

Figure 6.1991

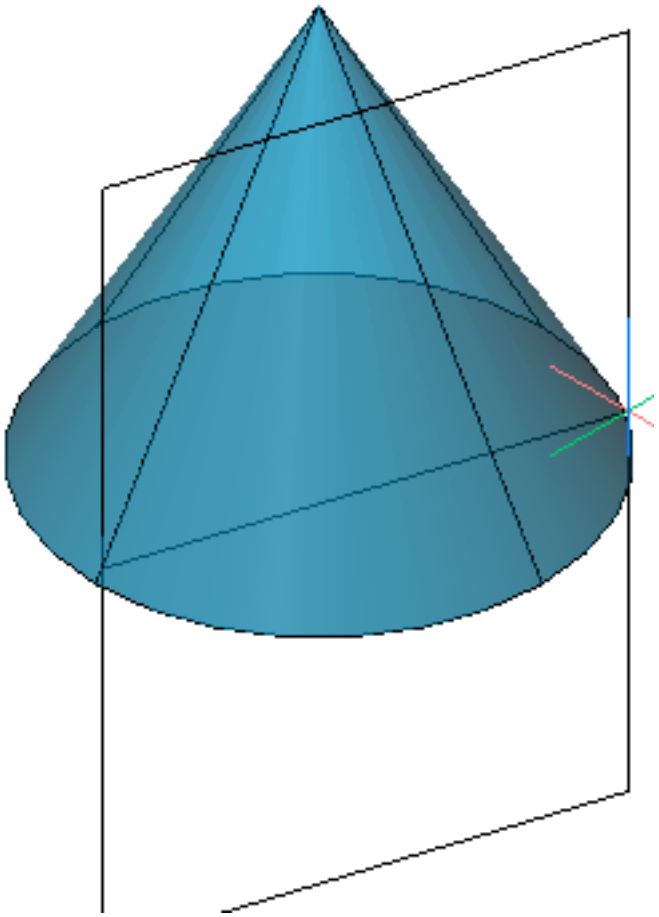


Figure 6.1992

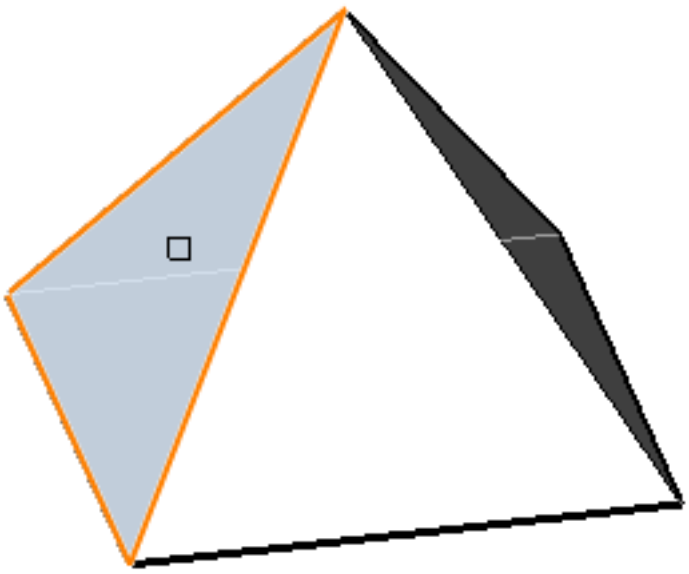


Figure 6.1993

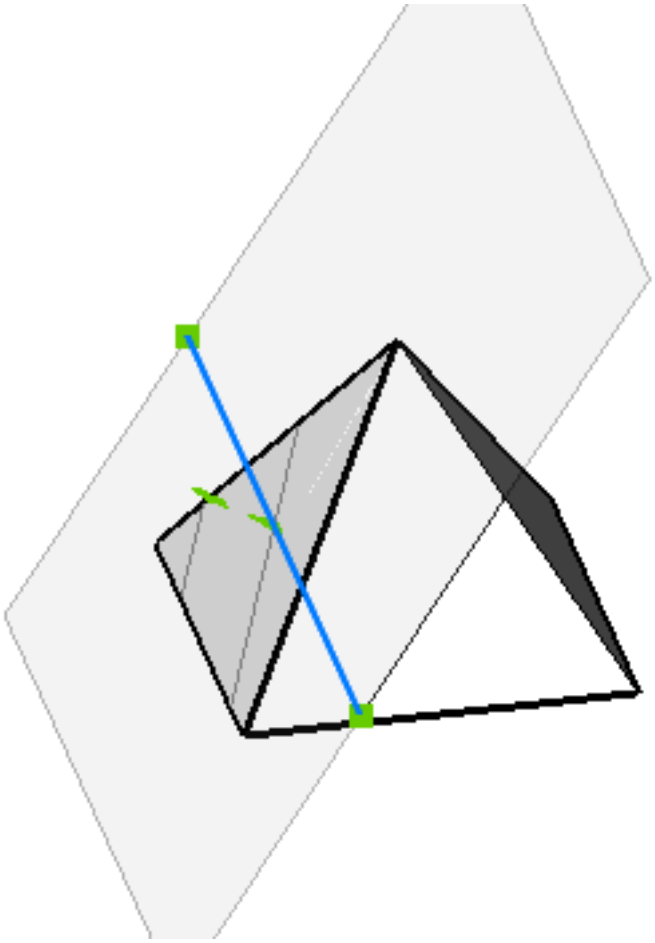


Figure 6.1994

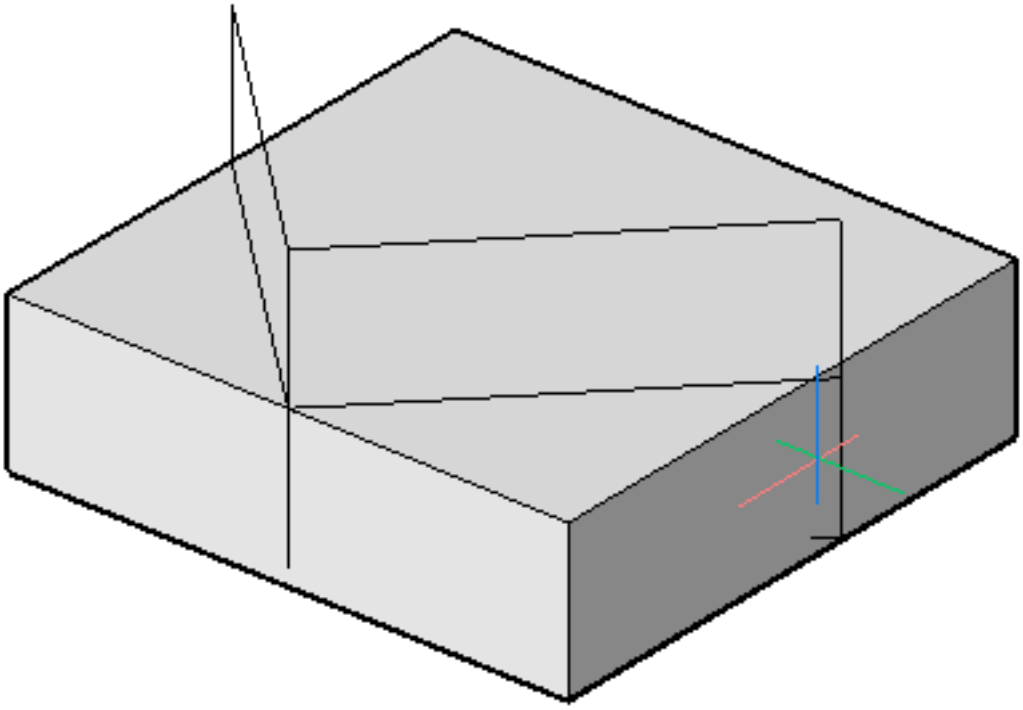


Figure 6.1995

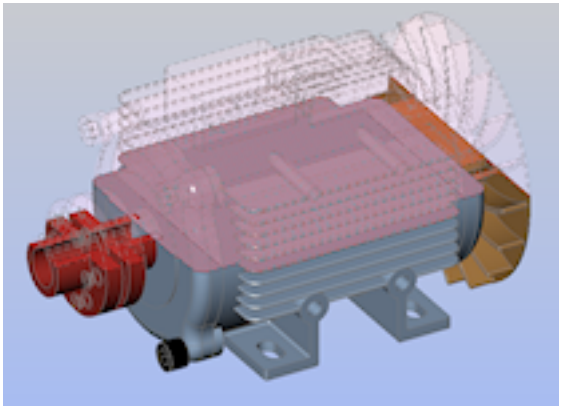


Figure 6.1996



6.27.13 SectionPlaneSettings

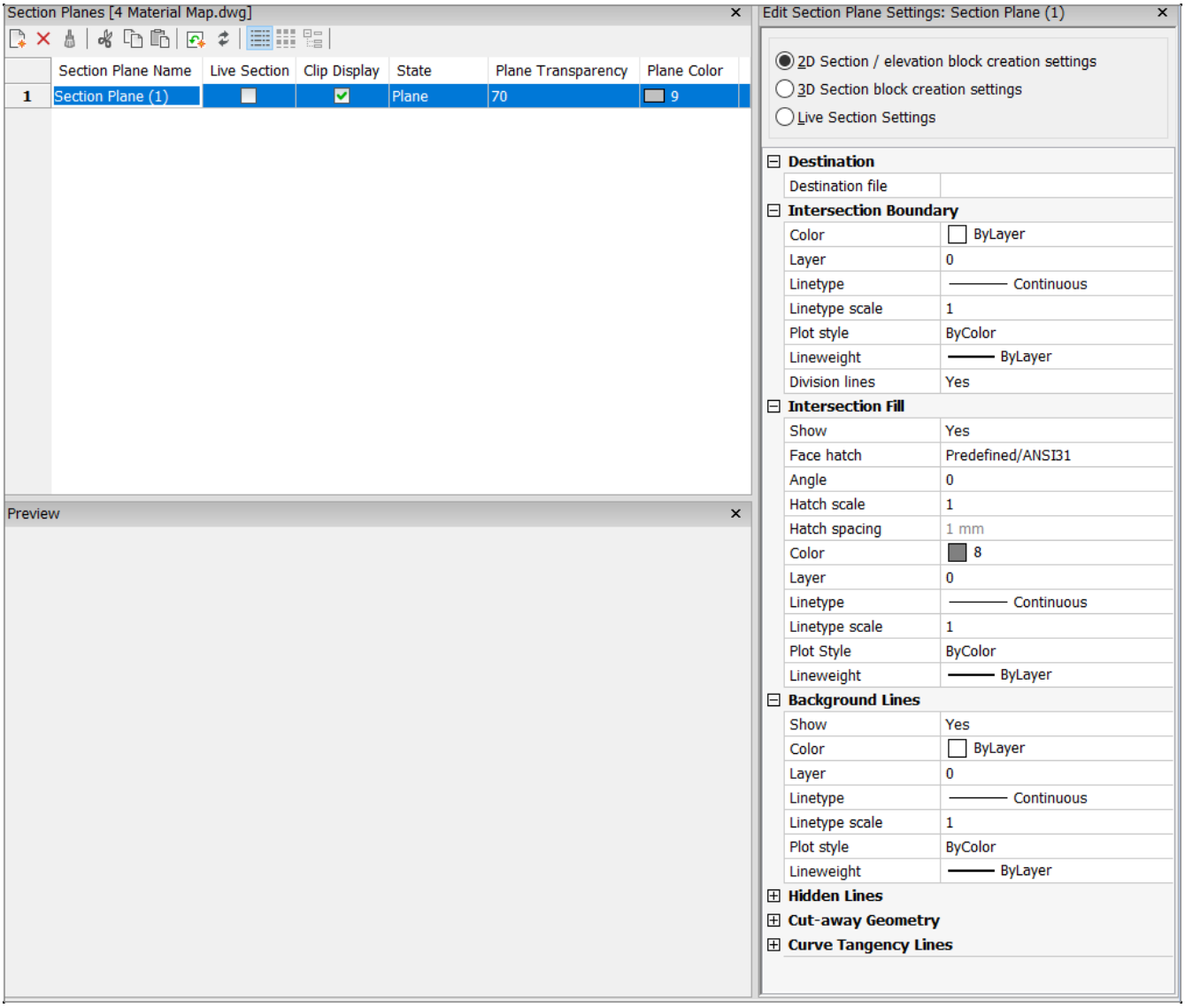
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and edits properties of named section planes, through a dialog box. **Note:** Section planes are created with the SectionPlane (6.27.12) command.

Accessing the Command



command bar: **sectionplanesettings** menu bar: **Tools | Drawing Explorer | Section Planes Model | Direct Modeling | Sections | Section Plane Settings** ribbon:**Home | Settings | Drawing Explorer | Section Planes** toolbar: **sections** | **: SectionPlaneSettings** Displays the Section Plane section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/sectionplanesettings-dg.png})

Figure 6.1997



Toolbar Options

Option	Description
New	Creates new sections planes; see the SectionPlane (6.27.12) command.
Delete	Deletes the currently selected section plane definition without warning

Header Options

Option	Description
Section Plane Name	Names the section plane; double click the field to rename it.
Live Section	Determines whether the section plane is of the live variety: • **On** - the section is a live one, and so can be edited interactively • **Off** - section plane cannot be edited directly
Clip Display	Determines whether the section plane clips the 3D model: • **On** - the model is clipped by the section plane • **Off** - the model is not clipped
State	Determines the state of the section plane: 1. Click the field 2. From the droplist, choose another state: • **Plane** - shows the cutting plane as aplane that extends infinitely in all directions. When cutting plane is covert toPlane state, the first segment of a jogged section plane entity defines the properties of the cutting plane. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S • **Boundary** - the cutting plane is a 2D box, indicating the extents of the cutting plane in the XY-plane.The cutting plane extends infinitely in the Z direction. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S • **Volume**- the cutting plane is a 3D box, indicating the 3D extents of the cutting plane, display (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

Plane Transparency	Specifies the level of transparency of the cutting plane: 1. Click the field 2. Enter a different level of transparency, from 0 (opaque) to 90 (nearly transparent)
Plane Color	Sets the color of the cutting plane in the drawing. 1. Click the field 2. In the Select Color dialog box, choose another color 3. Click **OK** See the Color (6.9.33) command.

Edit Panel Options

When you select two or more section plane names, you can edit their properties simultaneously in this panel

Edit Panel Option	**Description**
Type	Specifies the type of section plane: • **2D Section / elevation block creation settings** - specifies properties for 2D sections and elevations • **3D Section block creation settings** -specifies properties for 3D sections • **Live Section Settings** -specifies properties for live sections A different set of properties are available for each type of section plane, and each type can look different in drawings.
2D Section / elevation block creation settings	Sets the properties of 2D sections and elevations (used by the BIM module): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/2
3D Section block creation settings	Sets the properties of 3D sections: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3
Live section settings	Sets the properties of live sections: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/l

Related Commands

LiveSection (6.20.34) - switches the **Live Section** property of a section plane on or off. Section (6.27.11) - creates section planes of 3D solids; the results are region entities. SectionPlane (6.27.12) - creates a section entity that creates sections of 3D solids. SectionPlaneToBlock (6.27.14) - saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.1998

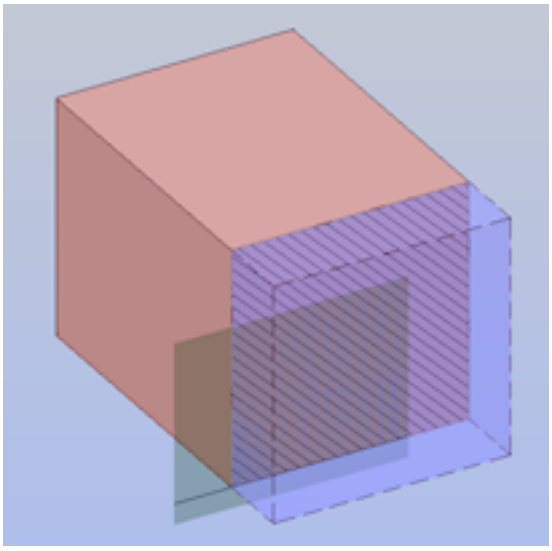


Figure 6.1999

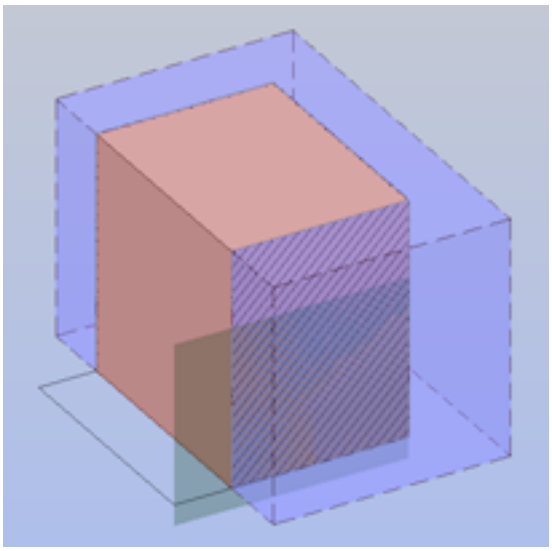


Figure 6.2000

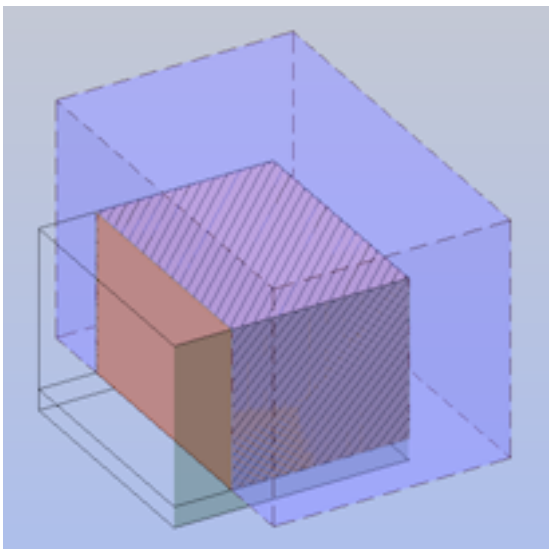


Figure 6.2001

Intersection Boundary	
Color	ByLayer
Layer	0
Linetype	Continuous
Linetype Scale	1
Plot Style	ByColor
Lineweight	ByLayer
Division Lines	Yes
Intersection Fill	
Show	Yes
Face Hatch	Predefined/SOLID
Angle	0.0000
Hatch Scale	1
Hatch Spacing	1.0
Color	8
Layer	0
Linetype	Continuous
Linetype Scale	1
Plot Style	ByColor
Lineweight	ByLayer
Background Lines	
Show	Yes
Hidden Line	Yes
Color	ByLayer
Layer	0
Linetype	Continuous
Linetype Scale	1
Plot Style	ByColor
Lineweight	ByLayer
Cut-away Geometry	
Show	No
Hidden Line	Yes
Color	Red
Layer	0
Linetype	DASHED
Linetype Scale	1
Plot Style	ByColor
Lineweight	ByLayer
Curve Tangency Lines	
Show	Yes
Color	Red
Layer	0
Linetype	DASHED
Linetype Scale	1
Plot Style	ByColor
Lineweight	ByLayer

Figure 6.2002

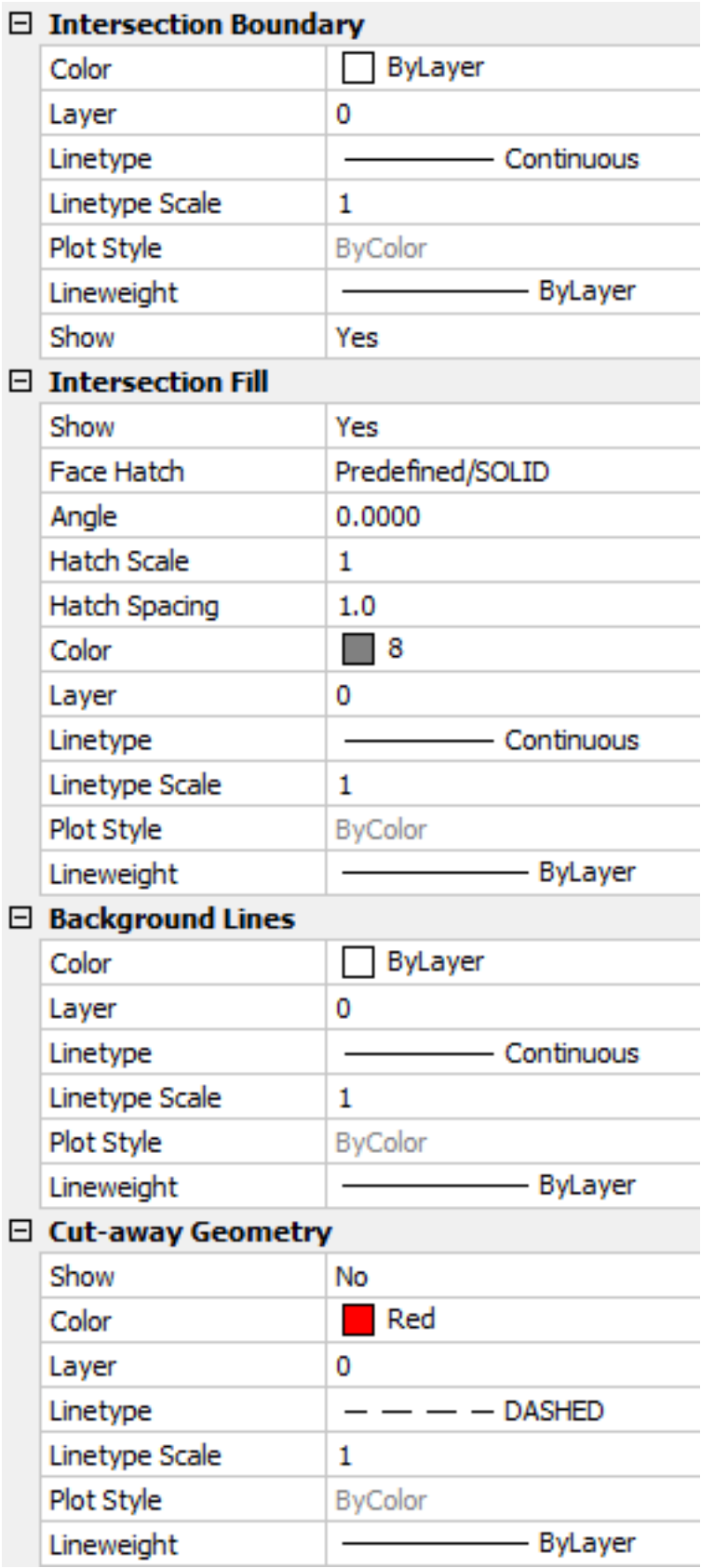


Figure 6.2003

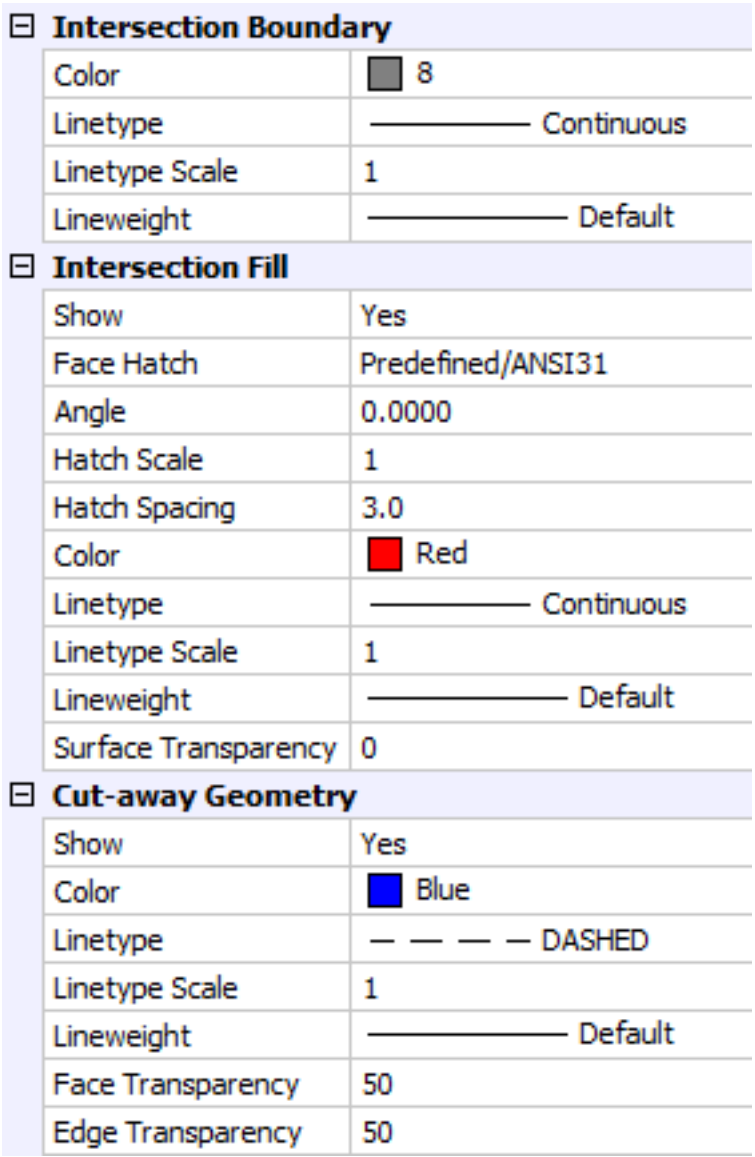
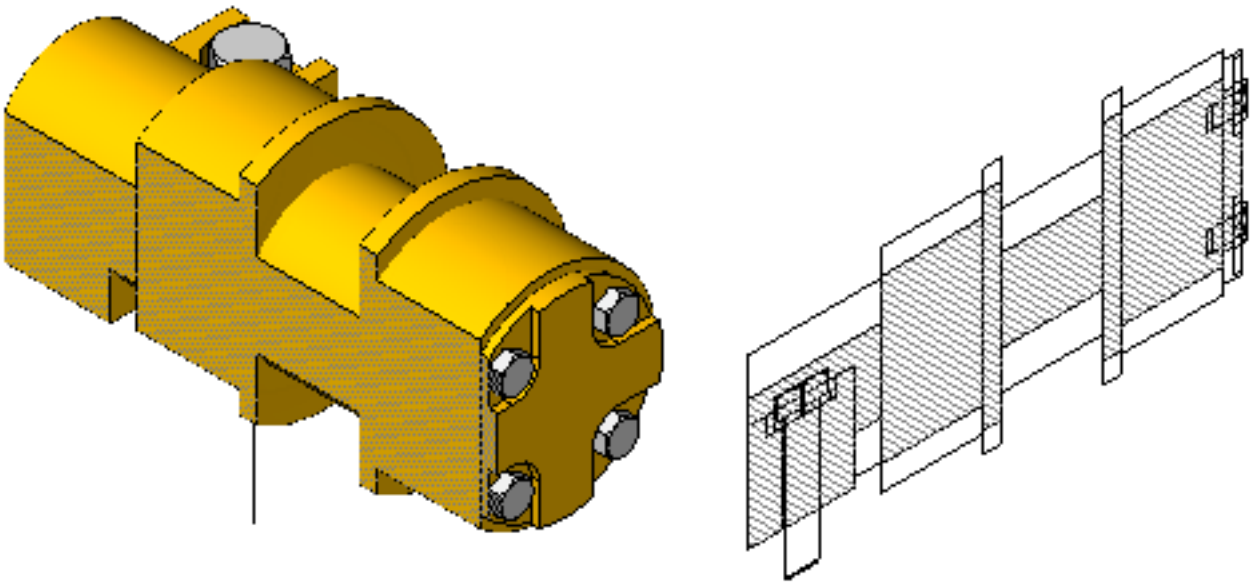


Figure 6.2004



Figure 6.2005



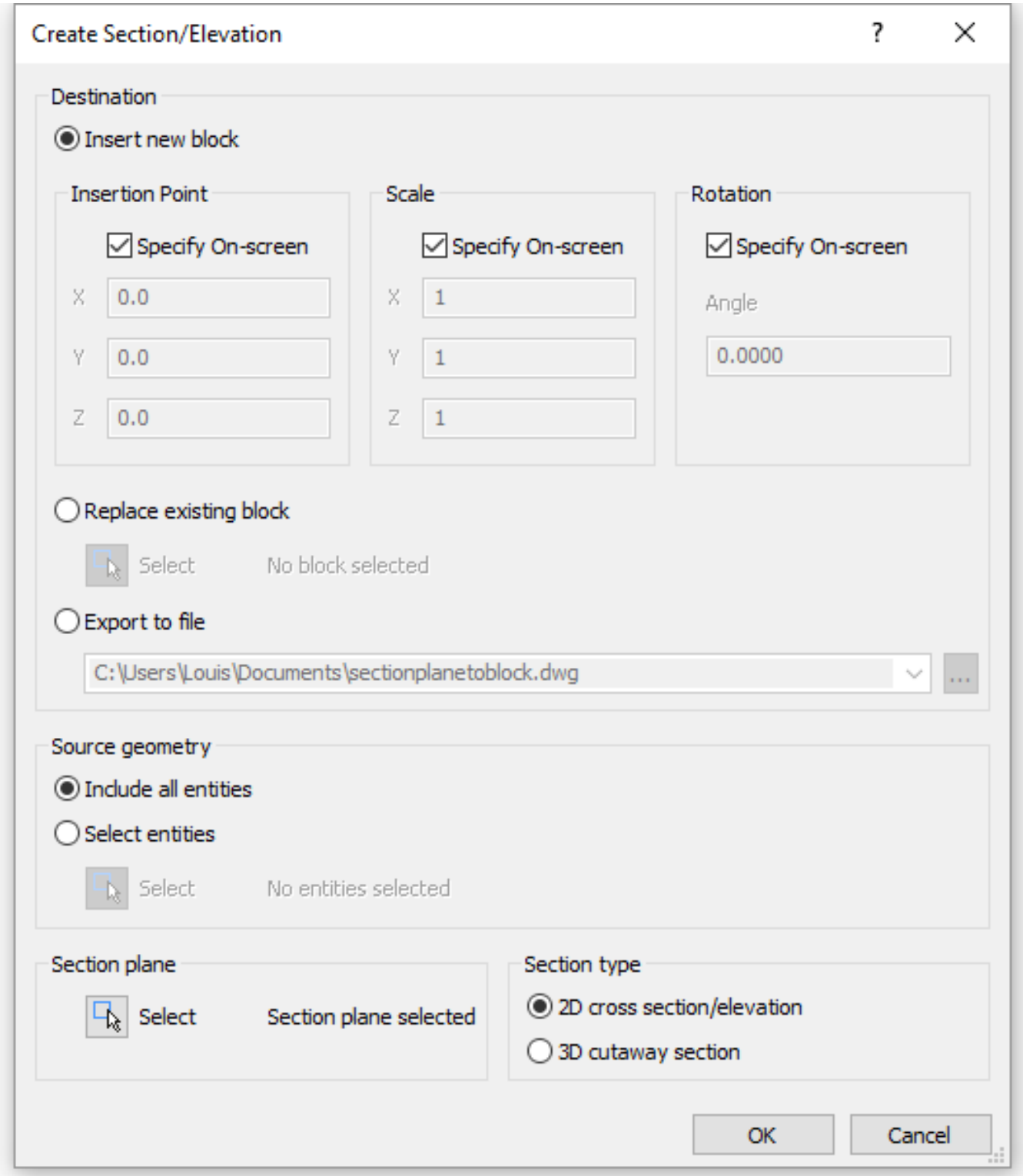
6.27.14 SectionPlaneToBlock

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Saves section planes as blocks inserted in the current drawing (see below, at right) or saved to disk as DWG files. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/sectionpeg.png}) **Note:** Section planes are created with the SectionPlane (6.27.12) command.

Accessing the Command

command bar: **sectionplanetoblock** menu bar: **Model | Sections | Section plane to block** ribbon: **Model | Sections | Section Plane | Section Plane to Block** toolbar: **Sections | : SectionPlaneToBlock** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SectionPlaneToBlockDialog.pn

Figure 6.2006



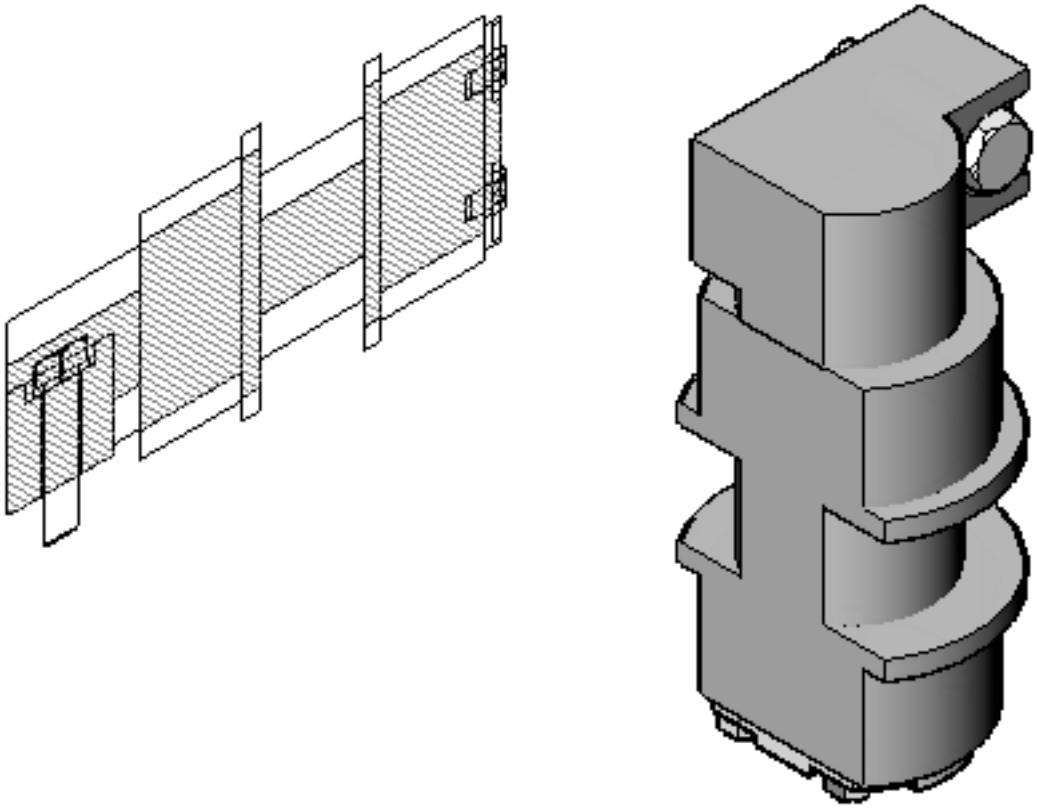
Dialog Box Options

Option	Description
Destination	Defines where the block will be placed: **Insert New Block** - converts the section plane geometry as a block, and then inserts it into the current drawing: • **Insertion point** - specify the insertion point after you dismiss this dialog box, or else enter the insertion point coordinates as X, Y, and Z • **Scale** -specify the scale factor after you dismiss this dialog box, or else enter individual scale factors for the X, Y, and Z directions • **Rotation** -specify the rotation angle after you dismiss this dialog box, or else enter an angle now **Replace Existing Block** - replaces a block that already sits in the drawing: • **Select** - selects the block to replace; use this option to update changed sections **Export to File** - saves the block as a file: • **Browse** - choose the folder in which to save the block, and whether to save the file in DWG or DXF format
Source geometry	Determines how many of the entities in the drawing will be included: **Include all entities** - all 3D geometry in the drawing is processed; this command works with3D solids, 3D surfaces, polyface meshes, and 3D faces. **Select entities** - chooses the entities to include: • Select( ) prompts you to select entities; **(press Enter to return to this dialog box.)**
Section plane	Selects the section plane to be converted to a block: **Select**) - prompts you to select entities; **(press Enter to return to this dialog box.)**
Section type	Determines the kind of section plane block to create; shown below is a 2D cross section (left) and 3D cross section (right) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/se 2d3d-eg.png}) **2D cross section/elevation** - block is created used the 2D section settings defined by the SectionPlaneSettings (6.27.13) dialog box **3D cutaway section** -block is created used the 3D settings defined by the SectionPlaneSettings (6.27.13) dialog box
OK	Closes the dialog box; when this command is creating a block, you are prompted to specify its insertion point, scale factor(s), and rotation angle.

Procedures

- Working with sections (1.18.4)
- To export a section to a file (1.18.4)

Figure 6.2007



Related Commands

LiveSection (6.20.34) - switches the **Live Section** property of a section plane on or off. Section (6.27.11) - creates section planes of 3D solids, 3D Faces, and Surfaces; the results are region entities. SectionPlane (6.27.12) - creates a section entity that creates sections of 3D solids, 3D Faces, and Surfaces. SectionPlaneSettings (6.27.13) - defines the properties of a section plane entity in the Drawing Explorer - Section Planes dialog.

Figure 6.2008



6.27.15 SecurityOptions

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Sets a password to protect your drawing from unauthorized access. **Note:** If you forget the password, you can no longer open the drawing.

Accessing the Command

command bar: **securityoptions** menu bar: **File | Drawing Utilities | Security Options : SecurityOptions** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SecurityOptionsDialog.png

1. Enter a password, and then click **OK**.A dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e

2. Confirm the password, and then click **OK**.

Figure 6.2009

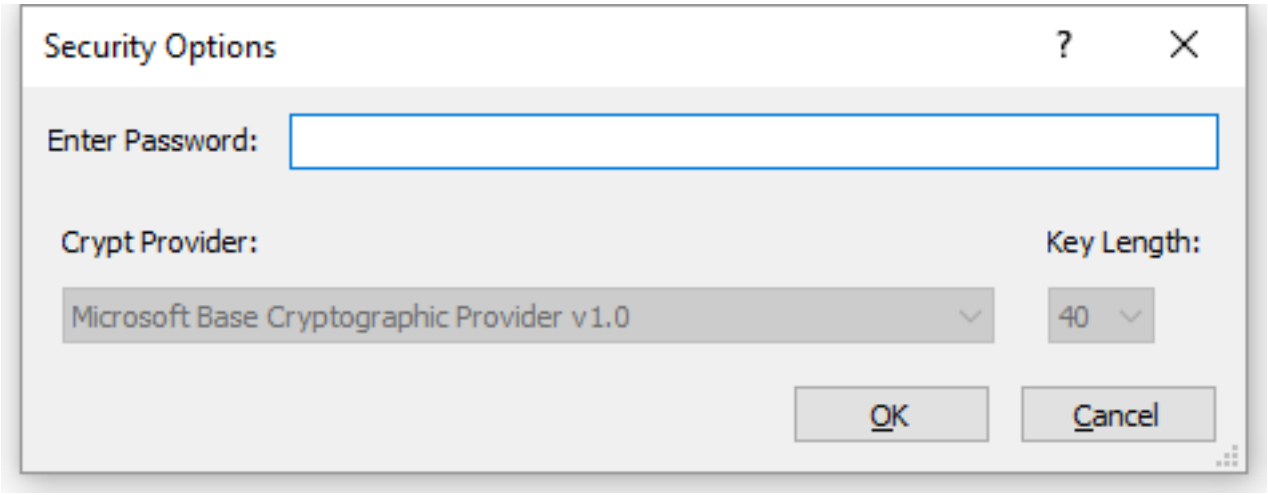
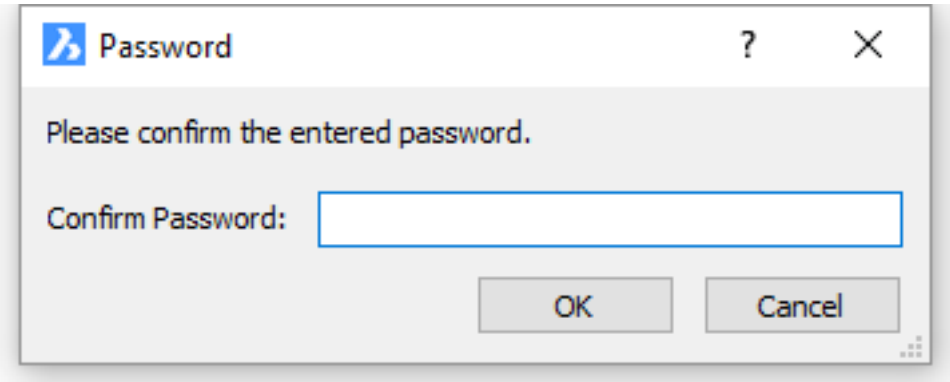


Figure 6.2010



Dialog Box Options

Option	Description
Enter password	Enter one or more characters; you cannot see what you are writing, unfortunately, as the characters are turned into dots.
Crypt Provider	Specifies the encryption level for **drawing properties**. Choose an encryption provider supplied the operating system. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C
Key Length	Select a length for the encryption key: The longer the key, the higher the level of protection is, because it takes longer to break the encrypted password.
Confirm Password	Enter the same password, and then click **OK**.

Procedures

- To set the password (1.14.5)
- To remove the password (1.14.5)

Related Commands

Dwgprops (6.10.42): opens the Drawing Properties dialog box QSave (6.25.4) - saves the drawing without displaying the Save dialog box. Save (6.27.1) - saves the current drawing through a dialog box. SaveAll (6.27.2) - saves all open drawings. Open (6.23.8): opens a drawing WhoHas (6.33.8) - reports the ownership of a drawing file.

Figure 6.2011

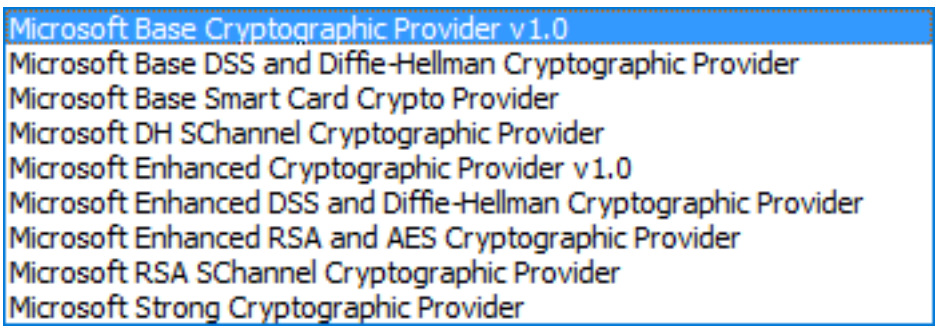


Figure 6.2012



6.27.16 SelectAlignedFaces

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Selects all faces of a 3D model that are in the same plane (coplanar) with a selected face.

Accessing the Command

 command bar: **selectalignedfaces** Quad cursor: **Solid/Face Selection**  **: SelectAlignedFaces Prompts you in the command bar:** Select face: **(Click a face in the model)** The program selects all other faces that are in the same plane, shown outlined in blue in the figure below **(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18 (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/SelectAlignedFaces.png})**

Figure 6.2013

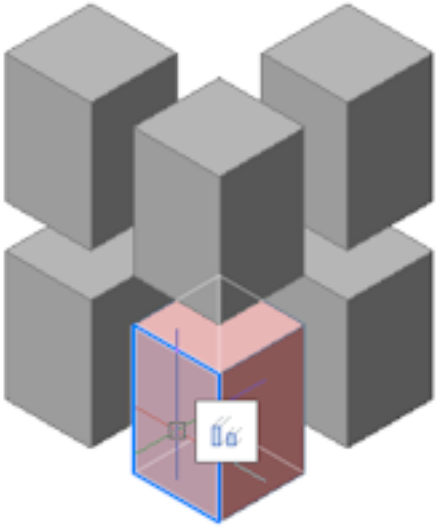
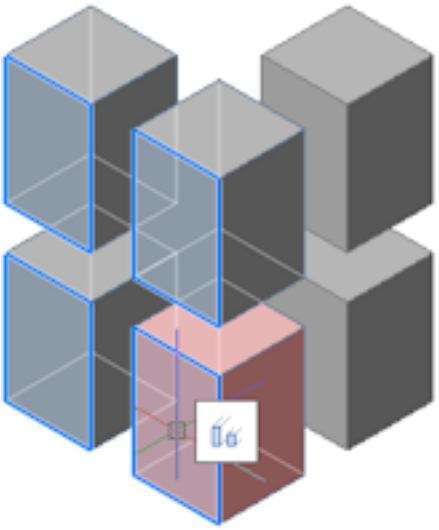


Figure 6.2014



Command Options

Option	Description
Select face	Select a single face for the program to match to other coplanar faces

Related Commands

BimMultiSelect (6.7.16) - selects linear solids with coplanar and/or parallel axes. dmSelect (6.12.26) - selects edges and faces of 3D solids or surfaces based on their geometric properties. SelectAlignedSolids (6.27.18) - selects all solids in a model of which a face is coplanar with a selected face. SelectConnectedFaces (6.27.19) - selects all faces in a model which are connected to a selected face. SelectConnectedSolids (6.27.20) - selects all solids in a model which are connected to a selected face.

Figure 6.2015



6.27.17 Select

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Selects one or more entities through a variety of selection methods **Note:** You access the selection set during subsequent commands with the Previous option.

Accessing the Command



command bar: **select** shortcut: **Ctrl+A** selects all entities on non-frozen layers ribbon: **Home | Settings | Select ：Select Prompts you in the command bar:** Select entities to include in set: **(Choose one or more entities)** Entities in set: 1 Select entities to include in set: **(Choose more entities, or else press Enter to end the command)** To see a list of the selection modes, press ? at the prompt: Select entities to include in set: **(Press ?)** Select entities: ALL/ Add/ +/ Remove/ -/ Previous/ Last/ Window/ Crossing/ Outside/ WPolygon/ CPolygon/ OPolygon/ WCircle/ CCircle/ OCircle/ Box/ POint/ Fence/ AUto/ Multiple/ Single/ PROperties/ Dialog/ Undo/ Group: **(Enter an option)**

Command Options

Option	Description
Select entities to include in set	Chooses entities to add to the selection set;use one of methods listed below. As you add entities to the selection set, the program highlights them with dashes, as shown below: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/seleg.png})
ALL	Selects all entities in the drawing, except for those on frozen layers.
Add **and** +	(Used after the Remove option.) Adds additional entities to the selection set; prompts you: 1. **Select entities to include in set** - **(add one or more entities to the selection set.)** 2. **Select entities to include in set** - **(enter an option:)** • Add more entities to the selection set • Enter another selection option, such as Remove • Press **Enter** to end the command

Remove **and** -

Removes entities from the selection set; as you remove entities from the selection set, the program unhighlights them Prompts you:

1. **Subtract entities from selection set**- **(select one or more entities to remove from the selection set.)**
2. **Subtract entities from selection set** - **(enter an option:)**

   - Select more entities to remove from the selection set
   - Enter another selection option, such as Add
   - Press **Enter** to end the command

Previous

Add the entities from the previous selection to the current select set, and then continues with the 'Select entities to include in set' prompt.

Last

Adds the last entity drawn to the selection set, and then continues with the 'Select entities to include in set' prompt.

Window

Selects all entities that are entirely inside a rectangular selection window; prompts you:

1. **First corner of inclusive window** - **(specify the first point of the selection rectangle.)**
2. As you move the cursor, the program draws a rectangle filled with transparent blue
3. **Opposite corner** - **(specify the other corner)**

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
All entities fully within the rectangle are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.

Crossing

Selects all entities that are inside of and cross a rectangular selection window; prompts you:

1. **First corner of crossing window** - **(specify the first point of the selection rectangle with a pick point or x,y coordinates.)**
2. As you move the cursor, the program draws a dashed rectangle filled with transparent green
3. **Opposite corner** - **(specify the other corner.)**

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
All entities inside the rectangle and crossing it are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.

Outside	Selects all entities fully outside a rectangular selection window; prompts you: 1. **First corner of crossing window** - **(specify the first point of the selection rectangle with a pick point or x,y coordinates.)** 2. As you move the cursor, the program draws a dashed rectangle 3. **Opposite corner** - **(specify the other corner.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i All entities fully outside the rectangle are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.
WPolygon (Windowed polygon)	Selects all entities fully inside a polygonal selection window; prompts you: 1. **First vertex of inclusive polygon** - **(specify the first point of the selection polygon.)** 2. **Next vertex** - **(specify the next point.)** 3. As you move the cursor, the program draws the outline of a polygon filled with transparent blue 4. **Next vertex** - **(pick another point, or else press Enter to close the polygon.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i All entities fully within the polygon are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.
CPolygon (Crossing polygon)	Selects all entities inside and crossing a polygonal selection window; prompts you: 1. **First vertex of crossing polygon** - **(specify the first point of the selection polygon.)** 2. **Next vertex** - **(specify the next point.)** 3. As you move the cursor, the program draws the outline of a dashed polygon filled with transparent green 4. **Next vertex** - **(pick another point, or else press Enter to close the polygon.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i All entities inside the polygon and crossing it are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.

OPolygon (Outside polygon)	Selects all entities fully outside a polygonal selection window; prompts you: 1. **First vertex of outside polygon** - **(specify the first point of the selection polygon.)** 2. **Next vertex** - **(specify the next point.)** 3. As you move the cursor, the program draws the outline of a polygon 4. **Next vertex** - **(pick another point, or else press Enter to close the polygon.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i All entities fully outside the polygon are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.
WCircle (Windowed circle)	Selects all entities fully inside a circular selection window; prompts you: 1. **Center of circle** - **(pick a center point or specify x,y coordinates.)** 2. As you move the cursor, the program draws the outline of a circle filled with transparent blue 3. **Radius of circle** - **(specify a radius by picking or entering a value.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i All entities fully within the circle are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.
CCircle (Crossing circle)	Selects all entities inside and crossing a circular selection window; prompts you: 1. **Center of circle** - **(pick a center point or specify x,y coordinates.)** 2. As you move the cursor, the program draws the outline of a dashed circle filled with transparent green 3. **Radius of circle** - **(specify a radius by picking or entering a value.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i All entities inside the circle and crossing it are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.

OCircle (Outside circle)

Selects all entities fully outside a circular selection window; prompts you:

1. **Center of circle** - **(pick a center point or specify x,y coordinates.)**
2. As you move the cursor, the program draws the outline of a circle
3. **Radius of circle** - **(specify a radius by picking a point, or entering a value.)**

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
All entities fully outside the circle are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.

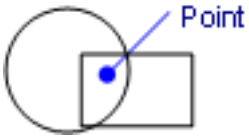
Box

Selects entities depending on the direction you move the cursor:

- Right to left - Box acts like Crossing mode; see Crossing mode
- Left to right - Box acts like Window mode; see Window mode

POint

Selects all closed entities surrounding the pick point, such as all walls of a room; prompts you: **Point to locate entity around** - **(specify a point inside closed**



objects.) All closed entities outside the pick point are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.

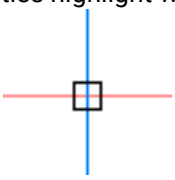
Fence

Selects all entities that cross the fence selection lines; prompts you:

1. **First point of fence**- **(specify the first point of the selection line.)**
2. **Next vertex** - **(specify the next point.)**
3. As you move the cursor, the program draws a dashed polyline
4. **Next vertex** - **(pick another point, or else press Enter to close the polygon.)**

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
All entities crossing the fence line are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.

AUto	Selects entities depending on how you use the cursor: • Pick an entity - the entity is selected • Pick two points right to left - Auto acts like Crossing mode; see Crossing mode • Pick two points left to right - Auto acts like Window mode; see Window mode All entities selected by one of these three modes are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.
Multiple	Should select entities **without** highlighting them during this selection mode in order to speed up entity selection in complex drawings, but entities are nevertheless highlighted by the program. All entities selected during this mode are added to the selection set; the commandcontinues with the 'Select entities to include in set' prompt.
Single	After you select a single entity, the command ends; prompt you: 1. **Select entities to include in set** - **(presss.)** 2. **Select entities to include in set** - **(pick a single entity, and the command ends.)**
Previous	Reuses the entities selected with the previous selection.
Properties	Selects entities based on their properties; prompts you: **Select by properties** - **(enter an option name.)** In some cases, you can enter * to specify all entities with the property; separate more than one property value with a comma. • **Color** - prompts you to enter a color name or number • **LAyer**- prompts you to enter one or more layer names, or * for all • **LType** - prompts you to enter one or ore linetype names, or * for all • **Name** - prompts you to enter one or more entity names, or * for all • **THickness** - prompts you to enter a value for the thickness • **TYpe**- prompts you to enter one or more entity types (such as line, circle, arc, and so on), or * for all • **Value** - prompts you to enter one or more values, or * for all • **Width**- prompts you to enter a value for the width of entities • **Handle**- prompts you to enter a value of a handle, a hexadecimal number that uniquely identifies each entity in the drawings; the List command reports handle numbers • **LOcation** - returns to the'Select entities to include in set' prompt • **Dialog** - displays the Entity Selection section of the Settings (6.27.23) dialog box For the panel version of this option, see the QSelect (6.25.5)command.

Dialog	Displays the Entity Selection section of the Settings (6.27.23) dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/sel sttings-dlg.png}) **PICKAUTO** - controls automatic inside or crossing window selection: • **1** - entity under the cursor is selected • **2**- start window selection even when the cursor is over an entity **PICKBOX** - sets the size in pixels of the small square attached to the crosshairs; entities highlight when  when captured by the pickbox. **PICKDRAG** - controls the method for window selection: • **0** - click to start the window, then pick a second point • **1** - press and hold the left mouse button, then move the mouse and release the mouse button **PICKFIRST** - controls whether commands accept an existing selection set. • **0** - existing selection sets are ignored when a command is launched • **1** - commands which prompt to select entities can use an existing selection set
Undo	Undoes the last action during this command.
Group	Add a group to the selection set; prompts you: **Enter group name** - (type the name of a group; see the Group (6.15.22) command.)

NOTE: In macros, the **^S** metacharacter selects the entity under the cursor automatically.

Procedures

- Selection and grip settings (1.23.5)
- Composing a selection set before (pre-pick) (1.23.6)
- Composing a selection set afterwards (post-pick) (1.23.6)
- Selecting overlapping entities (1.23.6)
- To create a selection set using QuickSelect (1.23.7)

Related Commands

DdSelect (6.10.20) - sets selection properties in the **Entity Selection** section of the **Settings** dialog box. SelectSimilar (6.27.21) - selects all entities of the same type as the current selection, based on a set of matching properties, stored in the **SELECTSIMILARMODE** system variable. dmSelect (6.12.26) - selects edges and faces of 3D solids or surfaces based on their geometric properties. SelGrips (6.27.22) - selects entities and displays grips. QSelect (6.25.5) - composes a selection set using filters.

Tutorial: Use Quick Select

Click here (1.4.7) to watch

Figure 6.2016

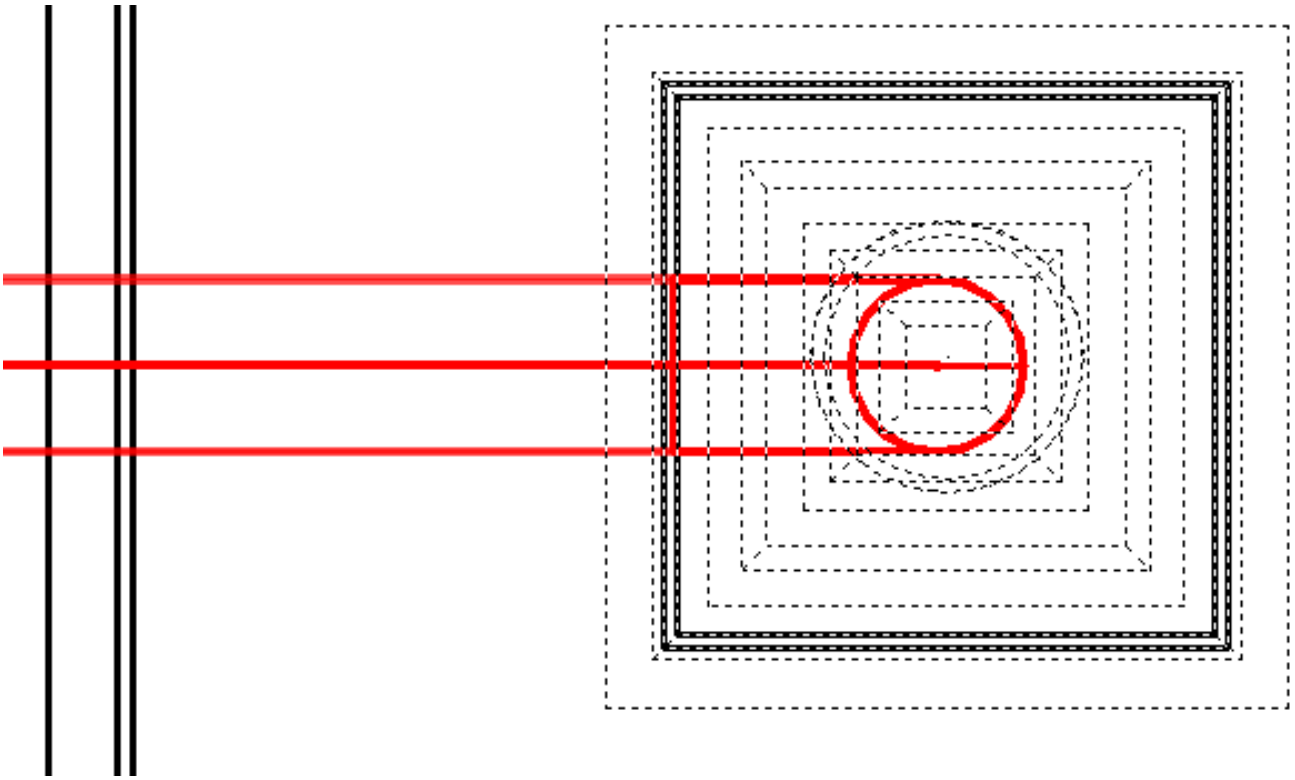


Figure 6.2017

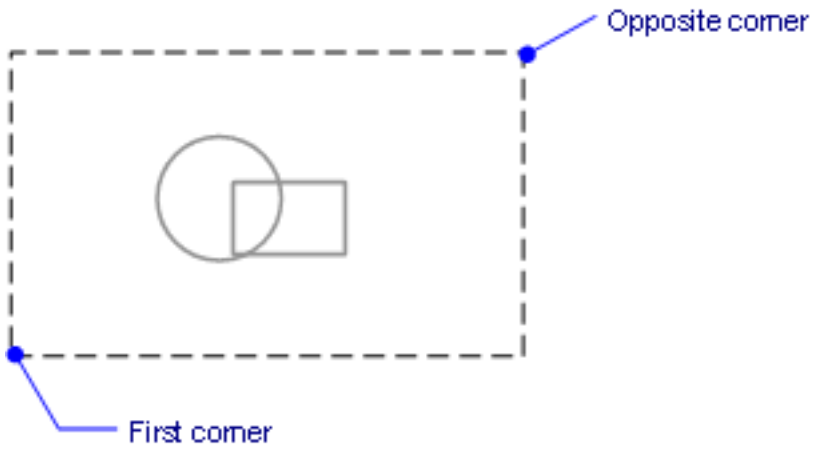


Figure 6.2018

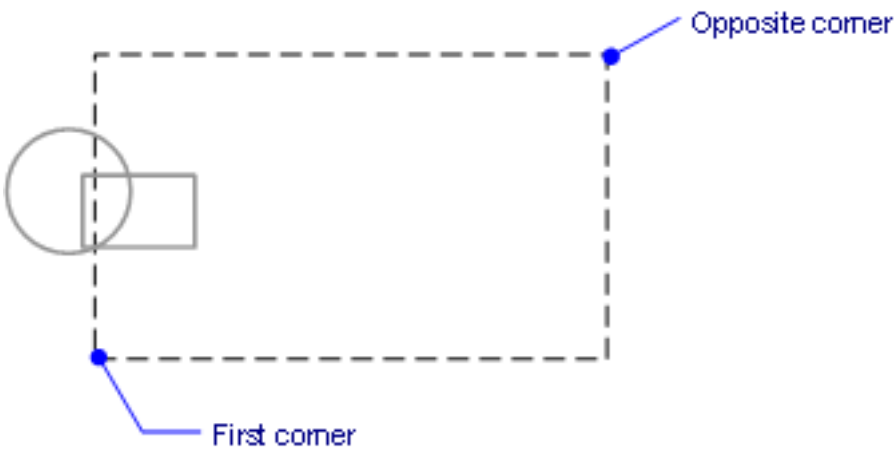


Figure 6.2019

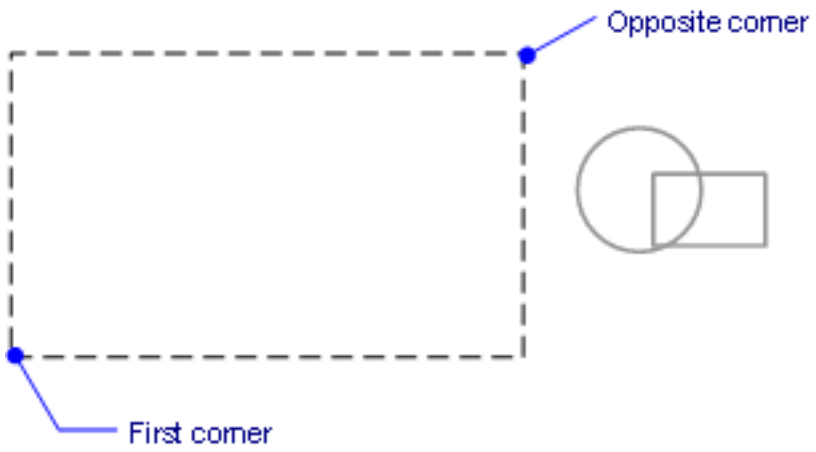


Figure 6.2020

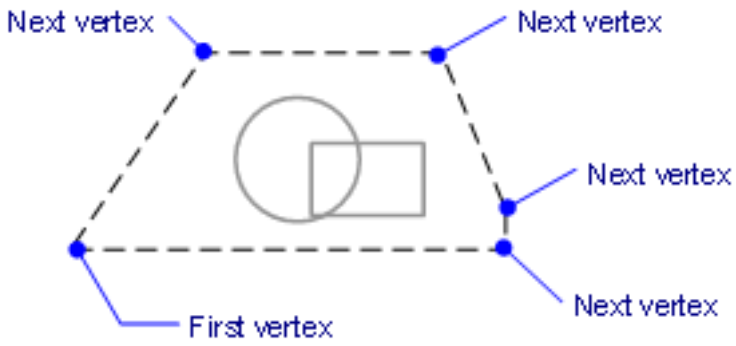


Figure 6.2021

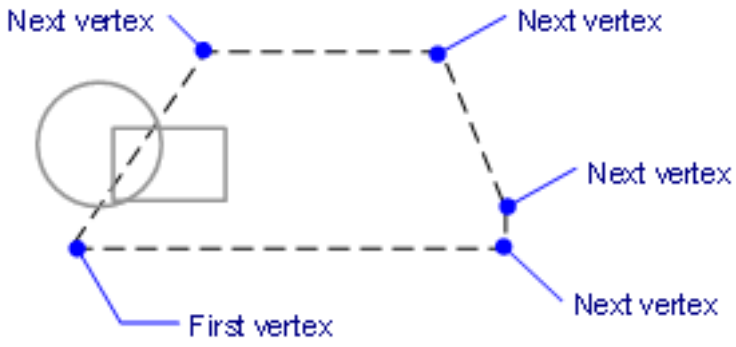


Figure 6.2022

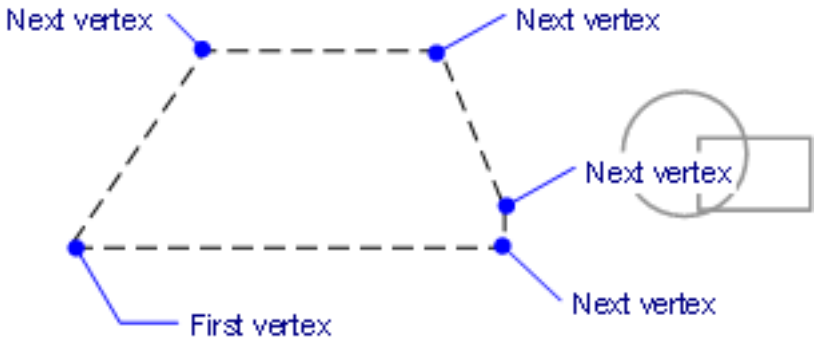


Figure 6.2023

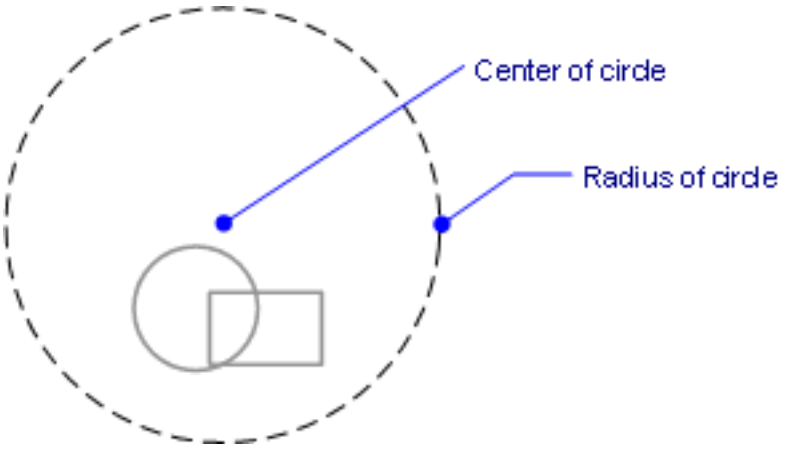


Figure 6.2024

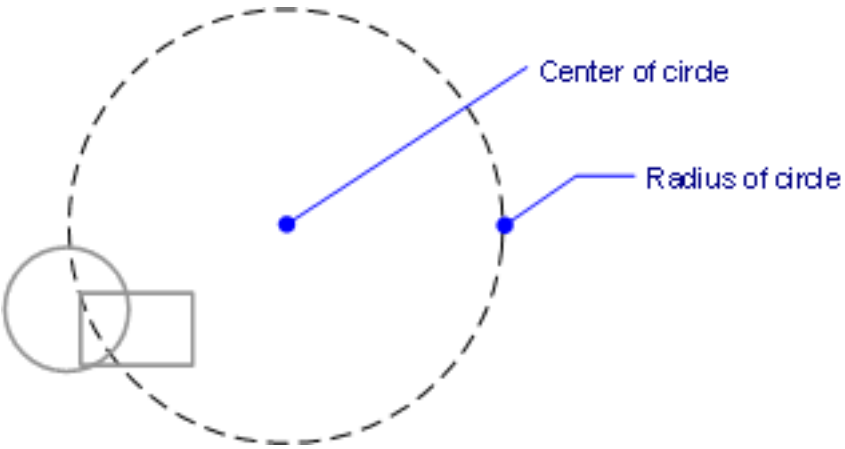


Figure 6.2025

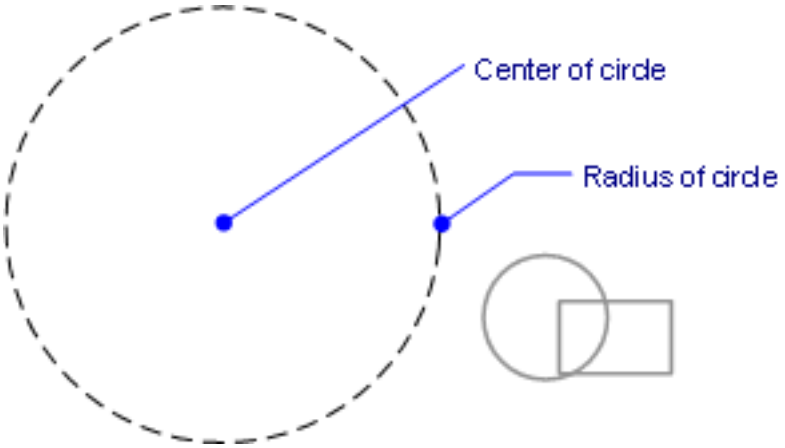


Figure 6.2026

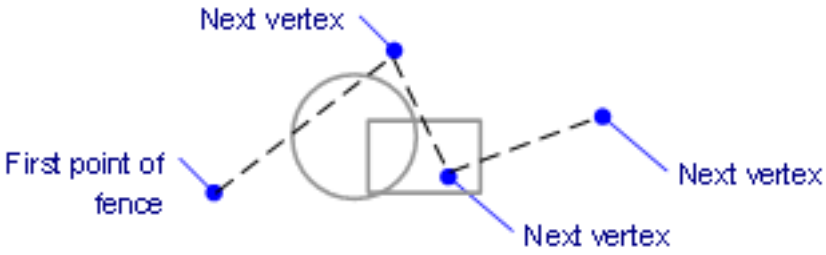


Figure 6.2027

Entity selection	
Pick automatic	0x0003 (3)
negative	☐ Disables window selection, saving earlier value
1	☑ Draws a selection window if mouse is not over any entity when clicked
2	☑ Draws a selection window if mouse is clicked over an entity and dragged
Pick box	4
Pick drag	[0] Draw selection window using two points
Pick first	[1] First select entities, then issue a command

Figure 6.2028



6.27.18 SelectAlignedSolids

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Selects all solids in a model that have faces coplanar (laying in the same plane) with the selected face.

Accessing the Command

 command bar: **selectalignedsolids** Quad cursor: **Solid/Face Selection** |  **: SelectAlignedSolids** Prompts you in the command bar: Select face: **(Click a face in the model)** The program selects all other solids that have faces in the same plane, shown outlined in blue in the figure below (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/SelectAlignedSolids.png})

Figure 6.2029

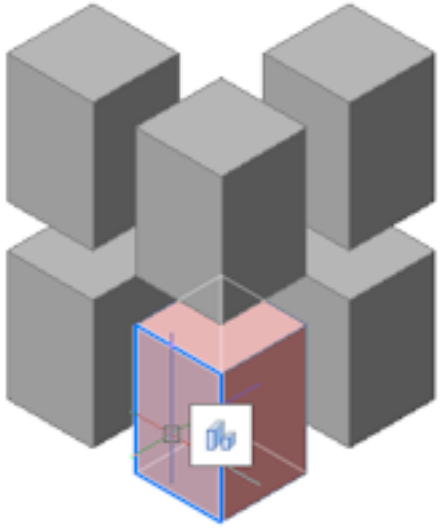
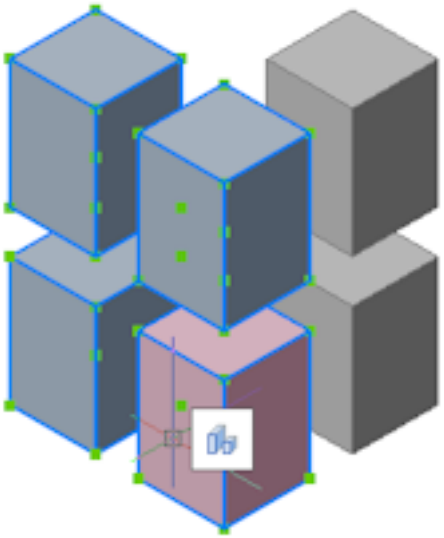


Figure 6.2030



Command Options

Option	Description
Select face	Select a single face for the program to match to other solids with coplanar faces

Related Commands

BimMultiSelect (6.7.16) - selects linear solids with coplanar and/or parallel axes. dmSelect (6.12.26) - selects edges and faces of 3D solids or surfaces based on their geometric properties. SelectAlignedFaces (6.27.16) - selects all faces in a model which are coplanar with a selected face. SelectConnectedFaces (6.27.19) - selects all faces in a model which are connected to a selected face. SelectConnectedSolids (6.27.20) - selects all solids in a model which are connected to a selected face.

Figure 6.2031



6.27.19 SelectConnectedFaces

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Selects all faces that connected a solid with the selected face.

Accessing the Command

 command bar: **selectconnectedfaces** Quad cursor: **Solid/Face Selection** |  : **SelectConnectedFaces** Select face: **(Click a face in the model)** The program selects all other faces that on solids that connect to the solids with the selected face, shown outlined in blue in the figure below (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e
(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/SelectConnectedFaces.png

Figure 6.2032

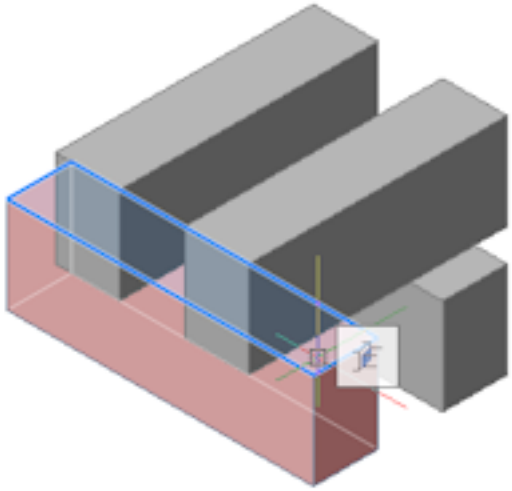
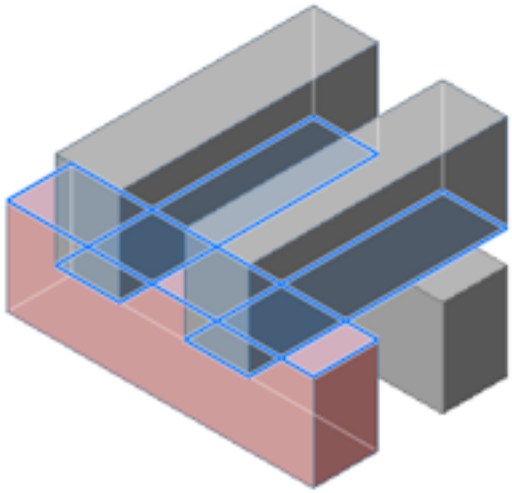


Figure 6.2033



Command Options

Option	Description
Select face	Select a single face for the program to match to faces on other, connected solids

Related Commands

BimMultiSelect (6.7.16) - selects linear solids with coplanar and/or parallel axes. dmSelect (6.12.26) - selects edges and faces of 3D solids or surfaces based on their geometric properties. SelectAlignedFaces (6.27.16) - selects all faces in a model which are coplanar with a selected face. SelectAlignedSolids (6.27.18) - selects all solids in a model of which a face is coplanar with a selected face. SelectConnectedSolids (6.27.20) - selects all solids in a model which are connected to a selected face.

Figure 6.2034



6.27.20 SelectConnectedSolids

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Selects all solids connected to a selected face.

Accessing the Command

 command bar: **selectconnectedsolids** Quad cursor: **Solid/Face Selection** |  **: SelectConnectedSolids** Select face: **(Click a face in the model)** The program selects all solids connect to the solid with the selected face, shown outlined in blue in the figure below: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SelectConnectedSolids.png})

Figure 6.2035

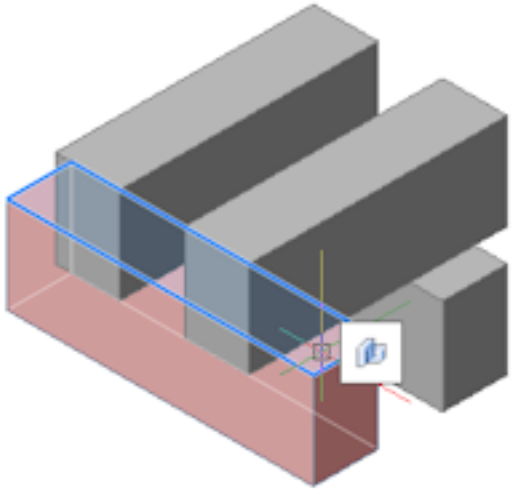
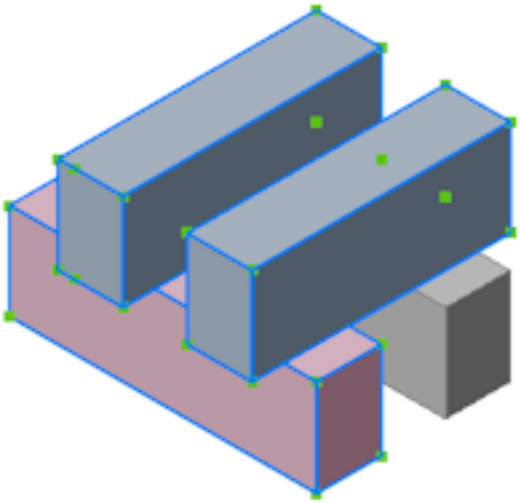


Figure 6.2036



Command Options

Option	Description
Select face	Select a single face for the program to match to connected solids

.

Related Commands

BimMultiSelect (6.7.16) - selects linear solids with coplanar and/or parallel axes. dmSelect (6.12.26) - selects edges and faces of 3D solids or surfaces based on their geometric properties. SelectAlignedFaces (6.27.16) - selects all faces in a model which are coplanar with a selected face. SelectAlignedSolids (6.27.18) - selects all solids in a model of which a face is coplanar with a selected face. SelectConnectedFaces (6.27.19) - selects all faces in a model which are connected to a selected face.

Figure 6.2037



6.27.21 SelectSimilar

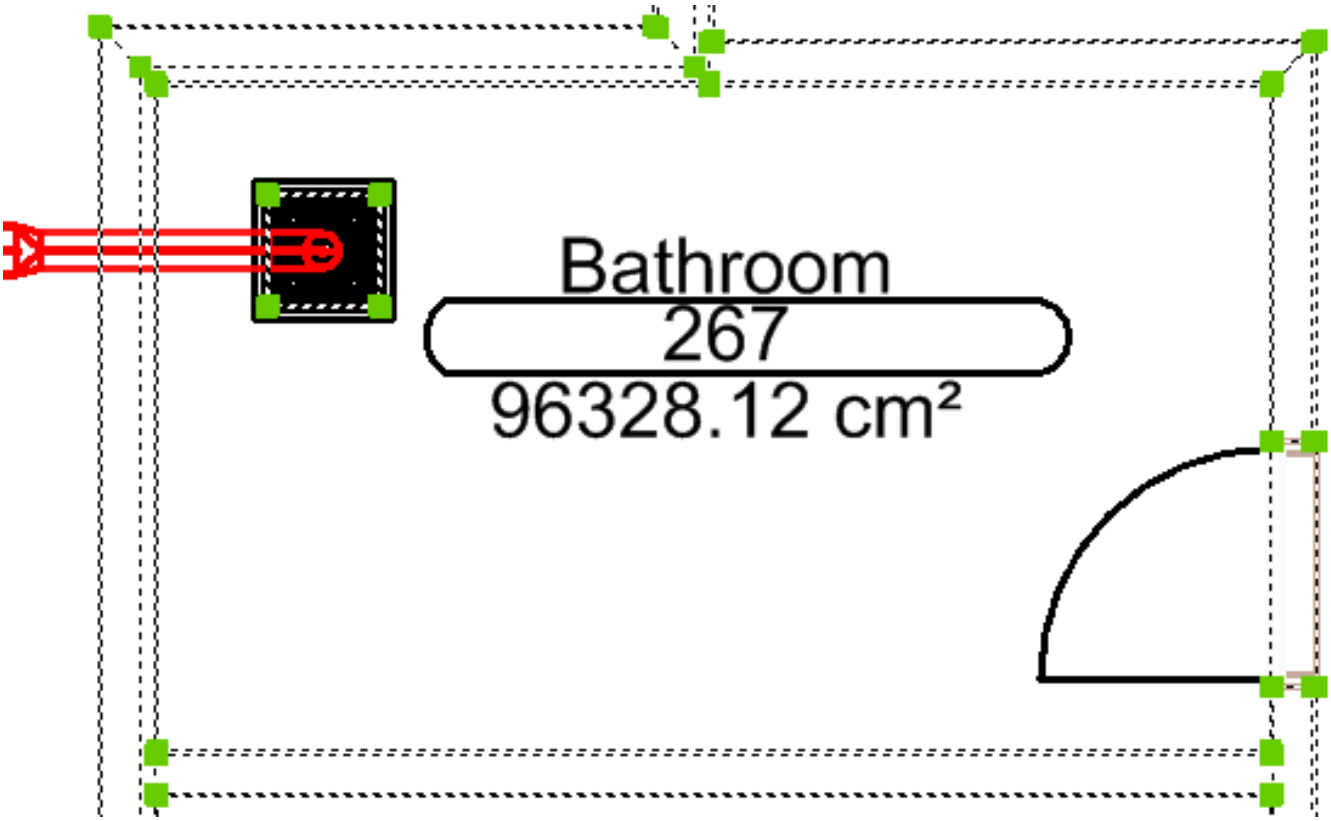
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Selects all entities with the same properties as the currently selected entity.

Accessing the Command



command bar: **selectsimilar** menu bar: **Edit | Select Similar** ribbon:**Home | Settings | Select | Select Similar** toolbar: **Standard |** **: SelectSimilar Prompts you in the command bar:** Select entities or [SEttings]: **(Select one or more entities, or else type SE to display a dialog box to specifythe properties to match)** Entities in set: 1 Select entities or [SEttings]: **(Select more entities, or else press Enter to end the command)** When you exit the command, the entities that match the properties are selected (shown as dashed and with entity grips): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/selectsimilar-eg.png})

Figure 6.2038



Command Options

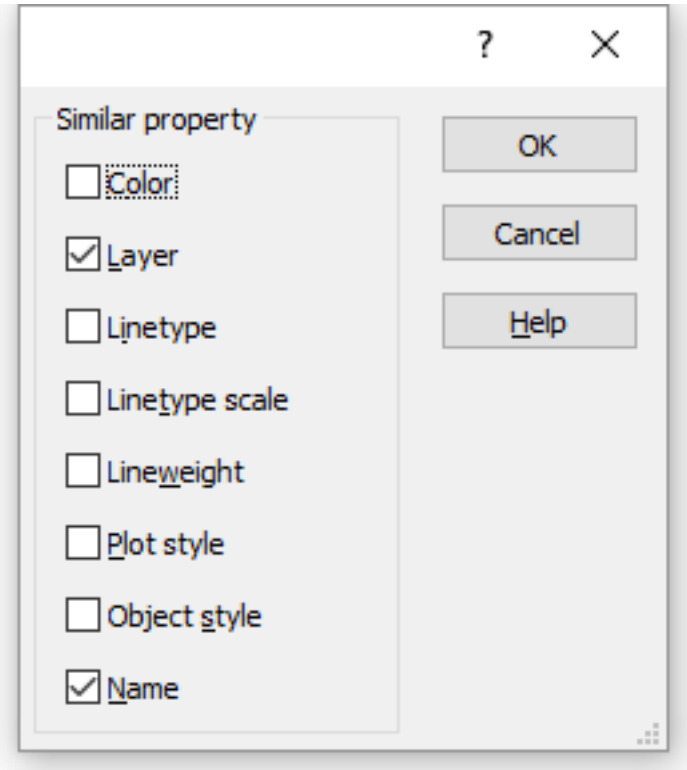
Option	Description
Select entities	Selects representative entities, whose properties should be matched by entities elsewhere in the drawing: • Select one or more entities • Press **Enter** to end the command and show the selected entities
SEttings	Displays a dialog box; see below

Dialog Box Options

Enter the SEttings option to access this dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cm

Option	Description
Similar Property	Specifies the properties that this command should match: • **On** - property is matched in other entities • **Off** - property is not matched For this command to operate as intended, at least one property must be turned on. When all properties are turned off, this command selects only the entity(ies) you pick at the 'Select entities' prompt. Selects all other entities with the same property as the selected entity(ies): • **Color** - color number or name • **Layer**-layer name • **Linetype** - linetype name • **Linetype Scale** - linetype scale factor

Figure 6.2039



Related Commands

QSelect (6.25.5) - composes a selection set using filters. Select (6.27.17) - selects one or more entities; can be accessed later with the Previous option. SelGrips (6.27.22) - selects entities and displays grips.

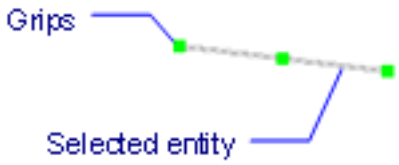
Figure 6.2040

Match options for SELECTSIMILAR	0x0082 (130)
1	☐ Color
2	☑ Layer
4	☐ Linetype
8	☐ Linetype scale
16	☐ Lineweight
32	☐ Plot style
64	☐ Object style
128	☑ Name

Figure 6.2041



Figure 6.2042



6.27.22 SelGrips

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Selects entities, and then displays their grips (short for "select grips"). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel

Accessing the Command

command bar: **selgrips** shortcut: select one or entities with the cursor keyboard: **Ctrl+A** alias: **selgrip** menu:**Edit | Select All : SelGrip Prompts you in the command bar:** Select entities to display grips: **(Choose one or more entities)** S elect entities to display grips: **(Choose more entities, or else press Enter to end the command)** BricsCAD highlights selected entities and displays their grips. You edit the entities by dragging the grips.To select all entities in the drawing, press Ctrl+A.

Command Options

Option	Description
Select entities to display grips	Chooses one or more entities.

Related Commands

Select (6.27.17) - selects entities. SelectSimilar (6.27.21) - selects all entities of the same type as the current selection, based on a set of matching properties, stored in the SELECTSIMILARMODE (1.13.11) system variable DdGrips (6.10.17) - sets the parameters for grips through the Settings dialog box.

Figure 6.2043



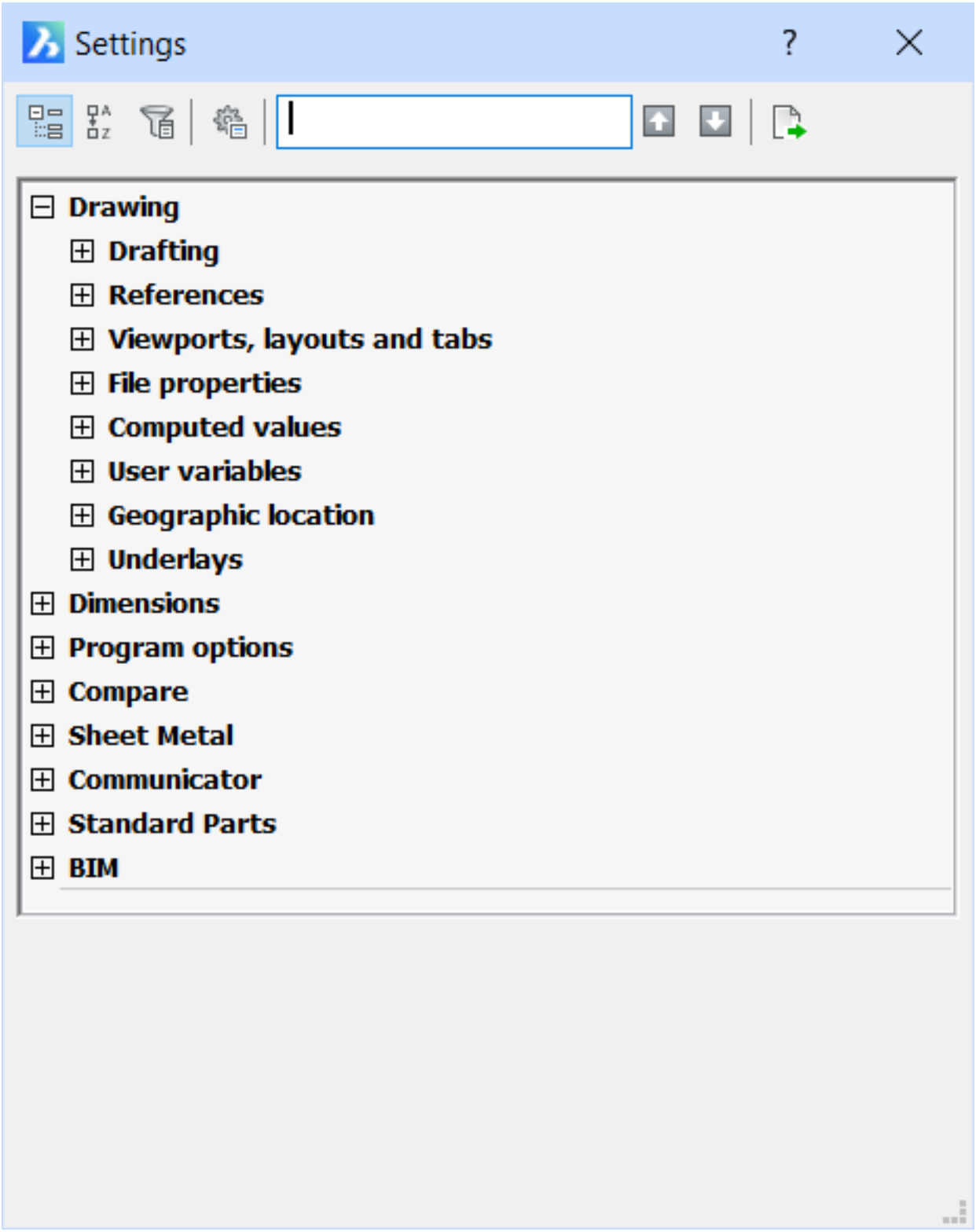
6.27.23 Settings

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Changes the values of system variables and user preferences through a dialog box.

Accessing the Command


command bar: **settings** aliases: **ddrmodes,rm,options,dsettings,ddsetva (6.10.19)r (6.10.19)** menu bar: **Settings | Settings** toolbar: **Settings** ribbon:**Home | Settings | Settings : Settings** Displays adialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/settings-dlg.png})

Figure 6.2044



Dialog Box Options

Toolbar Option	Description
Categorized	Lists variables in related groups
Alphabetic	Lists variables alphabetically by description (rather than by name)
Show Differences	Lists variables whose values are changed from the initial (default) values: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/set-changes-dlg.png}) Make a change to a variable, and it will show up here. This is handy for figuring out what might have changed. To change a value back to the default: 1. Right-click the name 2. Choose **Restore Default Value** from the shortcut menu
Dialog Configurations	Search for variables, configures how the search is carried out, and specifies how differences are handled; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/set-configure-dlg.png}) **Search** - finds variables and limits where the program searches: • **Find What** - specifies the search phrase • **Find** - executes the search • **Find Where** - determines which part(s) of the Settings dialog box should be searched • **Match Case** - when on, search phrase's capitalization must match that of the words in the Settings dialog box Modified Settings - determine which variables are displayed by the Show Differences dialog box: • **Display All**- lists variables whose values have changed • **Display Settings Stored in Drawing** - lists variables that affect the current drawing • **Display Settings Not Stored in Drawing** - lists variables that affect all drawings • **Display Modified Settings in a Different Color**- when on, displays changed entities in a color other than black • **Color swatch** - click the color swatch to choose a different color

Find	Real-time search field that lists variables when you enter the first few letters of their names or descriptions. As you type into the search field, BricsCAD immediately jumps to the first item that matches the letters. For instance, when entering “lastpoint,” the following occurs: 1. **l** — focus jumps to Insertion Unit, because the description contains the letter ‘l’ 2. **la** — focus jumps to Unit Mode, because ‘la’ is in “displayed” in the description 3. **las** — focus jumps to Text Angle, because of ‘las’ in “last” of the description 4. **lastp** — focus jumps to Last Point As you enter text, the color of the search box changes: • **Green** - one word matches the search phrase; you have arrived! • **White** - two or more words match the search phrase; click the Next and Previous buttons to review them • **Orange** - no words match the search phrase; check your spelling
Next	Jumps to the next candidate that matches the search phrase
Previous	Jumps to the previous candidate that matches the search phrase
Export	Saves all settings and values to a CSV file, short for “comma-separated values.” The file contains names of variables in true alphabetical order, along with their current values, and other information. The file can be imported into a spreadsheet.
X	Closes the dialog box.

Procedures

Understanding the Settings dialog box (1.13.11)

Related Commands

SetVar (6.27.26) - changes system variables through the command bar. SettingsSearch (6.27.24) - opens the Settings dialog box at the specified category, system variable or user preference.

Figure 6.2045

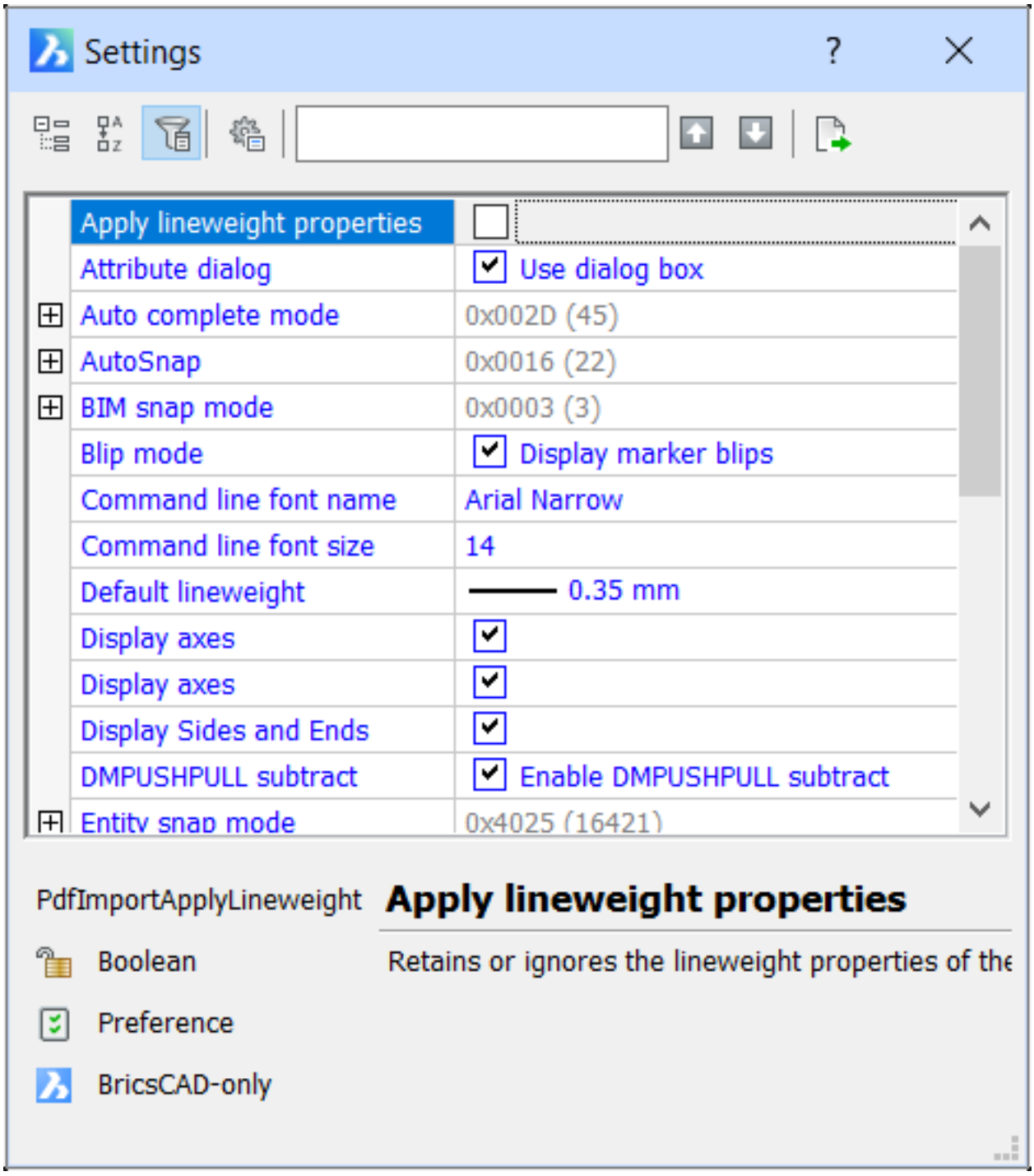


Figure 6.2046

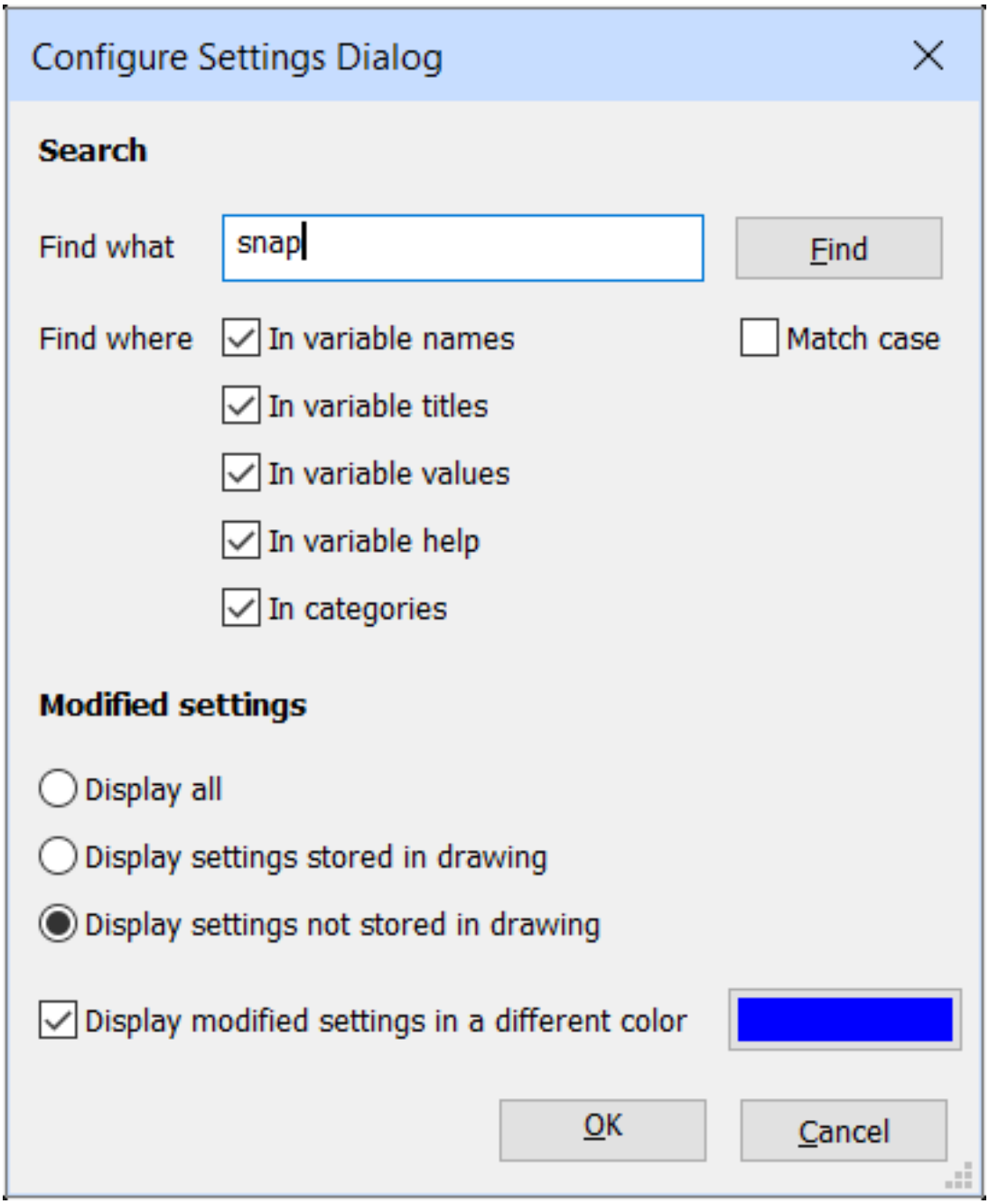


Figure 6.2047



6.27.24 SettingsSearch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Settings dialog box at the word specified by the user.

Accessing the Command

command bar: **settingssearch** : S**ettingsSearch Prompts you in the command bar:** Search setting/category: **(Type a word used in the Settings dialog box)** Search is case sensitive, which affects the results. In general, the Settings dialog box uses fully capitalizes variable names (like INSUNITS) but uses mixed-case for user preferences names, such as SnapMarkerSize.

Command Options

Option	Description
Search setting/category	Type the name of a setting category, variable, user preference, or other word that might be found in the Settings dialog box; accepts only a single word

Related Commands

Settings (6.27.23) and DdSetVar (6.10.19) - changes the values of system variables through a dialog box. SetVar (6.27.26) - changes system variables through the command bar.

Figure 6.2048



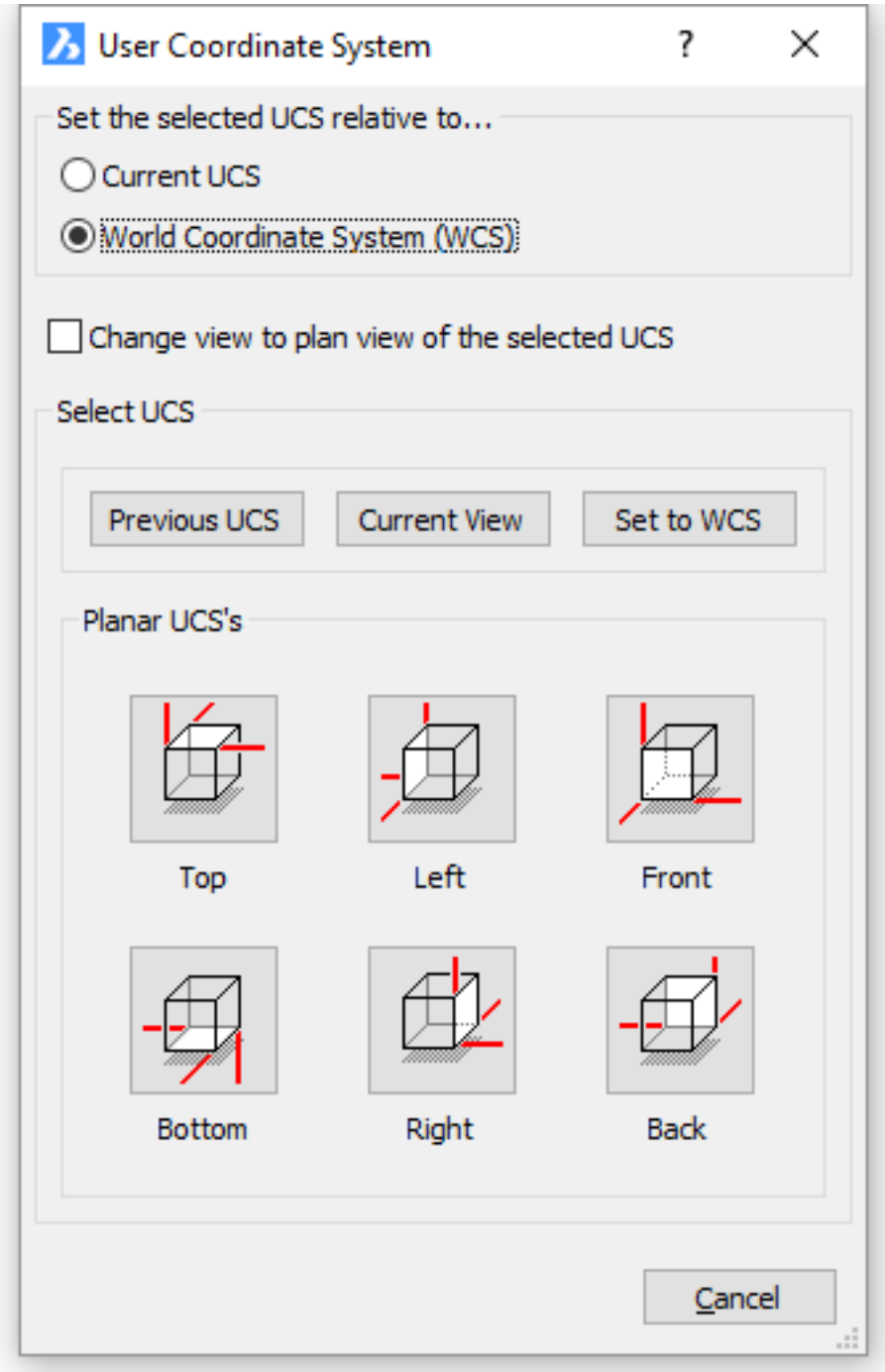
6.27.25 SetUCS

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the UCS to a specified viewpoint (short for "set user-defined coordinate system").

Accessing the Command

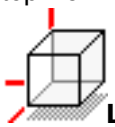
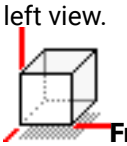
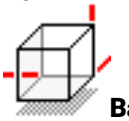

command bar: **setucs** aliases: **dducsp** **ucp** menu bar: **Settings | Set UCS** ribbon: **Settings | UCS** toolbar: **Settings** |  **: SetUCS** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef
Click on a button to use a perset UCS; the dialog box disappears automatically. Use the UCS (6.30.2) command's View option to name the UCS generated by this command.

Figure 6.2049



Dialog Box Options

Option	Description
Set the selected UCS relative to	Specifies how the new UCS is set: • **Current UCS** - sets the new UCS relative to the current UCS • **World Coordinate System -** sets the new UCS relative to the WCS WCS is short for "world coordinate system," and is the default coordinate system when no UCS is set; it cannot be changed.
Change view to plan view of the selected UCS	Toggles whether the plan view is displayed when switching UCSes: • **Yes** - plan view of the enw UCS is displayed • **No** - the viewpoint does not change Plan view looks down the z axis at the x,y plane.
Select UCS	Sets the UCS to one of the following coordinate systems. Clicking one of these buttons sets the UCS, and then immediately exits the dialog box: • **Previous UCS** - changes the UCS to the previous UCS • **Current View** - changes the UCS to match the current viewpoint • **Set to WCS** - changes the UCS to match the WCS

Planar UCSs	Sets the UCS to one of the following standard orthographic views: 1. Click one of the buttons 2. The program sets the UCS, and then exits the dialog box •  **Top** - changes the UCS to match the top view. •  **Left** - changes the UCS to match the left view. •  **Front** - changes the UCS to match the front view. •  **Bottom** - changes the UCS to match the bottom view. •  **Right** - changes the UCS to match the right view. •  **Back** - changes the UCS to match the back view.
Cancel	Closes the dialog box without changing the UCS.

Related Commands

Ucs (6.30.2) - creates and modifies user-defined coordinate systems. UcsIcon (6.30.3) - controls the appearance of the UCS icon. ExpUcs (6.13.18) - creates, modifies, and deletes named UCSs through the Drawing Explorer. Plan (6.24.25) - changes the viewpoint to the plan view of the WCS or current UCS. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Settings (6.27.23) - sets parameters of UCSs.

Figure 6.2050



6.27.26 SetVar

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays and changes the values of system variables (short for "set variables"). **Note:** You can also enter the names of system variables directly at the command prompt, without using this command.

Accessing the Command

command bar: **setvar** transparent: **'setvar** alias: **set** menu bar: **Tools | Set Variable** : **SetVar Prompts you in the command bar:** Variable name or ?: **(Enter the name of a system variable, or press ?)** For example: Variable name or ?: chamferd New current angle for CHAMFERD <0>: **(Enter a value, or press Enter to accept the current value)** The values of read-only variables cannot be changed. For example: : **program** PROGRAM = "BRICSCAD" (read only)

Command Options

Option	Description
Variable name	Specifies the name of a system variable.
?	Lists the names of system variables; prompts you: **Variable(s) to list <*>** - **(enter one of the following:)** • ***** to list all system variables • **name*** to list all system variables starting with name • **name** to list one system variable

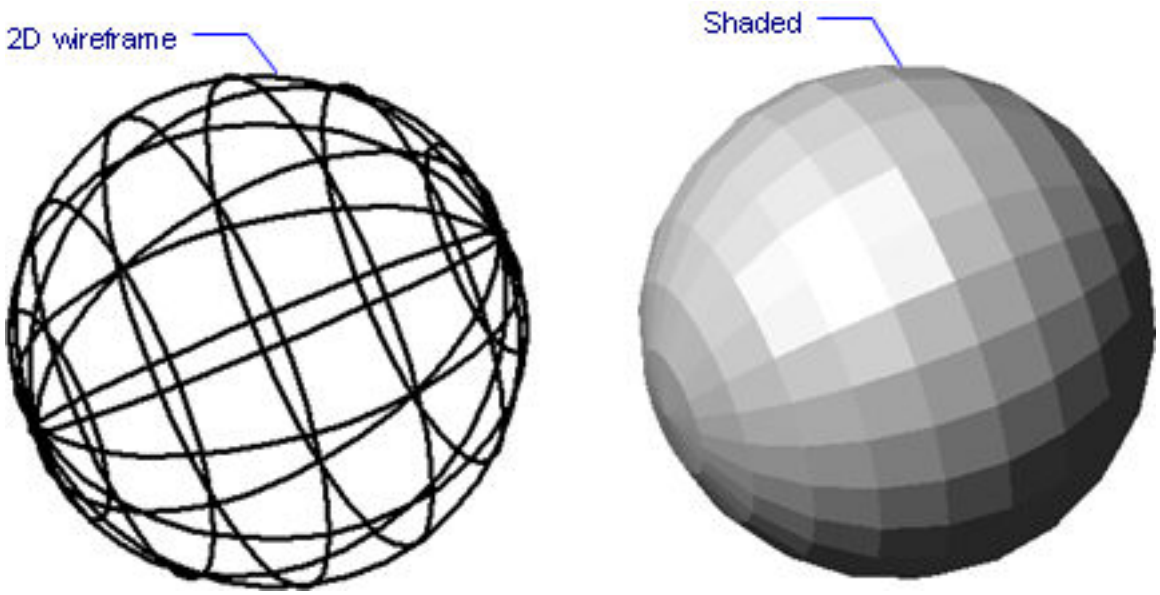
Related Commands

Settings (6.27.23) and DdSetVar (6.10.19) - change the values of system variables through a dialog box. SettingsSearch (6.27.24) - opens the Settings dialog box at the specified category, system variable or user preference.

Figure 6.2051



Figure 6.2052



6.27.27 Shade

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Generates shaded images of 3D drawings. **Note:** This command is largely superseded by the VisualStyles (6.31.13) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdR

Accessing the Command


command bar: **shade** alias: **sha** menu bar: **View | Rendering | Shade** ribbon:**View | Render Modes | Shade** toolbar: **Rendering |**  **: Shade** Renders the 3D entities with the shading mode specified by the ShadeMode (6.27.29) command, and then reports in the command bar: **Note:** In order to switch back to wireframe mode run command SHADEMODE

Command Options

No options.

Related Commands

Hide (6.16.10) - removes hidden lines from 3D entities. Render (6.26.22) - generates a photorealistic rendering using materials and lights ShadeMode (6.27.29) - sets the shading style for 3D entities.

6.27.28 Sh

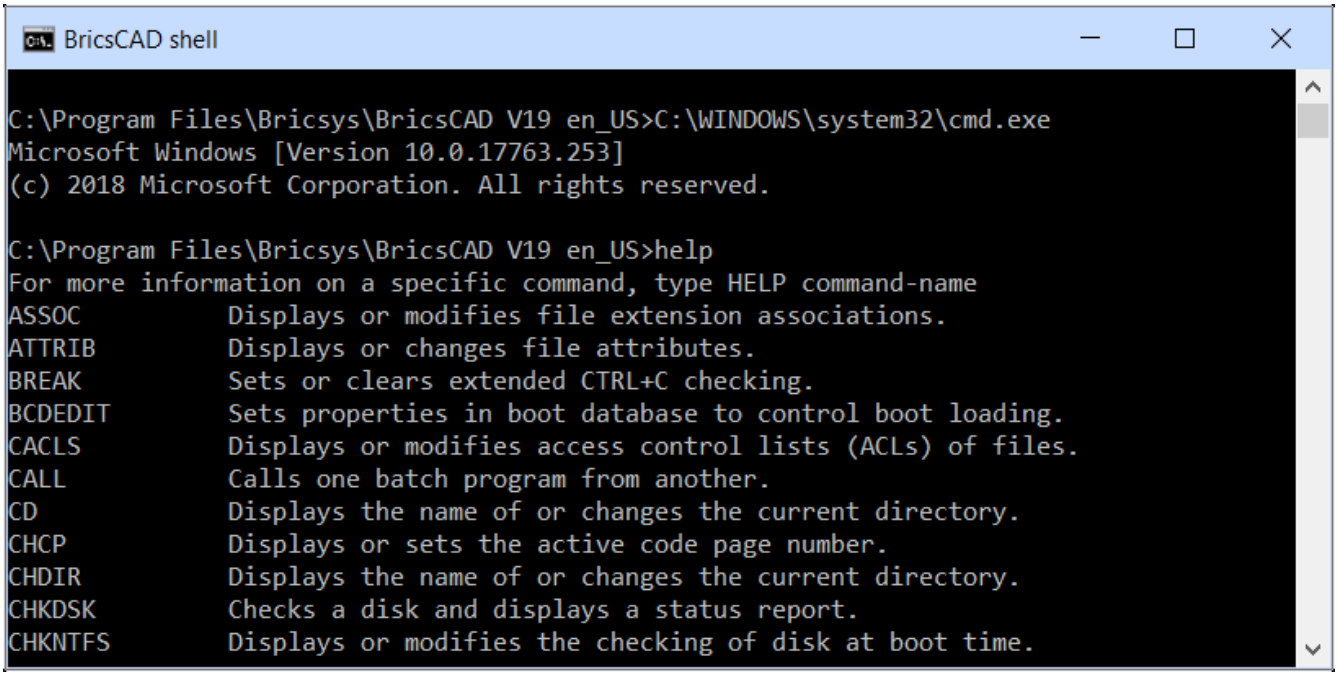
   Opens operating system's command prompt window so that you can run other applications (short for "shell").

Accessing the Command

command bar: **sh** transparent: **'sh** alias: **shell : Sh Prompts you in the command bar:** OS Command: **(Enter the name of a program, or press Enter for the command prompt window)** (\vref{fig:C:/Dev/Task-2448/images/staticbricsyscom/hdlg.png})

1. Enter commands or names of programs, such as 'Notepad'.
2. For a list of operating system commands, enter **Help**.
3. To exit the operating system window, enter **Exit**.

Figure 6.2053



Command Options

Option	Description
OS Command	Specifies the name of the Windows program to run. Press operating systemto open the Windows command prompt window.

Procedures

- Use the Customize (6.9.58) command to create custom shell commands.

Related Commands

Start (6.27.44) - starts other applications. Shell (6.27.32) - opens the operating system command prompt window; runs other applications. Files (6.14.5) - displays the operating system's File Manager.

Figure 6.2054



6.27.29 ShadeMode

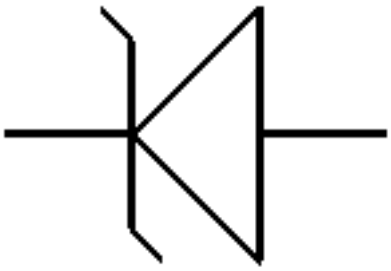
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) **Note:** This command is replaced by the VisualStyles (6.31.13) command.

Related Commands Hide (6.16.10) - removes hidden lines from 3D entities. Render (6.26.22) - generates a photo-realistic rendering using materials and lights RedSdkInfo (6.26.40) - displays rendering related hardware and driver specifications in the command bar. Shade (6.27.27) - generates shaded images of 3D drawings. Solprof (6.27.62) - creates a hidden line representation of 3D solids in a layout viewport. VisualStyles (6.31.13) - creates and controls visual style definitions in the Drawing Explorer.

Figure 6.2055

Shape Classic Pro Platinum Mechanical BIM

Figure 6.2056



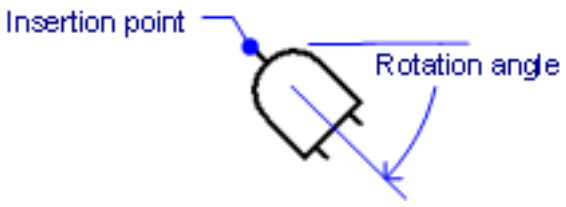
6.27.30 Shape

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Places shapes in drawings. **Note:**Before you can place shapes in drawings, you must first load a SHP shape file with the Load (6.20.36) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/shape-eg.png})

Accessing the Command

command bar: **shape** : **Shape Prompts you in the command bar:** ? to list/Shape to insert: **(Enter the name of a shape, or press ?)** Insertion point for shape: **(Pick a point, or enter x,y coordinates)** Scale factor for shape <1.0>: **(Enter a positive or negative number)** Rotation angle for shape <0>: **(Enter a positive or negative number)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image577.png}) Shapes are an early version of blocks that were highly efficient but difficult to code; shapes are rarely used anymore.The SHP format is also used by certain fonts; it contains a flag to distinguish between shapes and fonts.To code shapes, see **Customizing BricsCAD**.

Figure 6.2057



Command Options

Option	Description
Shape to insert	Specifies the name of the shape; prompts you: 1. **Insertion point for shape** - **(specify the insertion point; pick a point, or enter x,y coordinates.)** 2. **Scale factor for shape** - **(specify the scale factor; enter a positive or negative number larger or smaller than 1.0;)** Negative numbers mirror the shape. 3. **Rotation angle for shape** - **(specify the rotation angle; enter a positive or negative number;)** positive angles are measured clockwise from the positive x axis.
? to list	Lists the names of shapes loaded in the current drawing.Reports the file name and names of shapes: `Loaded shapes: Shape file: C:\dwg \ES .SHX CON1 RES` If no shapes are loaded, reports: *** no file *** Use the Load (6.20.36) command to load shapes.

Grips Editing

Shapes can be edited directly through grips:

1. Select the shape. Notice that it has one grip at its insertion point.
2. Drag the grip to move the shape.

(\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image763.png})

Figure 6.2058



Related Commands

Load (6.20.36) - loads SHP shape files. Insert (6.17.13) - inserts blocks into drawings.

Figure 6.2059



6.27.31 SheetSet

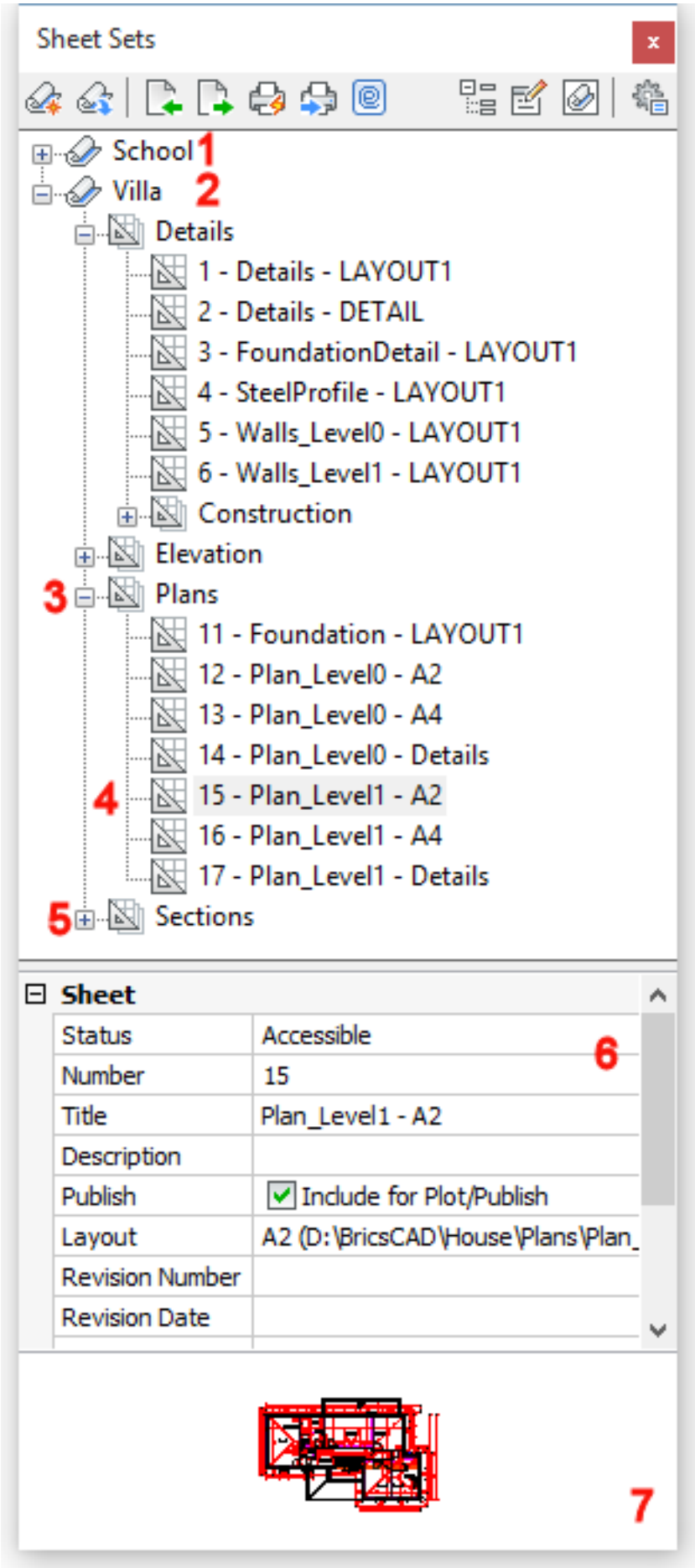
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Sheet Sets panel.

Accessing the Command

 command bar: **sheetset alias: ssm** menu bar: **Tools | Sheet Sets** toolbar: **Standard** |  context menu: right-click a toolbar or ribbon, then choose**Sheet Sets** : **SheetSet** Displays a panel: (\vref {fig:C:/Dev/Task-2448/images/staticb

1. Sheet Set (collapsed)
2. Sheet Set (expanded)
3. Sheet Subset (Expanded)
4. Sheet
5. Sheet Subset (collapsed)
6. Properties and variables of the currently selected sheet set, subset or sheet
7. Preview of the currently selected sheet

Figure 6.2060



Panel Options

Option	Description
New sheet set	Launches the NewSheetSet (6.22.5) command.
Open sheet set	Launches the OpenSheetSet (6.23.9) command.When a DST file is dropped on the application, it is opened in the Sheet Set Explorer.
Export to XML	Exports a sheet set in XML format.Displays the Export Sheet Set dialog box.
Import from XML	Displays the Import Sheet Set dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/I • **XML File** - click**Browse** , select an XML file in the Select XML File dialog box, then click **OK**. • **Save Sheet Set As**- choose whether to save the imported XML file as DST file using the same location and name as the XML file or to save it to a new location or name. If you choose the latter option, click**Browse** in theDST File field, then specify a folder and file name. • **OK**- imports the XML file and saves it to a DST file.
Plot	Plots the currently selected sheet set(s), subset(s) or sheet(s).Depending on the **Publish Sheets in Subset** property of a subset and the **Publish** property of sheets, sheets might not be included.
Publish	Publishes the currently selected sheet set(s), subset(s) or sheet(s).Depending on the **Publish Sheets in Subset** property of a subset and the **Publish** property of sheets, sheets might not be included.
eTransmit	Launches the eTransmit (1.14.10) command. The currently selected sheet set(s), subset(s) or sheet(s) are included in the eTransmit procedure.
Selection sets	Opens the Sheet Selection Sets dialog: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S • Select a selection set, then click the **Activate** button to restore the selection. • Right-click a selection set to display a context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

 Custom Properties	Opens the Custom Properties dialog, which allows you to add custom properties for sheet sets and sheets. Alternatively, in the sheet tree right-click the name of a sheet set or a sheet, and then choose **Custom Properties**in the context menu; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S • Click the **Sheet Set** tab to manage custom sheet set properties. • Click the **Sheets** tab to manage custom sheet properties. • Click the **Add** button to add a custom property; displays the Add Custom Property dialog box in which you add a property name and its initial value: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou • **Edit** - select a custom property, then click the **Edit** button to edit the value of a custom property.Displays the Custom Sheet Property or Custom Sheet Set Property dialog box in which you edit the name of the property: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou • **Delete** - select a custom property, then click the **Delete** button to delete the selected property.
 View Categories	Expands and collapses the currently loaded sheet sets.

Sheet Set Options

Opens the Sheet Sets options in the Settings (1.13.11) dialog. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

- **Sheet set found** (SSFOUND) - displays the sheet set file name associated with the current drawing.
- **Sheet set locate** (SSLOCATE) - controls whether BricsCAD tries to locate and open a sheet set when a drawing is being opened.
- **Sheet set manager auto open** (SSMAUTOOPEN) - controls whether or not the rpgram displays the Sheet Set Manager when a drawing is opened which is associated with a Sheet Set.
- **Sheet set manager poll time** (SSMPOLLTIME) - controls the time interval between automatic refereshes of the status data in a sheet set if the value of SSMSHEETSTATUS = 2.
- **Sheet set manager status** (SSMSHEETSTATUS) - controls whether the status data in a sheet set is refreshed.
- **Sheet set manager state** (SSMSTATE) - determines whether the Sheet Set Manager is active or not.
- **Sheet set template path** (SheetSetTemplatePath) - specifies the path to the Sheet Set Templates folder.Click the **Browse** button ( ), then select the path in the File dialog.The default path is:**C:\Users\<user name>\AppData\Local\Bric-sys\BricsCAD\Vxx\en_US\Templates\Sheet Sets**
- **Default new sheet template** (DefaultNewSheetTemplate) -specifies the default drawing template file (*.dwg or *.dwt) for new sheets.Click the **Browse** button ( ), then select the file in the F**ile** dialog.
- **Sheet number leading zeroes** (SheetNumberLeadingZeroes) -specifies how to format the **Number** value of new sheets.
- **Sheet set automatic backup** (SheetSetAutoBackup) - determines whether a backup is created whenever a sheet set is opened. The backup files have a *.ds$ extension.

New Sheet(s)	Right click the sheet set name, a subset or a sheet, then choose **New Sheet(s)**in the context menu.The New Sheet(s) dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N • **Create from scratch, add new drawing** - creates a new drawing and adds a layout in the drawing to the sheet list in the current sheet set or subset: **Number** - accept the automatically calculated number, or else type a new number **Title**- type a name in the Title field to give new sheet a name **File name**- accept the automatically generated name (<Number>+<Title>) or else type a new name **Folder Path**- accept the current path or else clickBrowse  to set a different path **Template** - accept the current template or else clickBrowse  to select a different template drawing **Open drawing**- when checked, opens the newly created drawing after you click**OK**. • **Import layouts from existing drawings**- creates new sheets through the New Sheet(s) dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so Click**Add drawing(s)**to select the drawings from which you want to import layouts.The layout(s) in the selected drawing(s) are listed: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so Uncheck the layouts you do not want to import as a sheet. Choose a option from the **Sheet titles** droplist. The options are: **Drawing name**, **Drawing name - Layout name**, and**Layout name**. Choose whether or not to number the new sheets automatically. Click**OK**to create the new sheet(s).

Context Menus

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sheetse

Right click a sheet set to open the **Sheet Set** context menu.

- **Close Sheet Set** - closes the sheet set.
- **Export to XML**- displays the **Export Sheet Set As** file dialog box. Writes the sheet set definition to an XML file.
- **New Sheet**- displays the New Sheet dialog box, which allows to add new sheet(s) to the current sheet set.
- **New Subset** - adds a new subset to the current sheet set.
- **Selection Sets**- displays the Selection Sets dialog box.
- **Custom Properties**- displays the Custom Properties dialog box.
- **Plot**- prints all sheets in the sheet set.
- **Publish** - launches the Publish (6.24.55) command and loads all sheets in the sheet set and its subsets.
- **eTransmit**- launches the eTransmit (1.14.10) command to package together all drawings and support files.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sheetse

Right click a sheet subset to open the **Subset** context menu.

- **Collapse** / **Expand**- collapses or expands the subset.
- **New Sheet**- displays the New Sheet dialog box, which allows to add a new sheet to the subset.
- **New Subset**- adds a new subset to the subset.
- **Rename Subset**- changes the name for the subset.
- **Remove Subset**- deletes the subset. The deletion is definitive when the modifications are saved when closing the sheet set.
- **Plot**- prints all sheets in the subset.
- **Publish**- launches the Publish (6.24.55) command and loads all sheets in the subset.
- **eTransmit**- launches the eTransmit (1.14.10) commandto package together all drawings and support files.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SheetContextMenu.png})

Right click a sheet to open the **Sheet** context menu:

- **Open**- opens the drawing with the layout of the selected sheet.
- **Open read-only**- opens the drawing with the layout of the selected sheet in read-only mode to protect it
- **New Sheet**- displays the New Sheet dialog box, which allows to add a new sheet to the sheet set or current subset.
- **Remove Sheet** - deletes the sheet. The deletion is definitive when the modifications are saved when closing the sheet set.
- **Plot** - prints the sheet.
- **Publish**- launches the Publish (6.24.55) command and loads the sheet.
- **eTransmit**- launches the eTransmit (1.14.10) command to package together all drawings and support files.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SheetSelectionContextMenu.png})

Right-click when multiple sheets are selected to open the **Sheet Selection** context menu. Hold the **Ctrl** key to select multiple sheets. Sheets in different subsets of the same sheet set can be selected. Choose **Save Sheet Selection** to save the currently selected sheets; displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
Type a name for the sheet selection, and then click **OK**.

Mouse actions

Double click

- Expands/Collapses a sheet set or a subset.
- Opens a sheet.

Define Properties

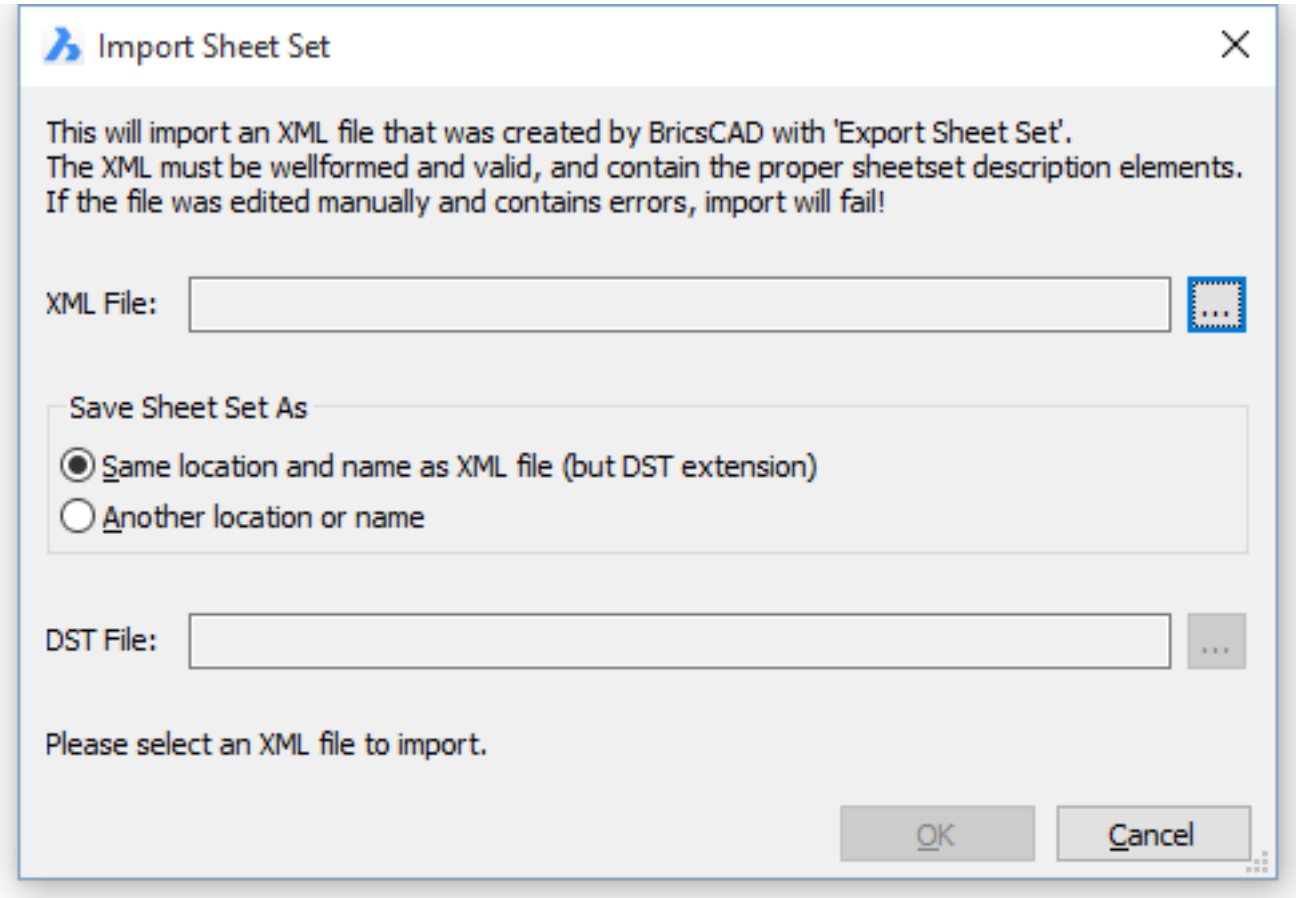
Sheet set properties, subset properties and sheet properties can be used in fields. See the Field (6.14.3) command. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/

Sheet Set properties	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S • Select a field, and then define the property. • **New Sheet Location** - click**Browse**  , andthenspecify a folder. • **Sheet Template Layout** - click**Browse**  , andthenchoose a template drawing and a layout. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou • **Prompt** - choose whether or not to be prompted for a template drawing when creating a new sheet from scratch.
Subset properties	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S • **Name** and **description** - select the field, and then type a name and description. • **Publish Sheets in Subset**- choose an option • **New Sheet Location**- click**Browse**  , andthen specify a folder. • **Sheet Template Layout**- click**Browse**  , andthenchoose a template drawing and a layout.
Sheet properties	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S • **Number**, **Title**, **Description**, **Revision Number**, **Revision Date**, **Issue Purpose** and **Category**- select a field, and then define the property The **Title** field is mandatory; the other fields are optional. • **Publish** - choose whether or not to include the sheet when the sheet set or the subset is published or plotted. • **Layout** - click**Browse** ) and then specify a different drawing and/or layout. • **Sheet Custom Properties** - specify the custom properties for the sheet.

Procedures

- Using sheet set properties in a title block (1.22.11)
- To open or close a panel (1.12.7)
- To dock a panel (1.12.7)

Figure 6.2061



Related Commands

SheetSetHide (6.27.33) - closes the Sheet Set panel. NewSheetSet (6.22.5) - creates a new sheet set. OpenSheetSet (6.23.9) - opens an existing sheet set; displays a file dialog box. -OpenSheetSet (6.23.10) - opens an existing sheet set; prompts you in the command bar. Publish (6.24.55) - allows to print a sheet list (= a list of model space or paper space layouts). Saves a sheet list to a file. -Publish (6.24.56)- prints a saved sheet list (= a list of model space or paper space layouts). -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Figure 6.2062

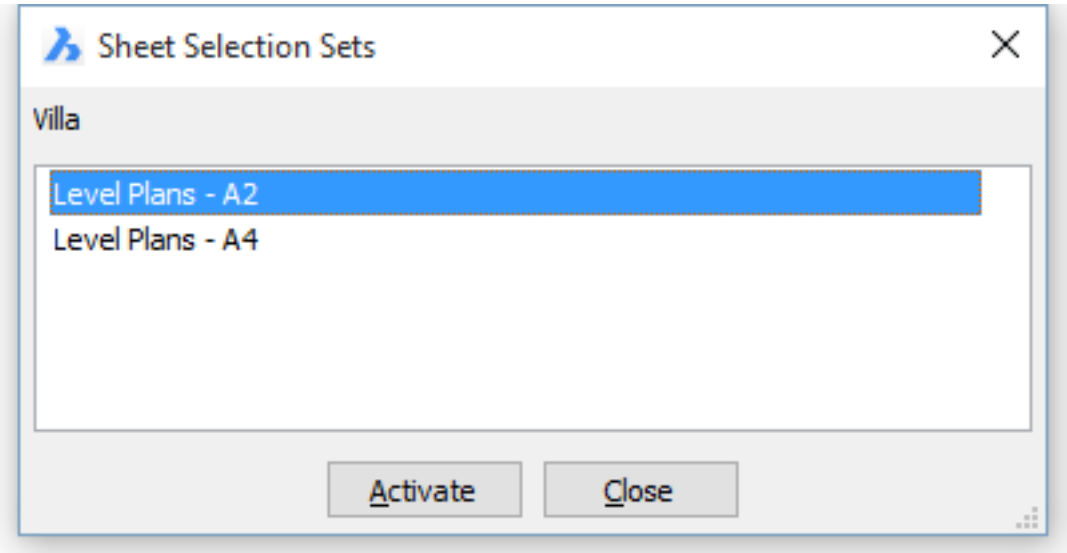


Figure 6.2063

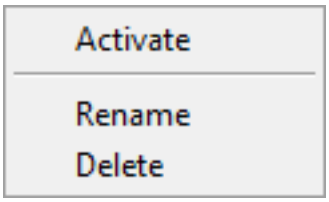


Figure 6.2064

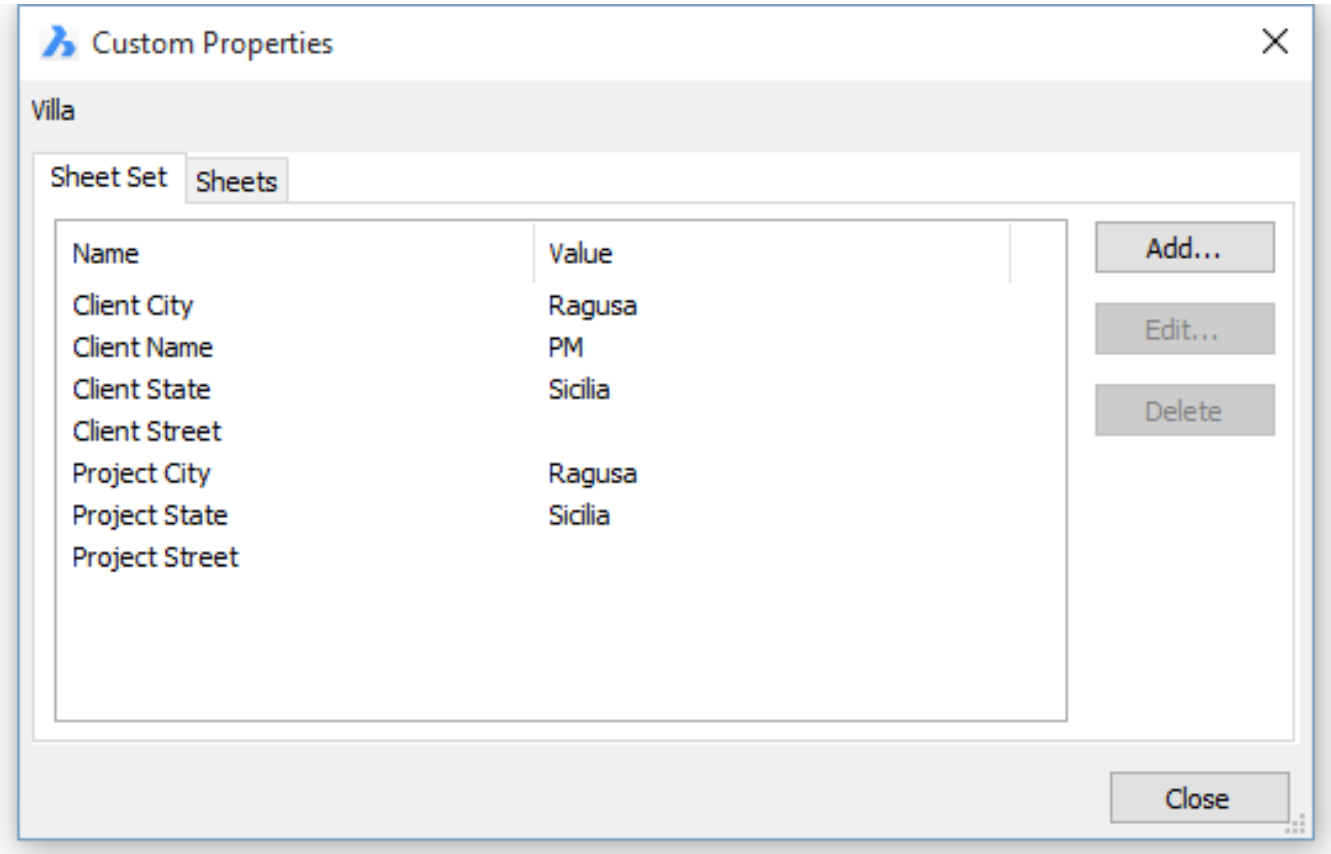


Figure 6.2065

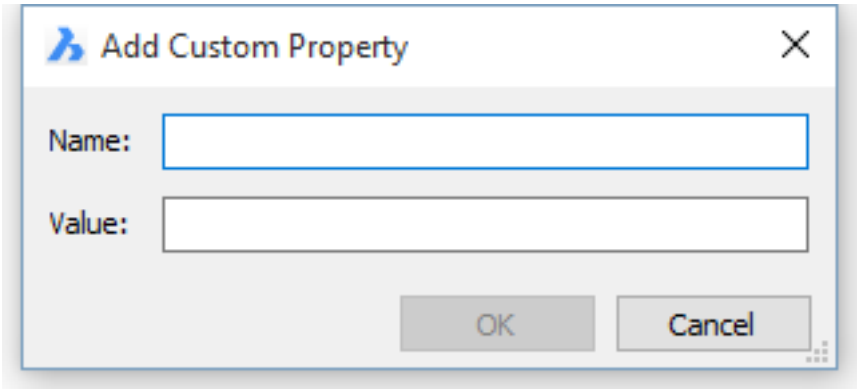


Figure 6.2066

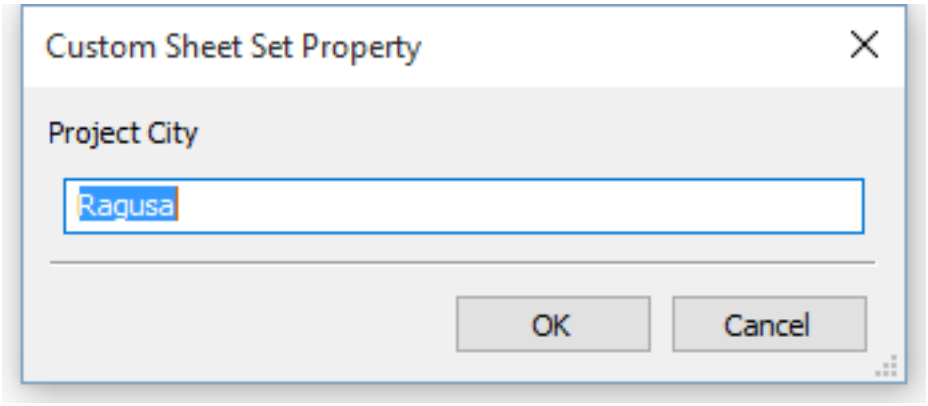


Figure 6.2067

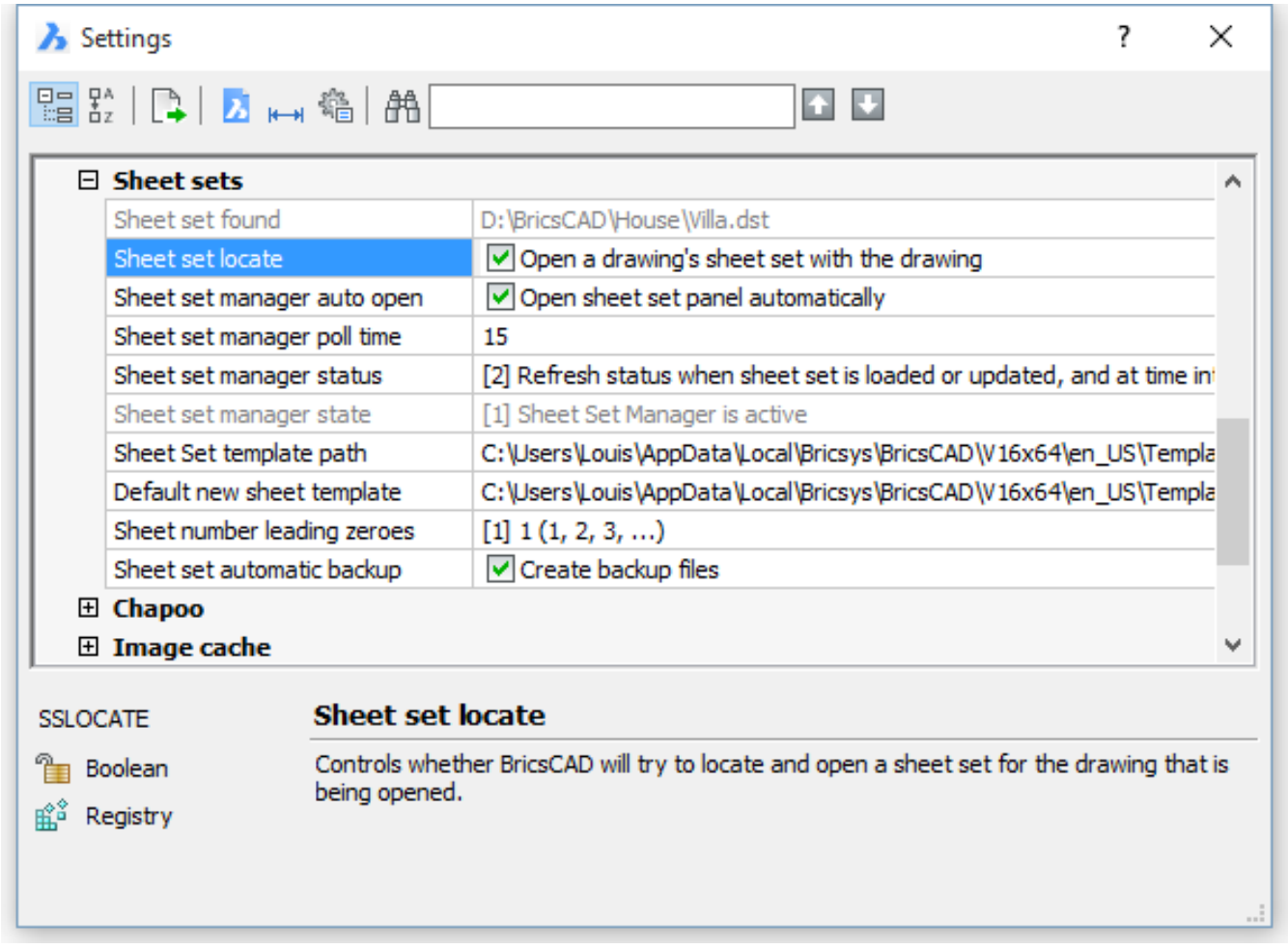
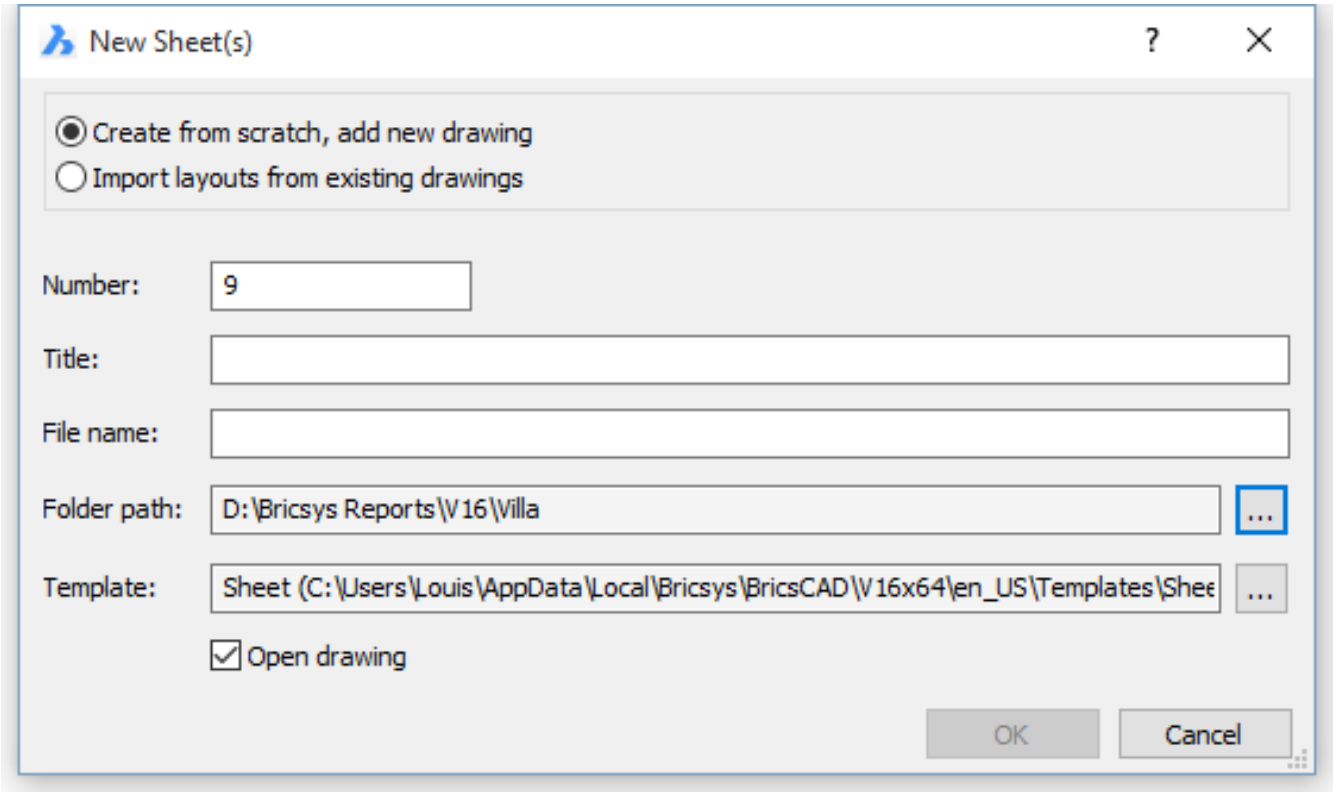


Figure 6.2068



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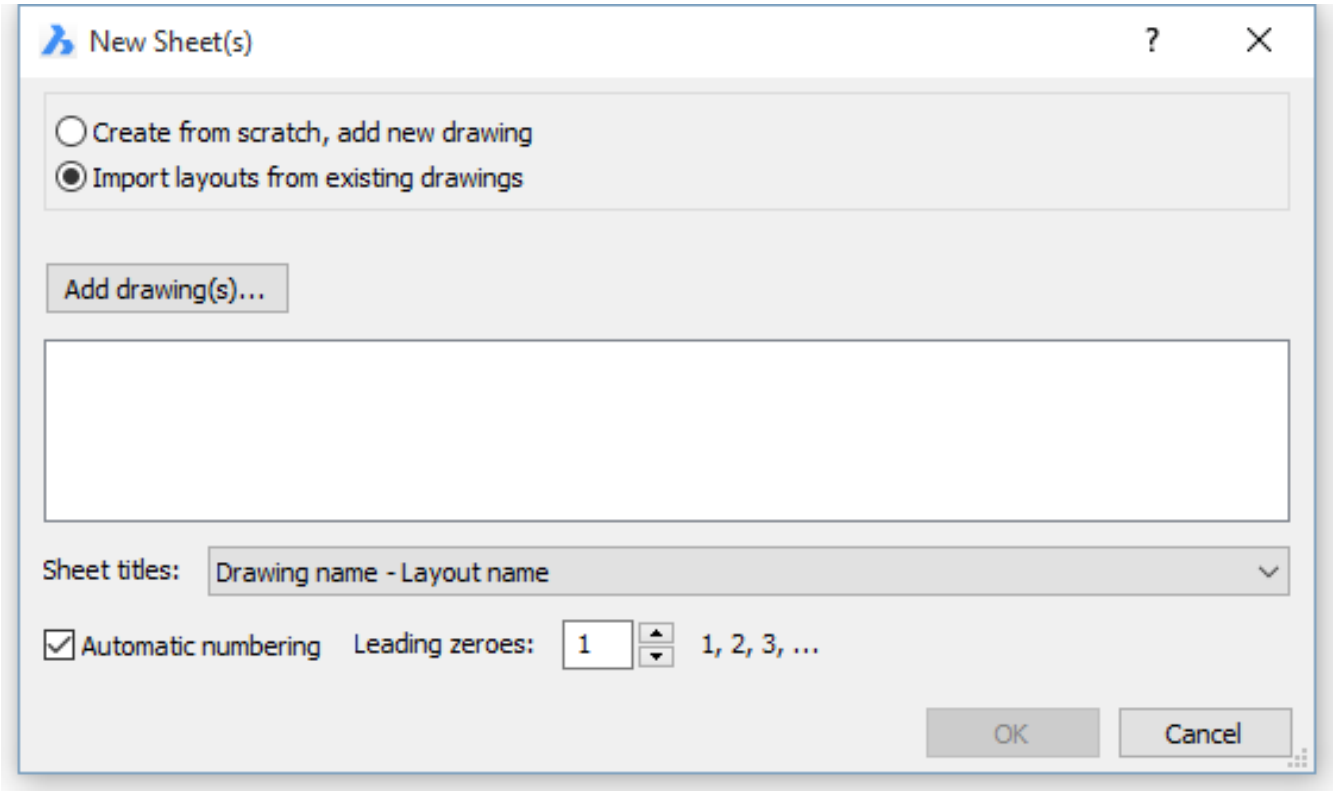
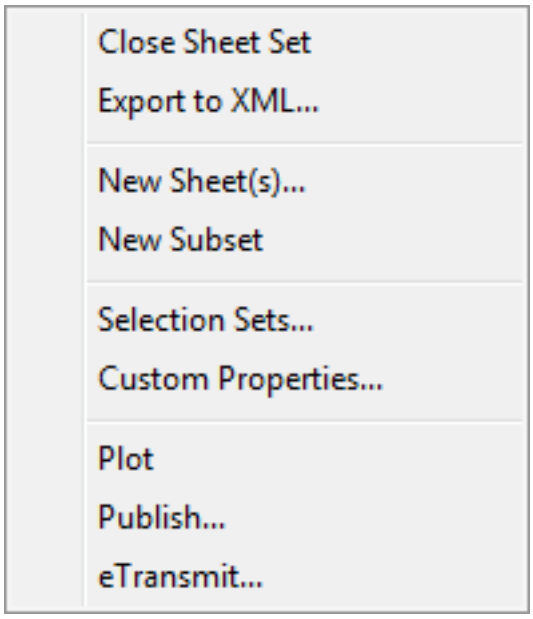


Figure 6.2070



Figure 6.2071



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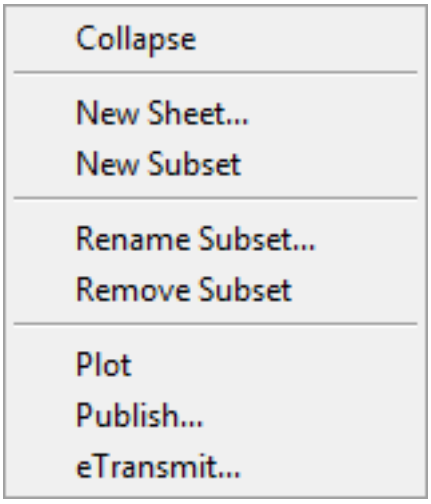

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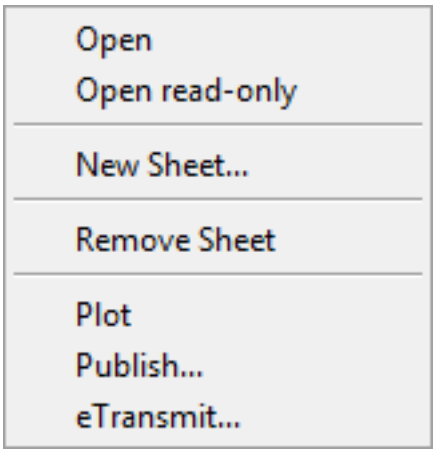

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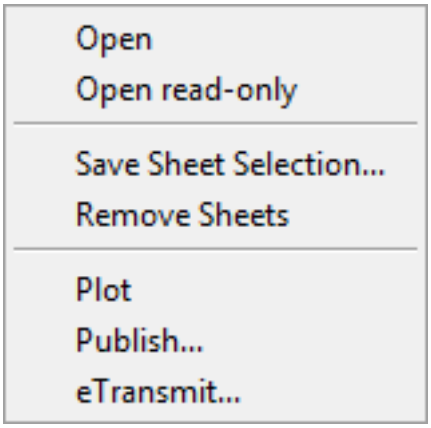

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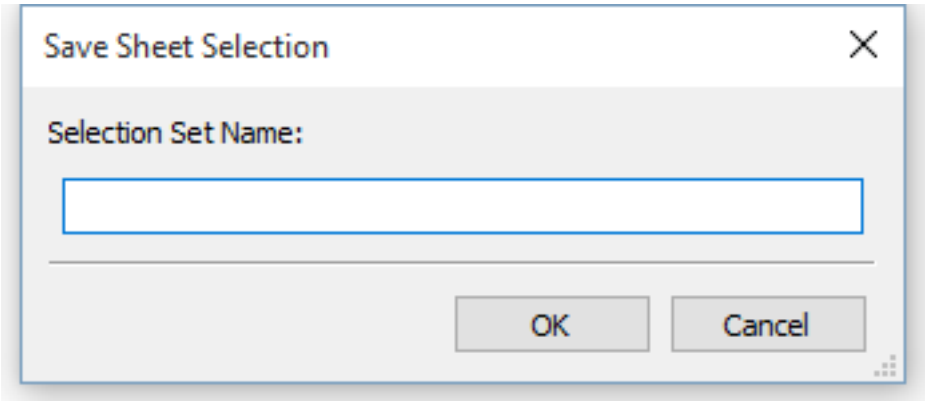


Figure 6.2076

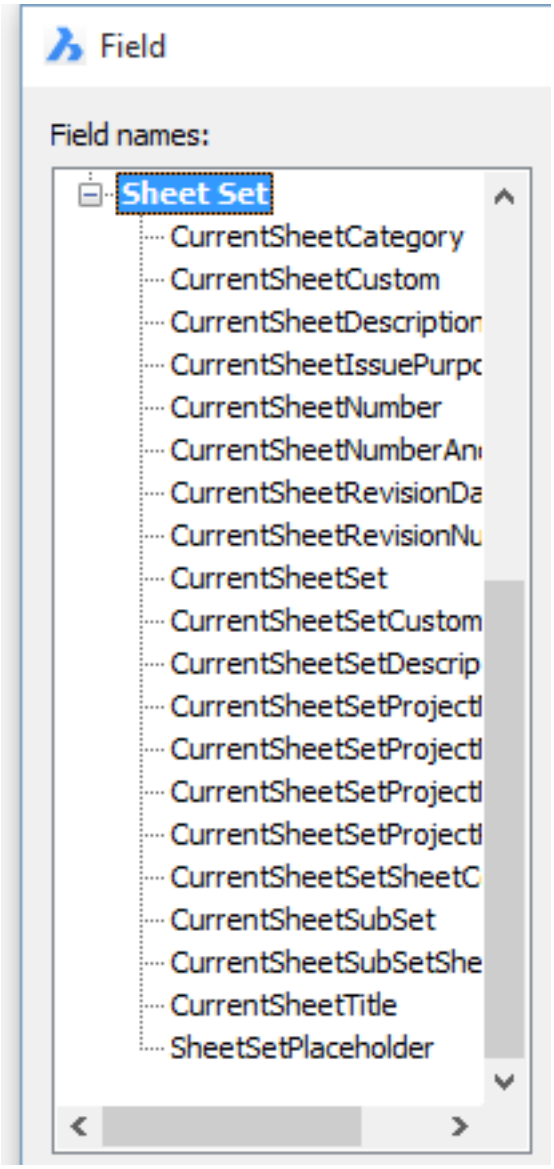


Figure 6.2077

Sheet Set	
Name	FamilyHouse
Description	House in Brighton project
File Path	C:\BricsCAD\House\House.dst
Total Sheets	23
Project Control	
Project Number	
Project Name	
Project Phase	
Project Milestone	
Sheet Creation	
New Sheet Location	C:\BricsCAD\House
Sheet Template Layout	A4_Level0 (C:\Bricscad Training\Bricscad\TrainingCD\EN\Exercises\House.dwg)
Sheet Set Custom Properties	
Address	5 Sea Drive, Brighton
Architect	Michael Brickstone
Owner	Mr. Fisherman
Sheet Custom Properties	
Size	A0

Figure 6.2078

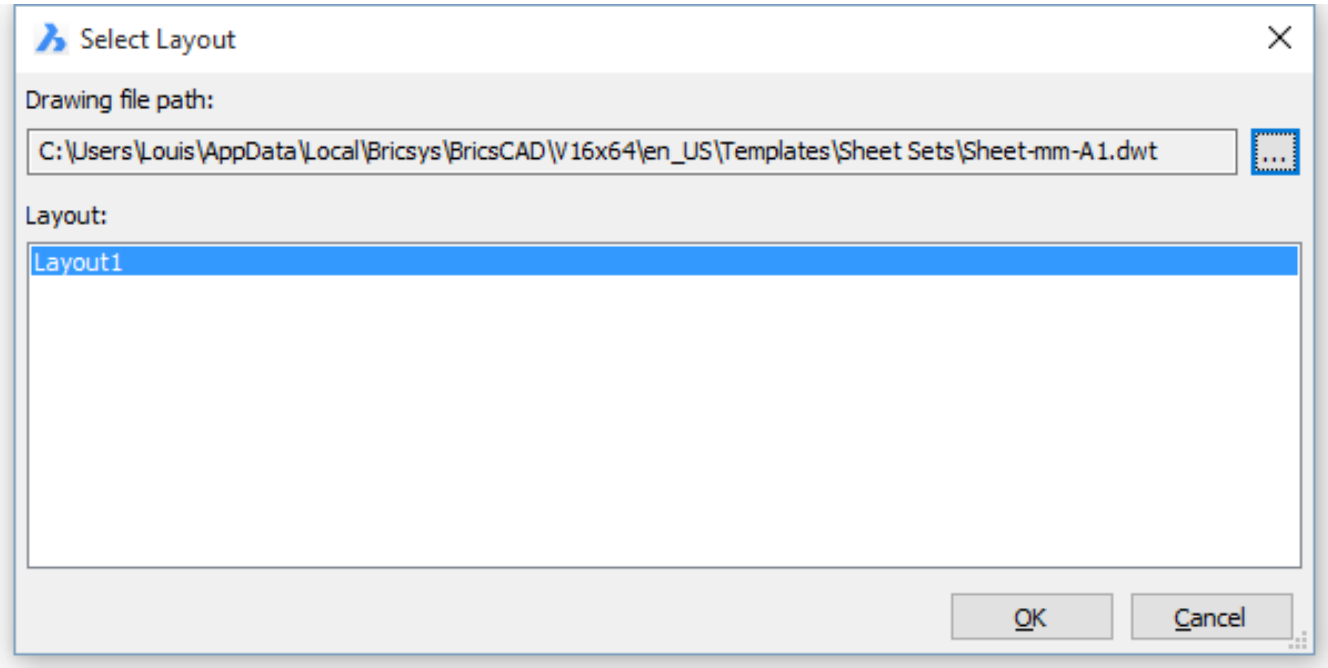


Figure 6.2079

Subset	
Name	Construction
Description	
Publish Sheets in Subset	Publish by sheet 'Include for publish' setting
New Sheet Location	C:\BricsCAD\House
Sheet Template Layout	A4_Level0 (C:\Bricscad Training\Bricscad\TrainingCD\EN\Exercises\House.dwg)
Total Sheets	3

Figure 6.2080

Sheet	
Number	6
Title	Walls_Level1 - LAYOUT1
Description	
Publish	☑ Include for Plot/Publish
Layout	LAYOUT1 (C:\BricsCAD\House\Details\Walls_Level1.dwg)
Revision Number	
Revision Date	
Issue Purpose	
Category	
Sheet Custom Properties	
Size	A0

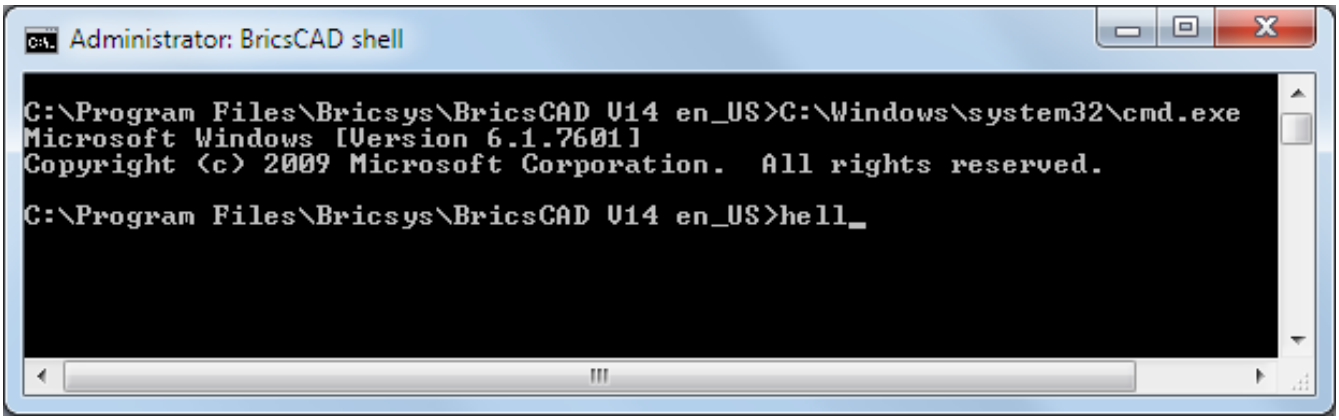
6.27.32 Shell

Opens Windows command prompt window; runs other applications.

Accessing the Command

command bar: **shell** transparent: **'shell** alias: **sh : Shell Prompts you in the command bar:** OS Command: **(Enter the name of a Windows program, or press Enter for the command prompt window.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/WindowsCommandWindow.png}) Enter commands or names of programs, such as 'Notepad'. For a list of operating system commands, enter Help . To exit the window, enter Exit .

Figure 6.2081



Command Options

Related CommandsStart (6.27.44) - starts other applications.Sh (6.27.28) - opens the operating system command prompt window; runs other applications.Files (6.14.5) - displays the operating system's File Manager.

Option	Description
OS Command	Specifies the name of the Windows program to run. Press Enter to open the Windows command prompt window.

Figure 6.2082



6.27.33 SheetSetHide

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes the Sheet Set panel.

Accessing the Command

command bar: **sheetsethide : SheetSetHide** No prompt

Command Options

No options

Related Commands

SheetSet (6.27.31) - manages sheet sets. Opens the Sheet Sets panel. -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Figure 6.2083



Figure 6.2084



6.27.34 Sketch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws freehand sketches made of polylines or lines. (\vref {fig:C:/Dev/Task-2448/images/stat

Accessing the Command

 command bar: **sketch** alias: **freehand** menu bar: **Draw | Sketch** ribbon: **Draw | Draw | Polylines | Sketch** tool-bar: **Draw |**  **: Sketch Prompts you in the command bar:** Sketch: Length of segments <0.1000>: **(Enter a number, pick two points, or else press Enter to accept the default)** Press ENTER or click to toggle [Erase/Connect/save and eXit/Quit no save/Record (save)]: **(Click to start sketching, click again to toggle pen up/down)** Press ENTER or click to toggle [Erase/Connect/save and eXit/Quit no save/Record (save)]:**(Enter an option, or press Xto exit the command)** During sketching, the lines are green until the sketch lines are saved: Saving sketch lines ... Depending of the status of the SKPOLY system variable sketches are made from lines or polylines.

Command Options

Option	Description
Length of segments	Specifies the length of segments.Enter a number.Too small a segment length results in large drawings.
Press ENTER or click to toggle	Allows you to move the pen without drawing: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Erase	Erases the sketch lines from their endpoints interactively. Click to stop erasing and restart sketching. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Connect	Connects cursor to end of the last segment; prompts you: **Point cursor to end of last sketch line** - **(move the cursor to the end of the last sketch.)** This option is meant for use after a pen up.
save and eXit	Saves the sketch; reports: **Saving sketch lines.** Exits the sketch command.
Quit no save	Exits the command without saving the sketch.
Record (save)	Saves the sketch; reports: **Saving sketch lines.** The green lines turns black. You can continue sketching.

Procedures

- Setting the Sketch Poly (SKPOLY) system variable (1.17.14)
- Creating a freehand sketch (1.17.14)

PLine (6.24.27) - draws polyline segments and arcs. RevCloud (6.26.29) - draws revision clouds.

Figure 6.2086

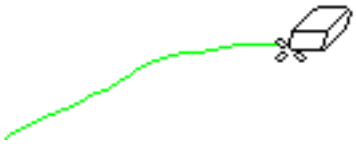


Figure 6.2087



6.27.35 Singleton

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Determines whether (or not) multiple copies of BricsCAD can run at the same time.

Accessing the Command

command bar: **singleton** transparent: **'singleton : Singleton Prompts you in the command bar:** Allow only single BricsCAD application instance? [Yes/No] <No>: (**Press Y or N)** When set to no, only a single instance of BricsCAD can run.When set to yes, you can launch two or more copies of BricsCAD at the same time.

Command Options

Option	Description
Allow only single BricsCAD application instance	Toggles single instance mode of BricsCAD: • **Yes** - allows only one copy of BricsCAD to run • **No** - allows two or more copies of BricsCAD to run Some third-party applications run only in a single instance of BricsCAD.

Related Commands

None.

Figure 6.2088



6.27.36 Slice

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Cuts a 3D solid with a plane into two parts. (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc

Accessing the Command


command bar: **slice** alias: **sl** menu bar: **Model | 3D Solids | Slice ribbonP** toolbar: **3D Solids : Slice Prompts you in the command bar:** Select entities: **(Choose one or more 3D solids)** Entities in set: 1 Select entities: **(Choose more 3D solids, or press Enter to continue)** Specify first point on slicing plane or [Object/Surface/Zaxis/View/XY/YZ/ZX/Line-point/3points/Multislice] <3points>: **(Pick a point or enter an option)**

Command Options

Option	Description
Select entities	Specifies the entities that will be sliced.Choose one or more 3D solids; this command does not work with 2D regions.
Specify first point on slicing plane	Slices a solid with a plane defined by two points; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S 1. **Specify first point on slicing plane- (pick a point, such as #1 in the figure, or enter x,y,z coordinates.)** 2. **Specify second point on slicing plane- (pick another point such as #2)** 3. **Specify a point on desired side of the plane or Both- (enter an option)**: • Choose one of the parts, the other is erased • **B**- retains both.The section plane is perpendicular to the XY-plane of the World coordinate system (WCS): The sliced solid can be separated and its two parts manipulated as independent entities: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

Object	Slices a solid with entity that intersects its; the entity defines the plane that slices the 3D solid. Prompts you: **Select a circle, ellipse, arc, 2D-spline, 2D-polyline, planar 3d Solid face, planar Surface face, or planar region- (choose an entity.In addition to the entities listed by the prompts, you can also use an elliptical arc.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S To use a planar 3D solid face as the slicing plane, ensure thatthe **Select Faces** option of the **SELECTIONMODES** system variable is on. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/((\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
Surface	Slices the 3D solid with a surface; prompts you: **Select a surface** - **(select a surface entity.)**
Zaxis	Slices the 3D model with a plane defined by one point on the plane, and a second point on the plane's normal (z axis); prompts you: 1. **Specify a point on the section plane** - **(pick a point to define the location of the section plane (1).)** 2. **Specify a point on the Z-axis (normal) of the plane** - (**pick another point to define the normal (2).)**The section plane is perpendicular to the line through the specified points. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S The section plane is defined by the centroid of the front face of the cube (1) and the Z axis through this point and a corner of the cube (2).The section plane is perpendicular to line 1-2.
View	Slices the 3D model with a plane defined by the current 3D viewpoint; prompts you: **Specify a point on the current view plane** - **(pick a point to define the height of the plane.)** The section plane is perpendicular to the view direction and passes through the picked point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
XY	Slices the 3D model with a plane parallel to the XY-plane of the current coordinate system; prompts you: **Specify a point on the XY-plane** - **(pick a point to define the position of the slicing plane.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
YZ	Slices the 3D model with a plane parallel to the YZ-plane of the current coordinate system; prompts you: **Specify a point on the YZ-plane** - **(pick a point to define the position of the slicing plane.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

ZX	Slices the 3D model with a plane parallel to the ZX-plane of the current coordinate system; prompts you: **Specify a point on the ZX-plane** - **(pick a point to define the position of the slicing plane.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
Line-point	Slices the 3D model with a plane defined by a linear entity and a point; prompts you: 1. **Select a line, 2d-polyline segment, linear edge of 3dSolid or surface, or axial entity to use its axis** - **(select a linear entity.)** 2. **Specify point on slicing plane** - **(pick a point.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
3points	Slices the solid with a plane defined by three points; prompts you: 1. **Specify first point on plane** - **(pick a point or enter x,y,z coordinates.)** 2. **Specify second point on plane** - **(pick another point.)** 3. **Specify third point on plane** - **(pick a third point.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
Multislice	Slices 3D solids in more than one place using a face; prompts you: 1. **Select a planar Face or a planer Surface or <XY>- (pick a face.) Specify distance to create slice- (pick a slice location.) Specify distance to create slice or Repeat [Repeat]- (pick another slice location,or else press Enter to exit the command.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/sli multislice.png})
Specify a point on desired side of the plane or Both	Specifies which of the two portions to keep: • Choose one of the parts; the other one is erased • **B** - retains both parts

Related Commands

BimSplit (6.7.32) -splits segmented solids into separated solids and allows to split a solid using a selection of cutting faces. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. Region (6.26.18) - creates 2D regions from 2D entities. Section (6.27.11) - creates 2D regions from 3D models. Subtract (6.27.56) - removes one solid model from a second. Union (6.30.6) - joins two or more solid models into a single entity.

Figure 6.2089

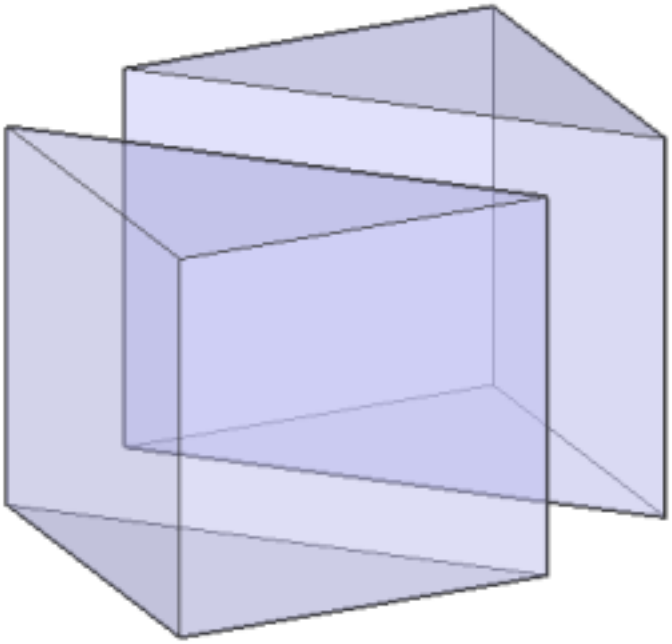


Figure 6.2090

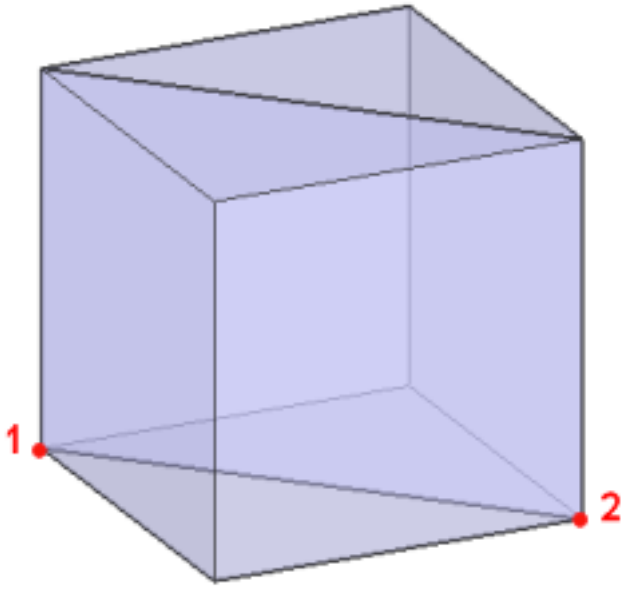


Figure 6.2091

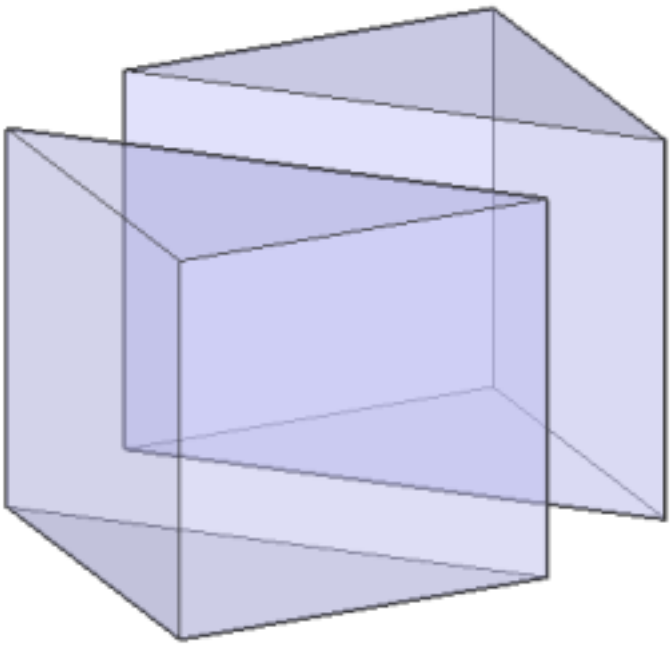


Figure 6.2092

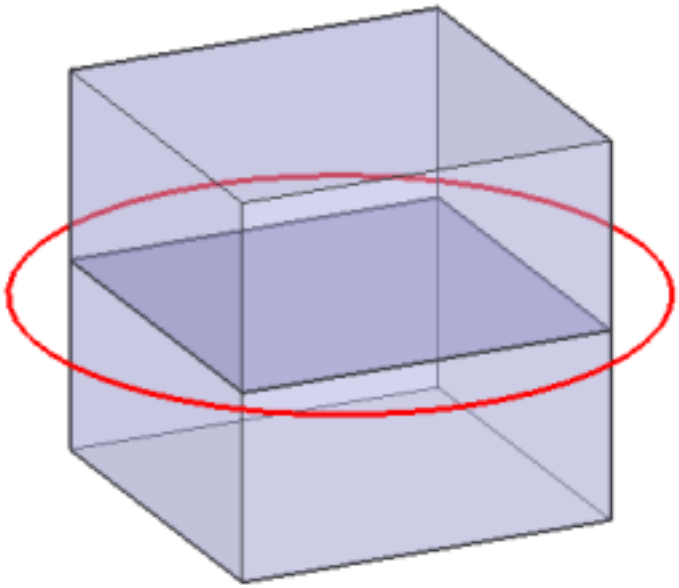
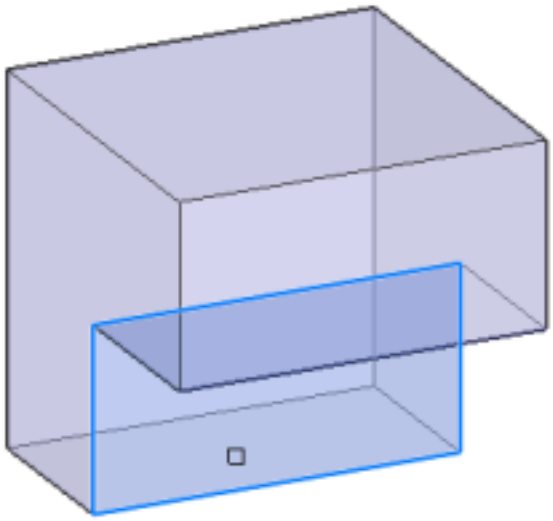


Figure 6.2093



Figure 6.2094



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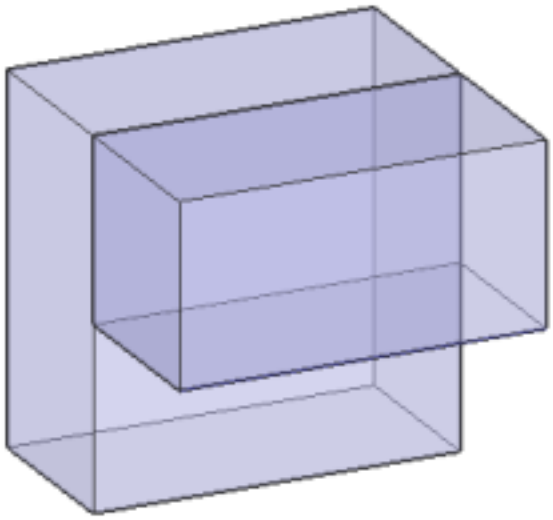

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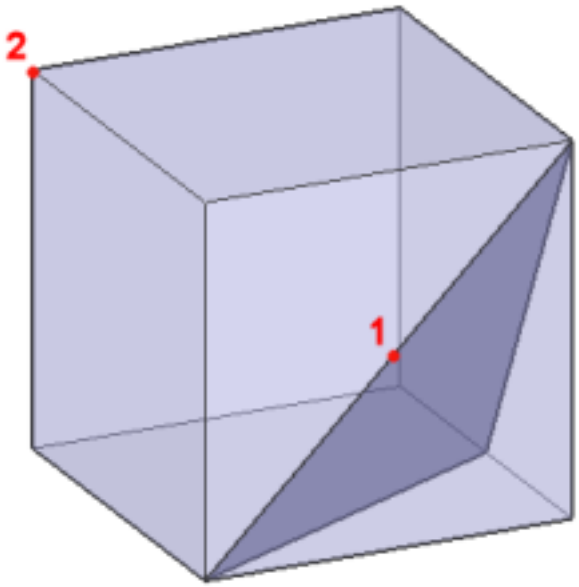

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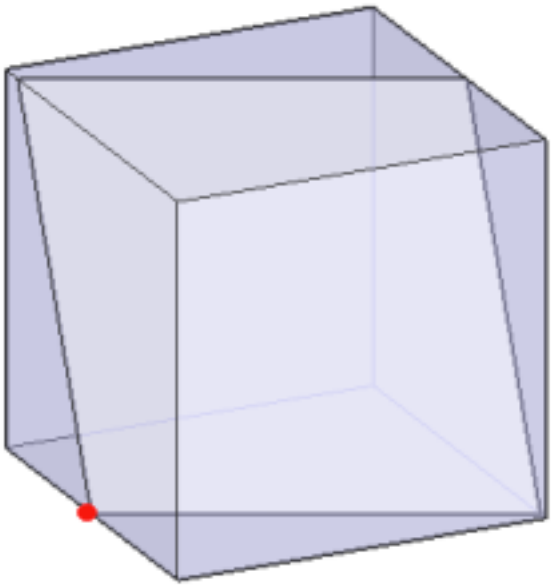
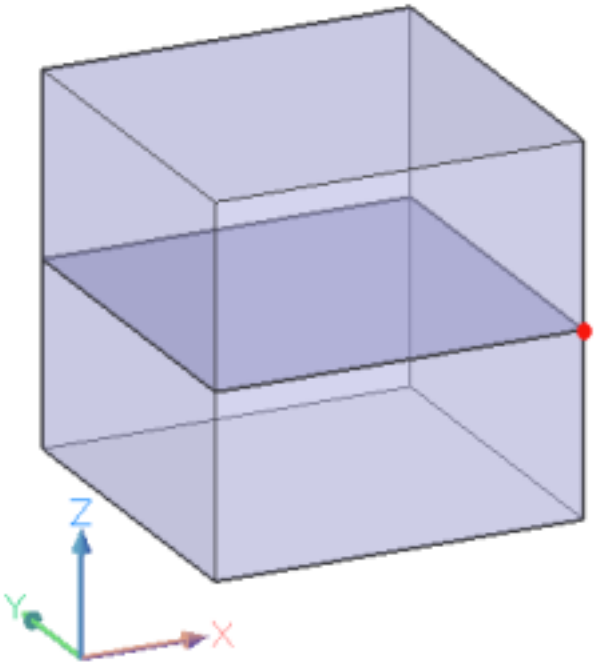


Figure 6.2098



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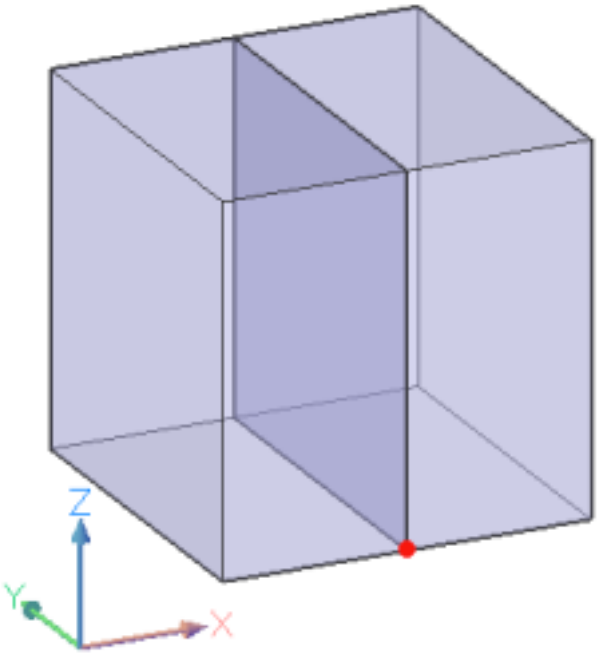


Figure 6.2100

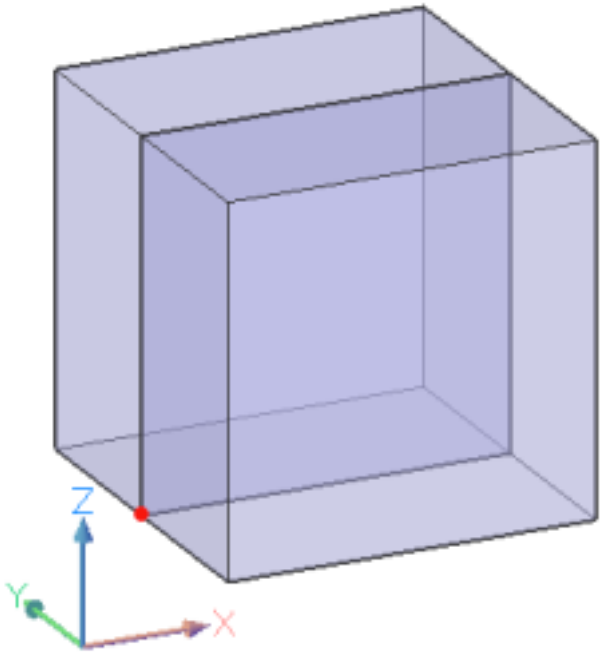


Figure 6.2101

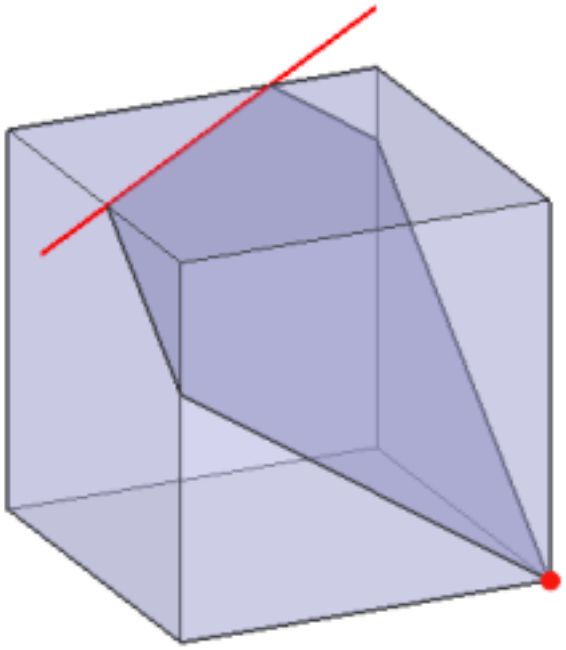


Figure 6.2102

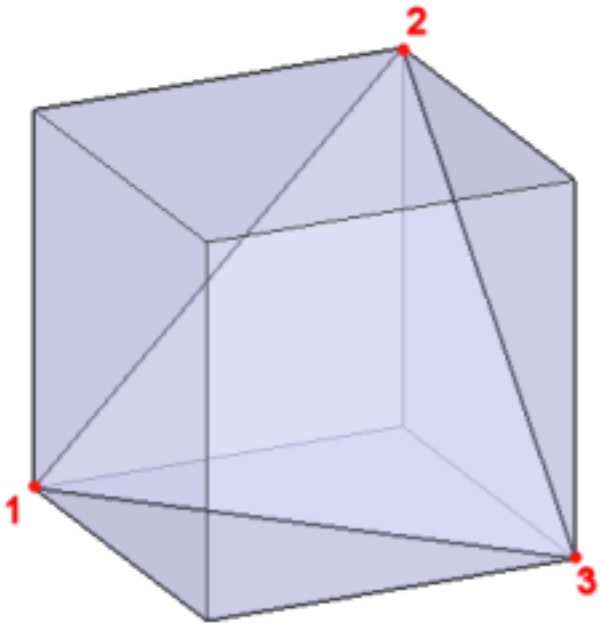


Figure 6.2103

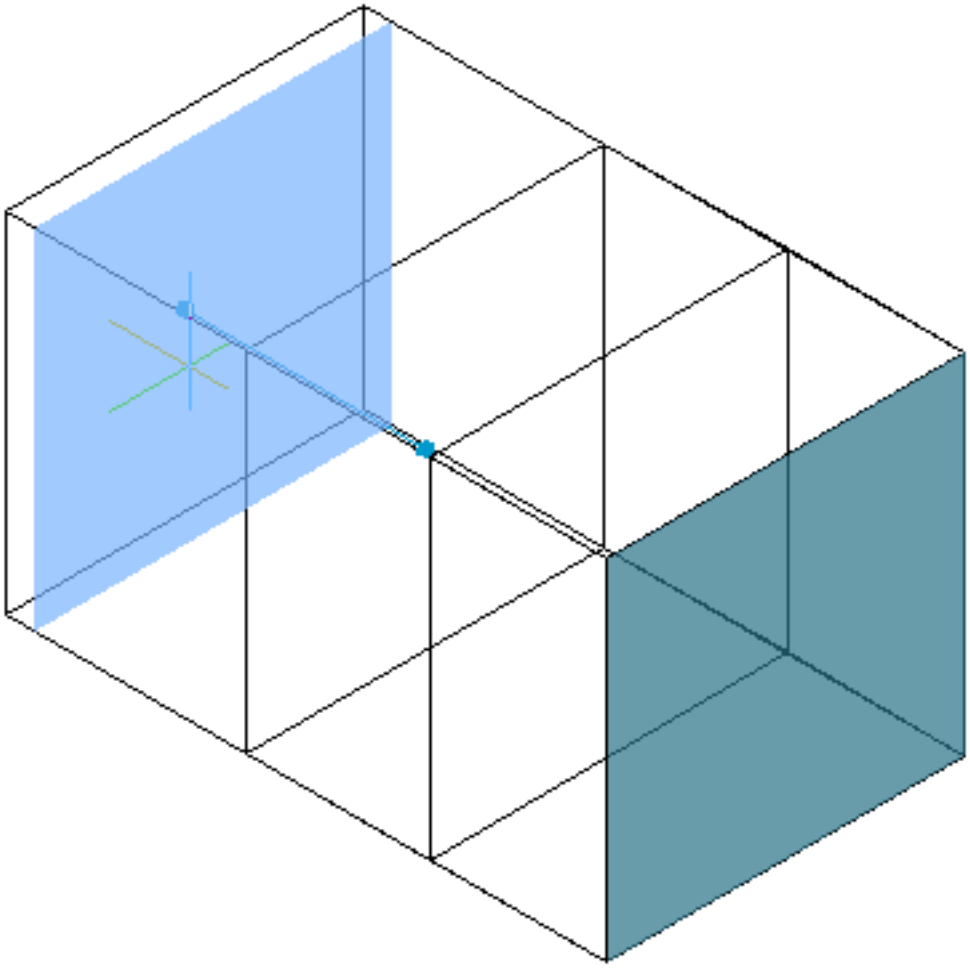
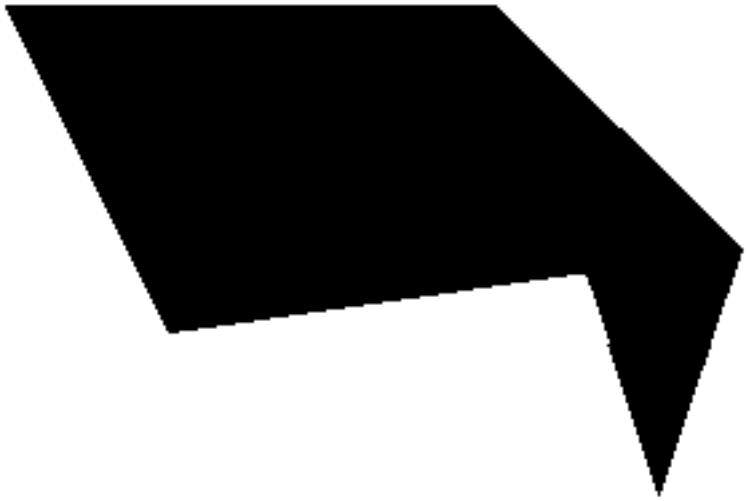


Figure 6.2104



Figure 6.2105



6.27.37 Solid

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws 3 and 4-sided solid-filled regular and irregular 2D polygons; does not draw 3D solids. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/solid-eg.png})

Accessing the Command


command bar: **solid** aliases: **plane so** menu bar: **Draw | Solid** ribbon: **Draw | Draw | Polygons | Solid** toolbar: **Draw | Polygons |**  **: Solid Prompts you in the command bar:** Set first point of plane or [Rectangle/Square/Triangle]: **(Pick a point, or enter an option)** Second point: **(Pick another point)** Third point of plane: **(Pick another point)** Fourth point: **(Pick another point to draw a quadrangle, or else press Enter to draw a triangle and exit the command)** Third point of plane: **(Pick another point to draw an adjoining triangle, or press Enter to exit the command.)**

Command Options

Option	Description
Set first point of plane	Specifies the first corner of the solid-filled polygon: • Pick a point • Enter x,y coordinates After picking this point, the program draws a ghost line to the next point.
Second point	Specifies the second corner of the polygon. The programpreviews the shape and size of the polygon.
Third point of plane	Specifies the third corner of the polygon. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Fourth point	To draw a solid-filled triangle, press **Enter** at this prompt; otherwise, specify the fourth corner of the polygon. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Third point on plane	Starts drawing an adjacent polygon.The third and fourth points of the first polygon become the first and second points of the second polygon. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i To end the command, press **Enter** at the this prompt.
Rectangle	Draws one or more connected solid-filled rectangles; prompts you: 1. **First point of plane** - **(pick a point or enter x,y coordinates.)** 2. **Other corner of rectangle** - **(pick another point.)** 3. **Rotation angle for plane** - **(pick a point, or enter an angle.)** 4. **Other corner of rectangle** - **(pick a point, or else press Enter to stop.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Square	Draws one or more connected solid-filled squares; prompts you: 1. **First point of plane** - **(pick a point or enter x,y coordinates.)** 2. **Second corner of square** - **(pick a point.)** 3. **Second corner of square** - **(pick a point, or else press Enter to stop.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Triangle	Draws one or more connected equilateral solid-fileld triangles; prompts you: 1. **First point of plane** - **(pick a point or enter x,y coordinates.)** 2. **Second corner of triangle** - **(pick a point; the cursor's movement determines the orientation of the triangle:)** • **Right** - triangle is drawn upside down. • **Left** - triangle is drawn right side up. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 3. **Second corner of triangle** - **(pick a point, or else press Enter to stop.)**

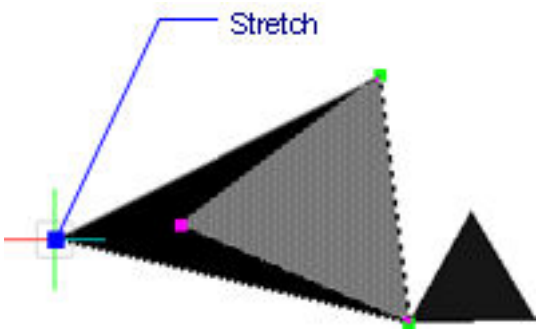
Grips Editing

2D solids can be edited directly through grips:

1. Select the solid. Notice that it has one grip at each vertex.
2. Drag a grip to stretch the solid.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image479.jpg})

Figure 6.2106



Procedures

- Drawing Solids (1.17.23)

Related Commands

Fill (6.14.6) - toggles the display of solid areas. Trace (6.29.24) - draws wide lines. BHatch (6.6.5) and Hatch (6.16.1) - place solid-filled hatch patterns through a dialog box. -BHatch (6.6.6) and -Hatch (6.16.2) - place solid-filled hatch patterns through the command bar. Boundary (6.6.15) and -Boundary (6.6.16) place a polyline boundary inside closed areas. Properties (6.24.49) - changes the properties of solid.

Figure 6.2107

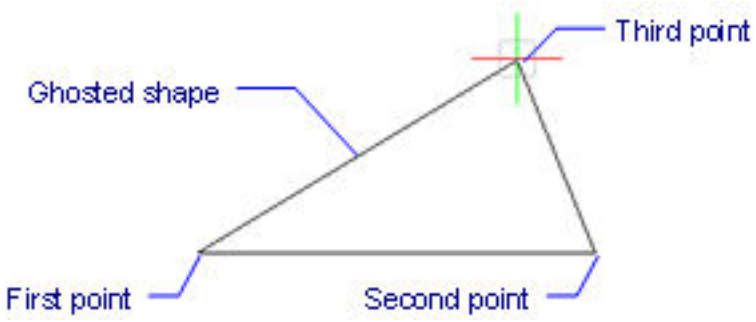


Figure 6.2108

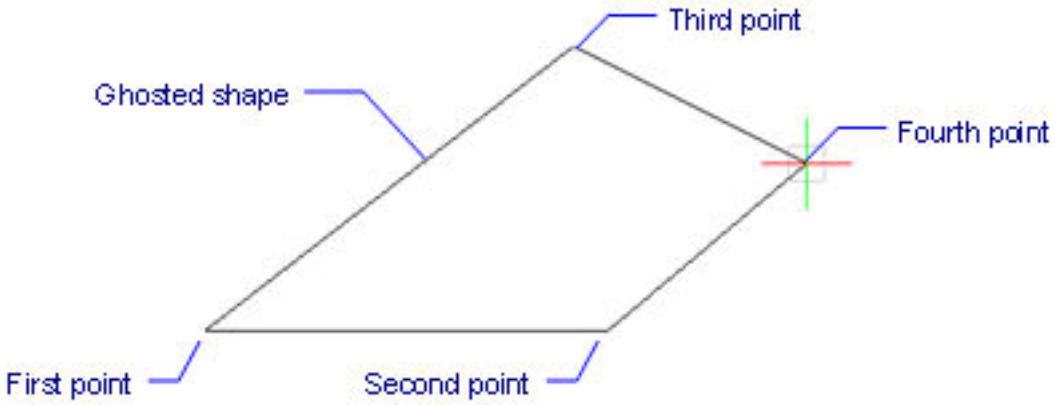


Figure 6.2109

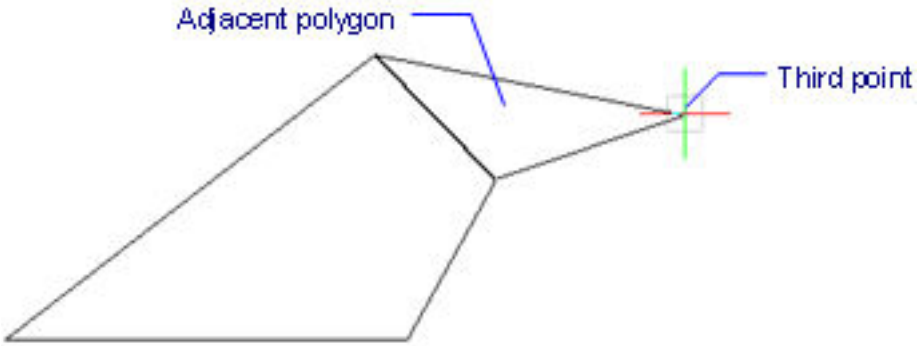


Figure 6.2110

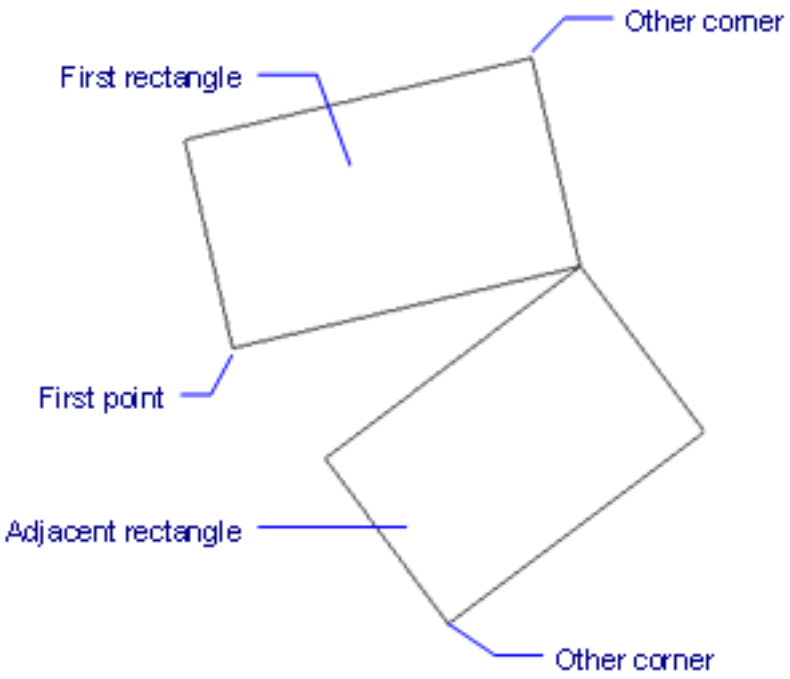


Figure 6.2111

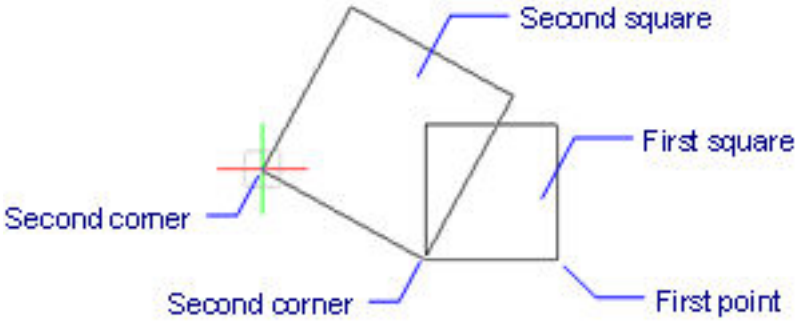


Figure 6.2112

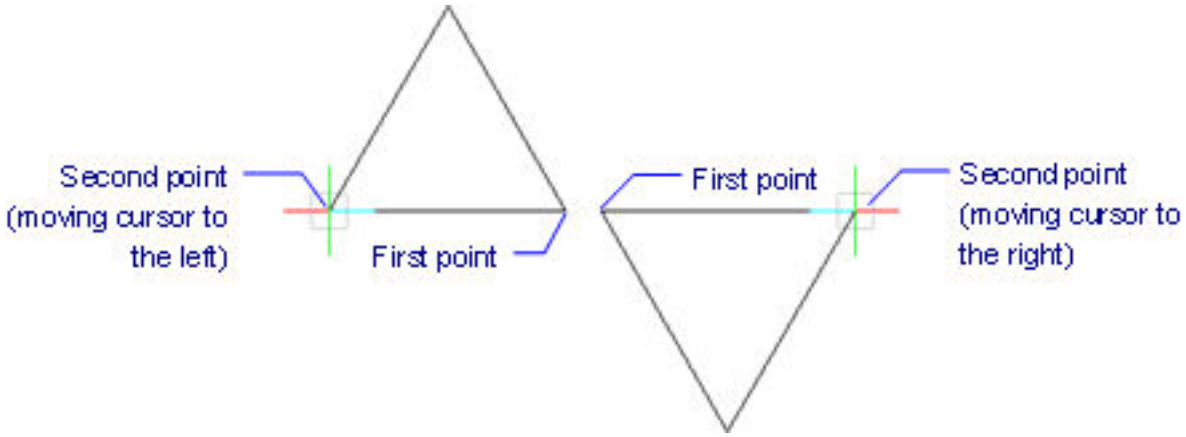


Figure 6.2113



6.27.38 Snap

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles snap mode for setting cursor movement increments, and sets properties for snap distances.

Accessing the Command

 command bar: **snap** transparent: **'snap** shortcut: **F9** **Ctrl+B** alias: **sn** menu bar: **Settings | Snap** ribbon: **Settings | Snap** status bar: **SNAP** toolbar: **Settings |**  **: Snap Prompts you in the command bar:** Snap is off (x and y = 10): Set snap spacing or [snap ON/Rotate/snap Style/Aspect]: **(Enter a number for spacing, or enter an option)** Snap acts as a cursor resolution, constraining cursor movement to discrete distances, such as 0.5".

Command Options

Option	Description
Set snap spacing	Specifies the distance the cursor moves: • Enter a single number to specify the spacing between snap points • Pick a point, or else enter x,y coordinates to show the snap spacing; prompts you: **Second point** - **(pick another point, or enter x,y coordinates.)**
snap ON	Turns on snap in the current viewport, and so the cursor's movement jumps by the snap increment, with some exceptions: • Snap must be turned on in each viewport separately • When DRAGSNAP is off, snap is only in effect during drawing and editing commands; when DRAGSNAP is on, the cursor also snaps when no command is active. • When SNAPSTYL is on, the cursor snaps instead to the isometric grid • When the SNAPTYPE variable = 1, the cursor snaps instead to the polar (radial) distance specified by the POLARDIST variable. • When the SNAPTYPE variable = 2, the cursor's snap distance is adjusted by the AdapticeGripdStepSize variable.
snap OFf	Turns off snap.

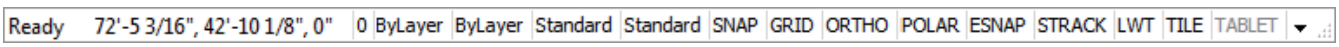
Rotate	Rotates the snap grid; prompts you: 1. **Base point for snap grid** - **(specify the point about which the snap grid rotates; pick a point or enter x,y coordinates.)** 2. **Rotation angle** - **(specify the angle to which to rotate the snap grid.)** This option also rotates the grid, the crosshair cursor, and the orthographic axes. It does not affectthe WCS or the UCS icon.
snap Style	Toggles between standard and isometric style; prompts you: **[Standard/Isometric] <Standard>** - **(enter an option:)** • **Standard** - rectangular snap distances • **Isometric** - sets grid, snap distance and crosshair cursor to the current isometric plane (determined by the SNAPISOPAIR variable), and adds the Isocircle option to the Ellipse (6.13.5) command.
Aspect	Specifies the snap spacing independently in the x and y directions; creates a rectangular snap grid. Prompts you: 1. **Horizontal spacing** - **(specify the space between snap points in the x direction.)** 2. **Vertical spacing** - **(specify the space between snap points in the y direction.)**

Status Bar Menu

Right-click SNAP on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image791.png})

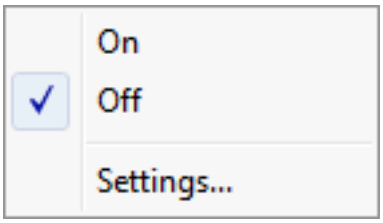
Shortcut Menu	Description
On	Turns on snap.
Off	Turns off snap.

Figure 6.2114



Settings	Opens the Snap/Grid section of the Settings dialog box: **Snap angle** - sets the angle of the snap, grid display, and ortho mode in the current viewport, relative to the current UCS. **Snap base** - specifies the x,y-coordinates about which the snap rotates. **Snap isometric pair** - specifies the current isometric plane (requires Snap Style to be set to Isometric Snap; see theIsometric (6.17.22)command): • **Left** - left isometric plane • **Top** - top isometric plane • **Right** - right isometric plane **Snap mode** - toggles snap mode: • On- turns on snap • Off- turns off snap **Snap style** - specifies the snap style; click the droplist and choose one: • **Rectangular snap** - regular snap mode • **Isometric snap** - isometric mode **Snap type** - specifies the snap type; click the droplist and choose one: • **Grid snap** - uses distances along the x and y axes • **Polar snap** - uses relative distance and angle **Snap unit** - specifies the snap increment distance: • **X** - spacing in the x direction • **Y** - spacing in the y direction There is no snap in the z direction.

Figure 6.2115



Procedures

- Using snap and grid (1.15.11)

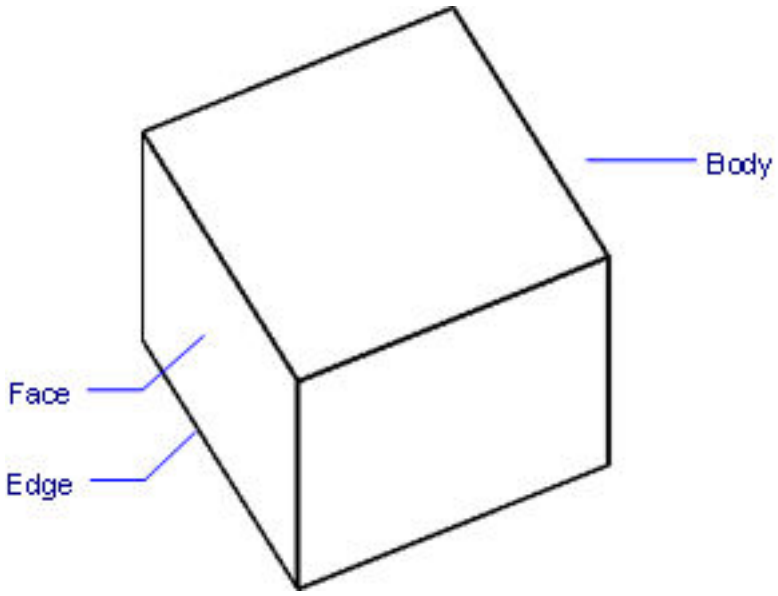
Related Commands

Settings (6.27.23) - sets parameters related to snap mode. Grid (6.15.21) - controls the display of the grid. Isoplane (6.17.22) - switches to the next isometric plane.

Figure 6.2116



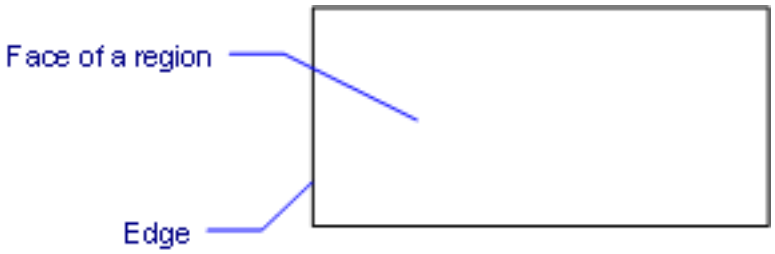
Figure 6.2117



6.27.39 SolidEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Edits the faces, edges, and bodies of 3D solids and 2D regions. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image427.jpg}) (\vref {fig:C:/Dev/Task-2448/images/static

Figure 6.2118



Accessing the Command

command bar: **solidedit** menu bar: **Model | 3D Solid Editing | Extrude - Copy Faces Model | 3D Solid Editing | Extrude Copy, Color Edges Model | 3D Solid Editing | Clean, Separate, Shell, Check** ribbon:**Solid | Edit | Shell** toolbar: **3D Solid Editing** |(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ART/0112SolidEdit : **SolidEdit Prompts you in the command bar:** Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: **(Enter an option, or press Enter to end the command)**

Figure 6.2119



Face Editing Options

To access these options, enter the Face option: Enter a face editing option: Extrude/ Move/ Rotate/ Offset/ Taper/ Delete/ Copy/ coLor/Undo/<eXit>: **(Enter an option, or press Enter to return to the previous prompt)**

 Extrude

Extrudes the selected face or region; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Select faces: Remove/Undo/ALL** - **(choose the face(s) to change:)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

- Pick one or more faces)
- **Remove** - removes faces from the selection set
- **Undo** - undoes the last selection operation
- **ALL** - selects all faces in the drawing
- **Enter** - continues the command to the next prompt:

2. **Specify height of extrusion or Path** - **(specify a distance, or pressP to select an entity that defines the path:)**

- **Positive distance** - extrudes outwards from the face (creates a post)
- **Negative distance** - extrudes into the body (creates a hole)

3. **Specify angle of taper for extrusion** - **(specify an angle, or enter 0 for no taper:)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

- **Positive angle** - tapers inwards (gets narrower)
- **Negative angle** - tapers outwards (gets wider)

Path - Specifies the entity for defining the extrusion shape and direction; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Select extrusion path** - **(choose a open entity, such as line, arc, polyline, or spline.)** See the Extrude (6.13.21) command.

 Move

Moves the selected face; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

1. **Select faces: Remove/Undo/ALL** - **(choose one or more faces to move.)**
2. **Select faces: Remove/Undo/ALL** - **(choose more faces, or else press Enter to continue.)**
3. **Specify <Base point>/Vector** - **(pick a point, enter x,y,z coordinates, or press V.)**
4. **Specify end point** - **(pick the destination point.)**

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Vector** - specifies the displacement through a vector measured from the origin (0,0,0) to the cursor's pick point; prompts you: **Specify end point** - **(pick a point or enter x,y,z coordinates.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i See theMove (6.21.30) command.

Rotate

Slants the selected face about an axis; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

1. **Select faces: Remove/Undo/ALL** - **(choose one or more faces to rotate.)**
2. **Select faces: Remove/Undo/ALL** - **(choose more faces, or else press Enter to continue.)**
3. **Specify the base point** - **(pick the first point of the axis of rotation.)**
4. **Specify another point along the axis** - **(pick the axis of rotation's second point.)**
5. **Specify the rotation angle** - **(enter an angle:)**

- **Positive angle** - face rotates counterclockwise around the axis.
- **Negative angle** - face rotates clockwise.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
See theRotate (6.26.34) command.

Offset

Extends the selected face; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

1. **Select faces: Remove/Undo/ALL** - **(choose one or more faces to offset.)**
2. **Select faces: Remove/Undo/ALL** - **(choose more faces, or else press Enter to continue.)**
3. **Specify offset distance** - **(enter a distance:)**

- **Positive distance** - face is offset outwards (3D body becomes larger)
- **Negative distance** - face is offset inwards (3D body becomes smaller)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
See theOffset (6.23.3) command.

Taper

Tapers (slants) the selected face; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

1. **Select faces: Remove/Undo/ALL -(choose one or more faces to taper.)**
2. **Select faces: Remove/Undo/ALL - (choose more faces, or else press Enter to continue.)**
3. **Specify the base point - (pick the first point of the axis of rotation.)**
4. **Specify another point along the axis - (pick the axis of rotation's second point.)**
5. **Specify the rotation angle - (enter an angle:)**

- **Positive angle** - face rotates counterclockwise around the axis.
- **Negative angle** - face rotates clockwise.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Delete

Deletes the selected face, and then repairs the model; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

1. **Select faces: Remove/Undo/ALL -(choose one or more faces to delete.)**
2. **Select faces: Remove/Undo/ALL - (choose more faces, or else press Enter to continue.)**

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
The selected faces are deleted, and the program repairs the 3D solid to keep it solid.This option cannot remove inate faces, such as those on the cube illustrated above.

Copy

Copies faces; selected faces turn into regions; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

1. **Select faces: Remove/Undo/ALL-(choose one or more faces to copy.)**
2. **Select faces: Remove/Undo/ALL - (choose more faces, or else press Enter to continue.)**
3. **Specify <Base point>/Vector- (pick a point, enter x,y,z coordinates, or press V.)**
4. **Specify end point - (pick the destination point.)**

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Vector - specifies the displacement through a vector measured from the origin (0,0,0) to the cursor's pick point; prompts you: **Specify end point - (pick a point or enter x,y,z coordinates.)** To copy edges to lines, use the **Edge | Copy** option. See the Copy (6.9.50) command.

coLor	Changes the color of the selected face; prompts you: 1. **Select faces: Remove/Undo/ALL-(choose one or more faces to color.)** 2. **Select faces: Remove/Undo/ALL - choose more faces, or else press Enter to continue.)** 3. **Color to select [BYLayer/ BYBlock/ Red/ Yellow/ Green/ Cyan/ Blue/ Magenta/ White] <BYLAYER> - (specify a color.)**
Undo	Undoes the last editing operation.
eXit	Returns to the previous prompt.

Edge Editing Options

To access these options, enter the Edgeoption: Enter an edge editing option: Copy/coLor/Undo/<eXit>: **(Enter an option, or press Enter to return to the previous prompt)**

Copy	Copies the selected edge, and turns it into a line; the copied edges are turned into entities such as lines and splines. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Prompts you: 1. **Select edges: Remove/Undo/ALL - (enter an option:)** • Pick one or more edges • **Remove** - removes edges from the selection set • **Undo** - undoes the last selection operation • **ALL** - selects all edges in the drawing • **Enter** - continues the command to the next prompt: 2. **Select edges: Remove/Undo/ALL - (choose more edges, orelse press Enter to continue.)** 3. **Specify <Base point>/Vector - (pick a point, enter x,y,z coordinates, or else press V)** 4. **Specify end point - (pick the destination point, or enter x,y,z coordinates.) Vector** - specifies the displacement through a vector measured from the origin (0,0,0) to the cursor's pick point; prompts you: **Specify end point - (pick a point or enter x,y,z coordinates)**
coLor	Changes the color of the selected edge; prompts you: 1. **Select faces: Remove/Undo/ALL - (choose one or more edges to color.)** 2. **Select faces: Remove/Undo/ALL - (choose more edges, orelse press Enter to continue.)** 3. **Color to select [BYLayer/ BYBlock/ Red/ Yellow/ Green/ Cyan/ Blue/ Magenta/ White] <BYLAYER> - (specify a color.)**
Undo	Undoes the last editing operation.
eXit	Returns to the previous prompt.

Body Editing Options

To access these options, enter the **Body**option: Enter a body editing option Imprint/seParate/Shell/cLean/Check/Undo/<eXit>: **(Enter an option, or press Enter to return to the previous prompt)**


Imprint

Imprints 2D entities onto 3D solids; the 2D entity must lie on, or intersect, one of the 3D solid's faces. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Prompts you:

1. **Select 3d solid** - **(choose a 3D solid.)**
2. **Select an object to imprint -(choose a 2D entity lying on or intersecting at least one face.)**

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
3. **Delete the source object Yes/<No>** - **(enter an option:)**

- **Yes** - deletes the 2D entity
- **No** - does not delete the 2D entity, which can be reused

4. **Select an object to imprint** - **(choose another 2D entity, or press Enter to end.)** You can extrude the imprint. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i


seParate

Separates disjointed 3D solids; prompts you: **Select 3d solid** - **(choose a 3D solid.)** This option does not work on bodies created through Boolean operations (Union, Intersect, Subtract).


Shell

Turns the 3D solid into a hollow entity (shelled); prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

1. **Select 3d solid** - **(choose a 3D solid.)**
2. **Remove faces: Add/Undo/ALL** - **(press Enter)**; removing faces creates unexpected results.
3. **Enter the shell offset distance** - **(specify a distance):**

- **Positive distance** - shells body outwards
- **Negative distance** - shells body inwards

If the shell offset distance is larger than the 3D solid, this command fails.


cLean

Cleans the 3D solid; the cleaning operation checks for errors in the 3D solid. Prompts you: **Select 3d solid** - **(choose a 3D solid.)**

Check	Reports on the 3D solid; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Select 3d solid** - **(choose a 3D solid; atypical report looks like this:)** `checked: 1 lumps 2 shells 0 wires 12 faces 12 loops 48`
Undo	Undoes the last editing operation.
eXit	Returns to the previous prompt.

Related Commands

BimSplit (6.7.32) - splits segments of wall-like solids into separated solids. Box (6.6.17) - creates 3D boxes as solid models. Cone (6.9.63) - creates 3D cones as solid models. Cylinder (6.9.60) - creates 3D cylinders as solid models. Dish (6.10.30) - creates the lower half of 3D balls as solid models. Dome (6.10.34) - creates the upper half of 3D balls as solid models. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Imprint (6.17.12) - mprints 2D entities onto planar faces of 3D solids and surfaces. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Revolve (6.26.30) - revolves 2D entities into 3D solid models. Section (6.27.11) - creates 2D regions from 3D models. Slice (6.27.36) - cuts 3D solids into two parts. Sphere (6.27.64) - creates 3D balls as solid models. Subtract (6.27.56) - removes one solid model from a second. Torus (6.29.22) - creates 3D donuts as solid models. Union (6.30.6) - joins two or more solid models into a single entity. Wedge (6.33.7) - creates 3D wedges as solid models.

Figure 6.2120

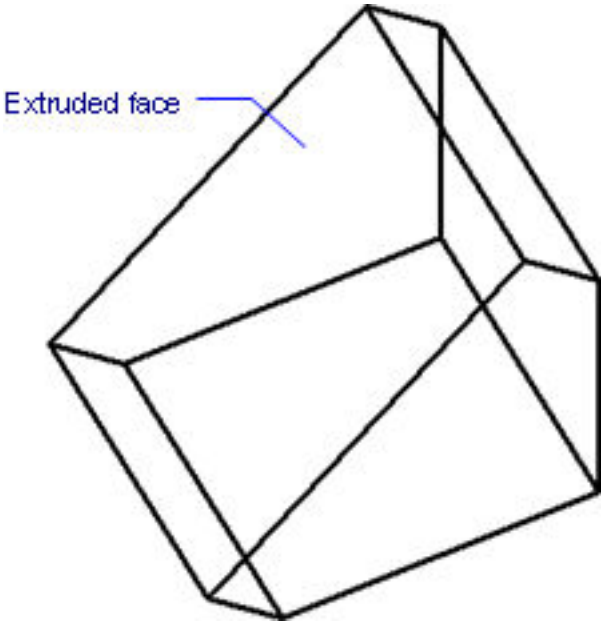


Figure 6.2121

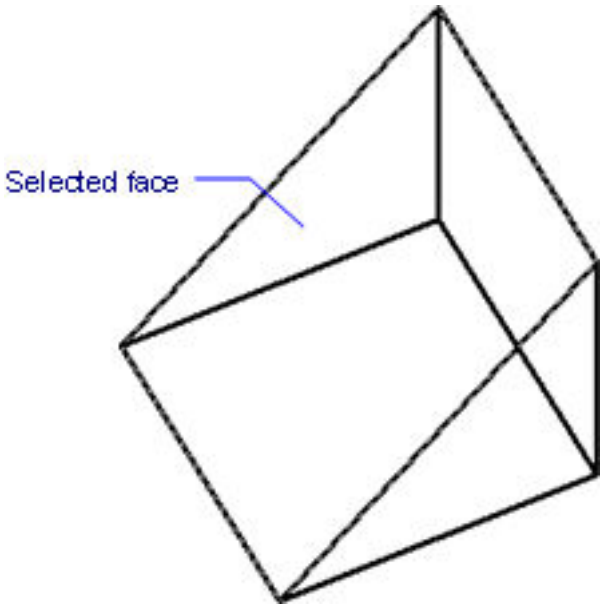


Figure 6.2122

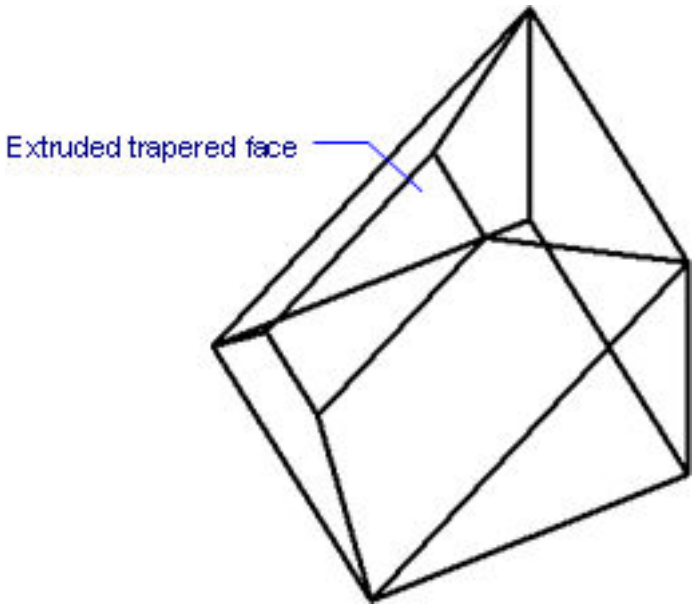


Figure 6.2123

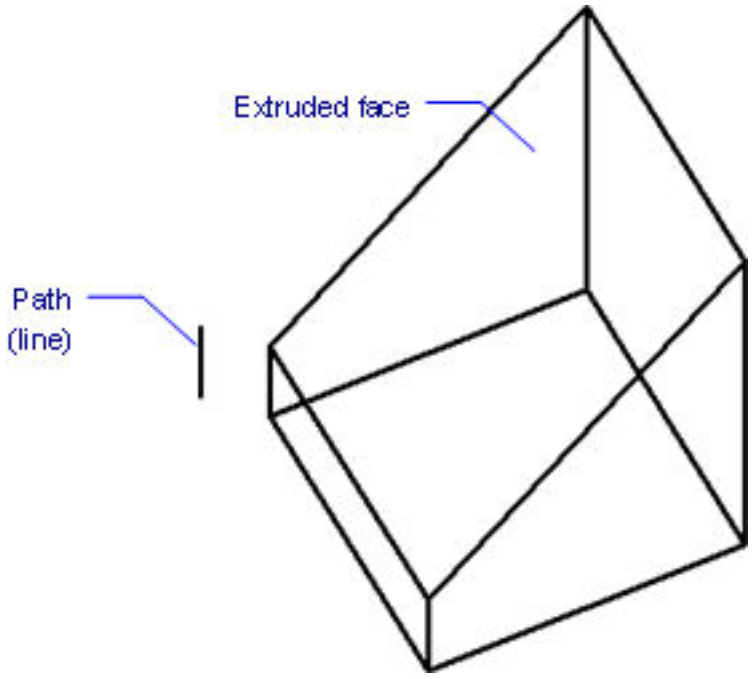


Figure 6.2124

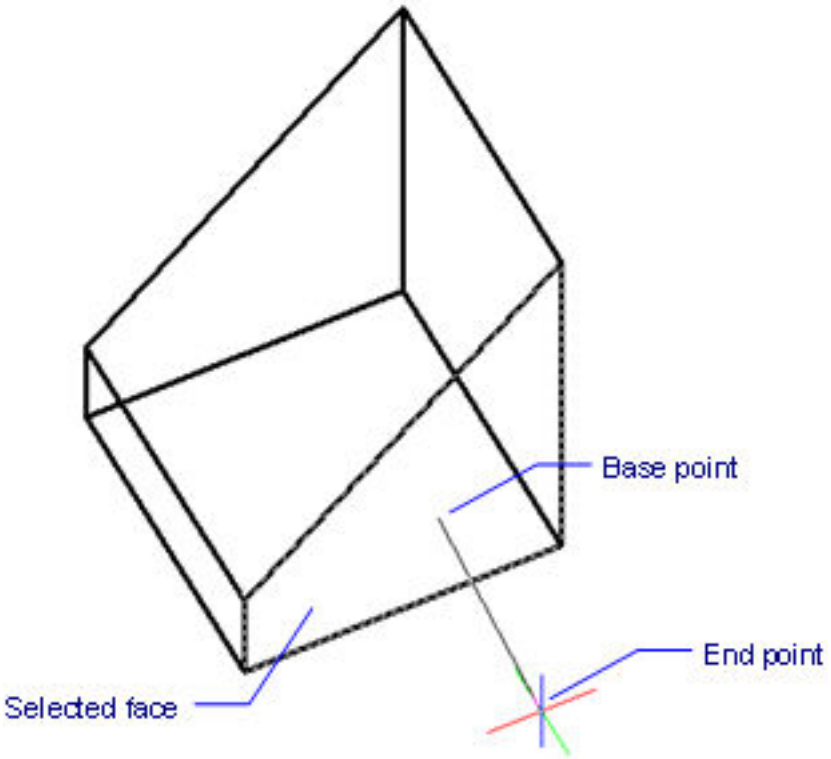


Figure 6.2125

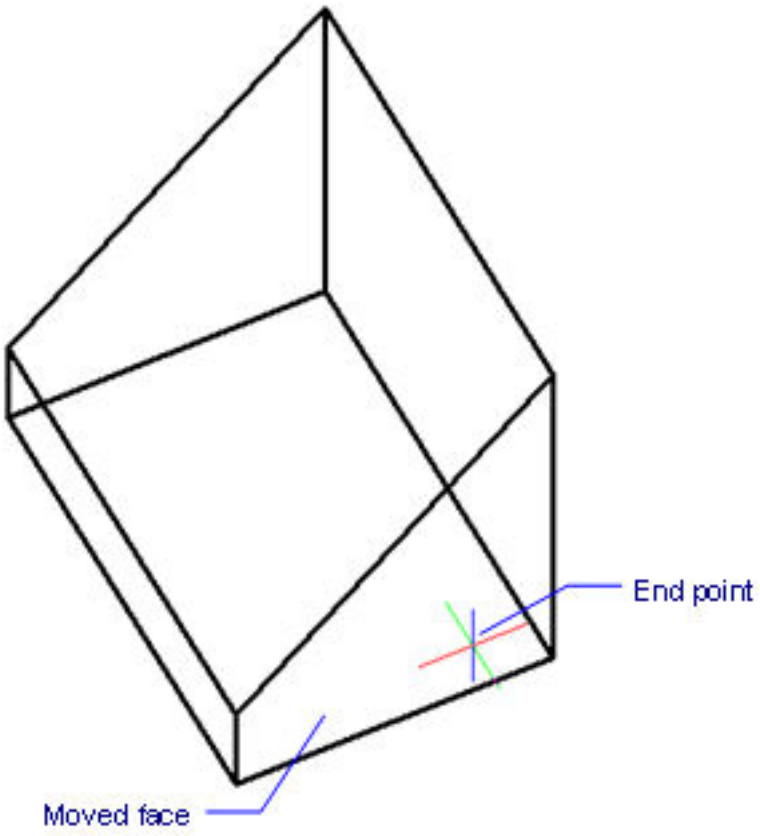


Figure 6.2126

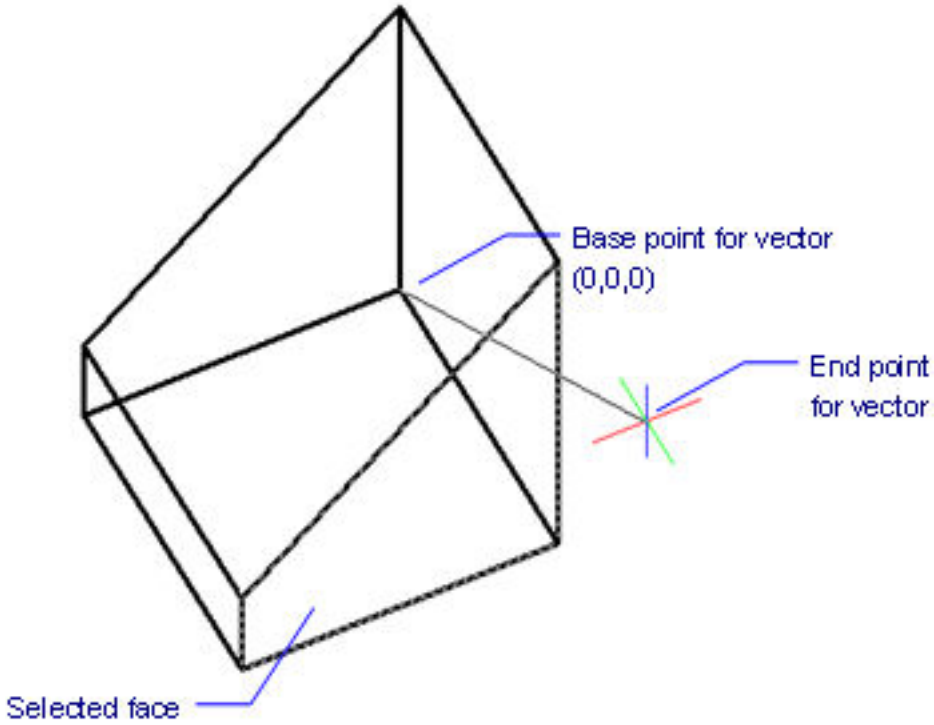


Figure 6.2127

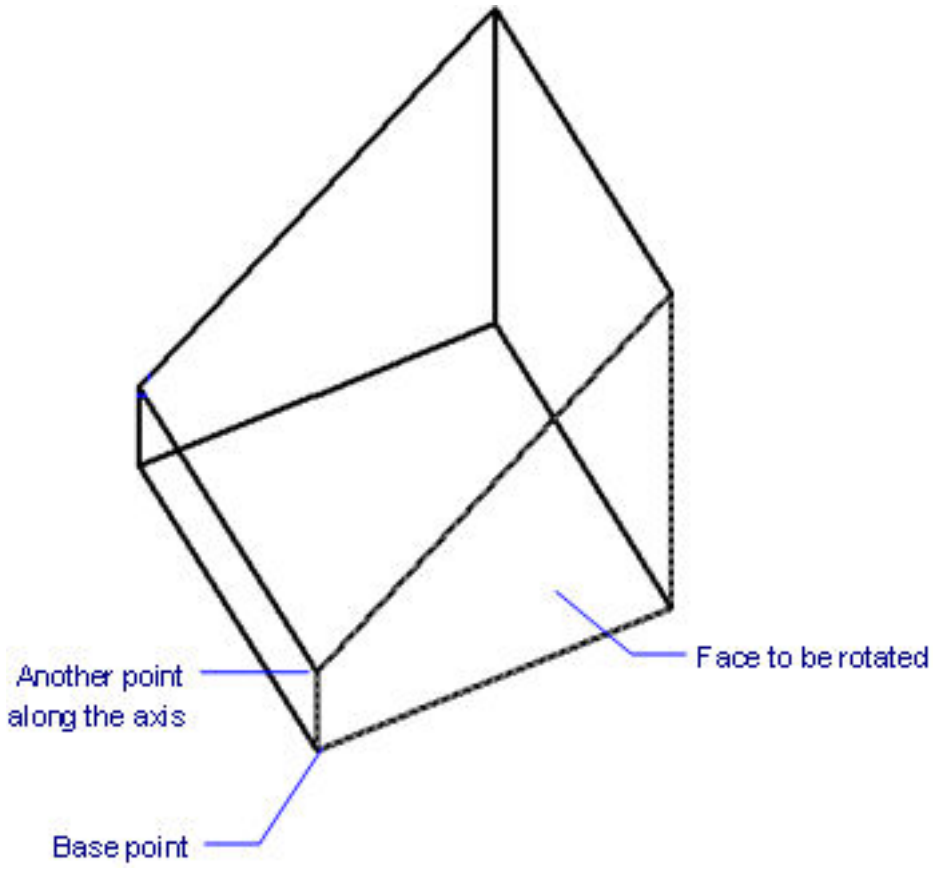


Figure 6.2128

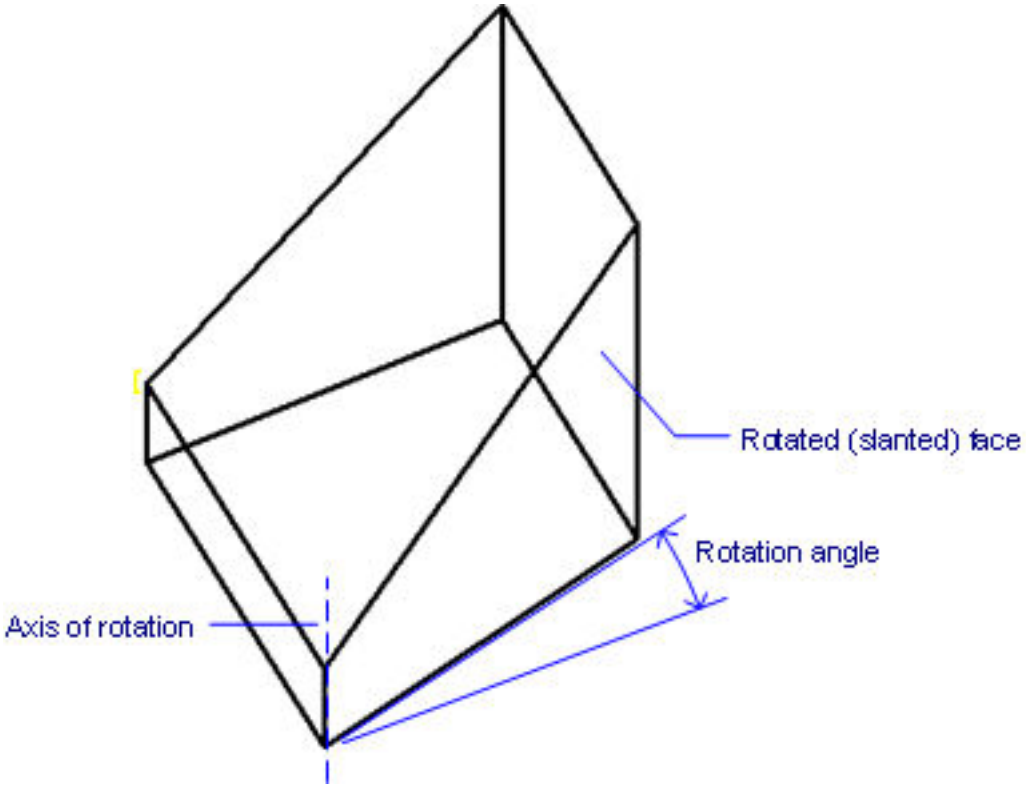


Figure 6.2129

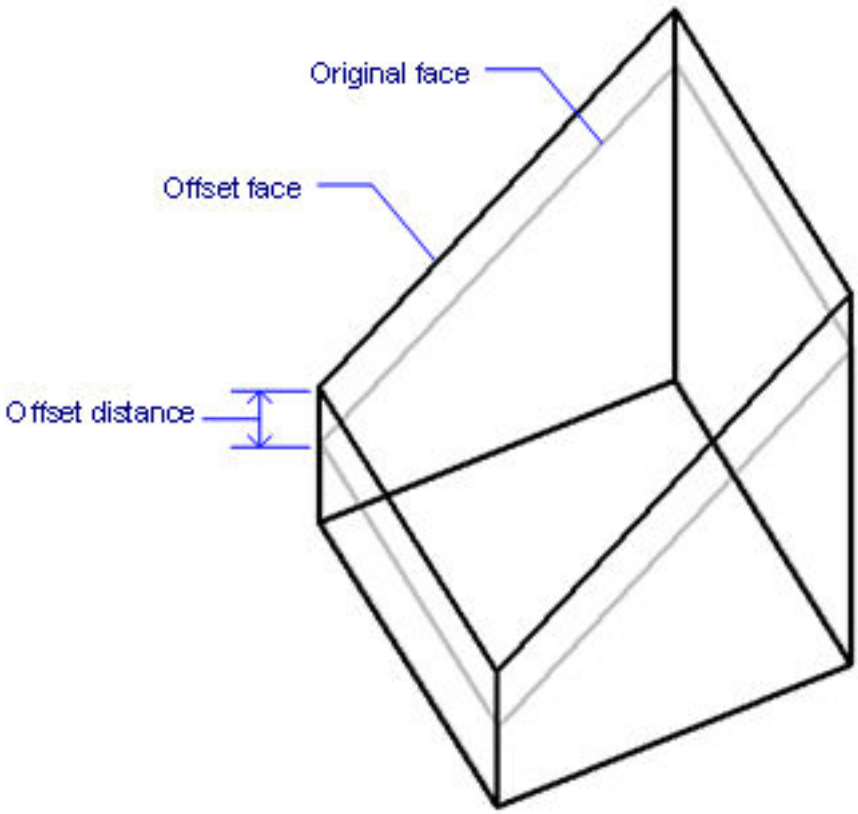


Figure 6.2130

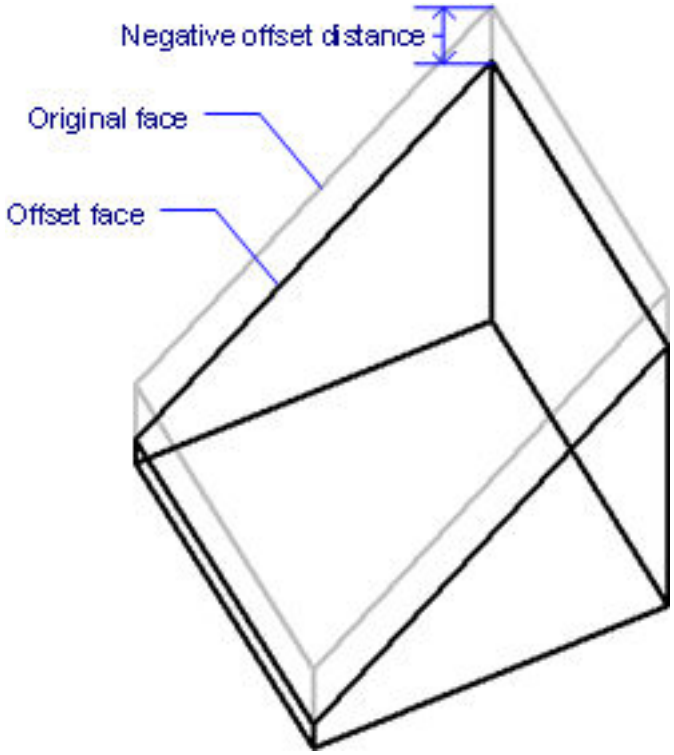


Figure 6.2131

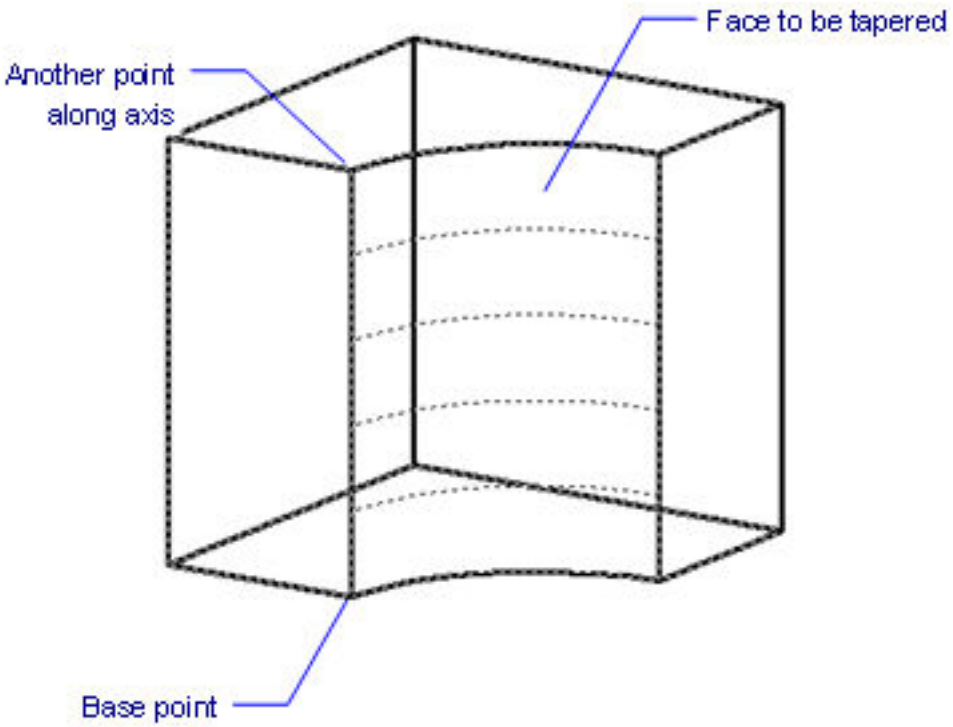


Figure 6.2132

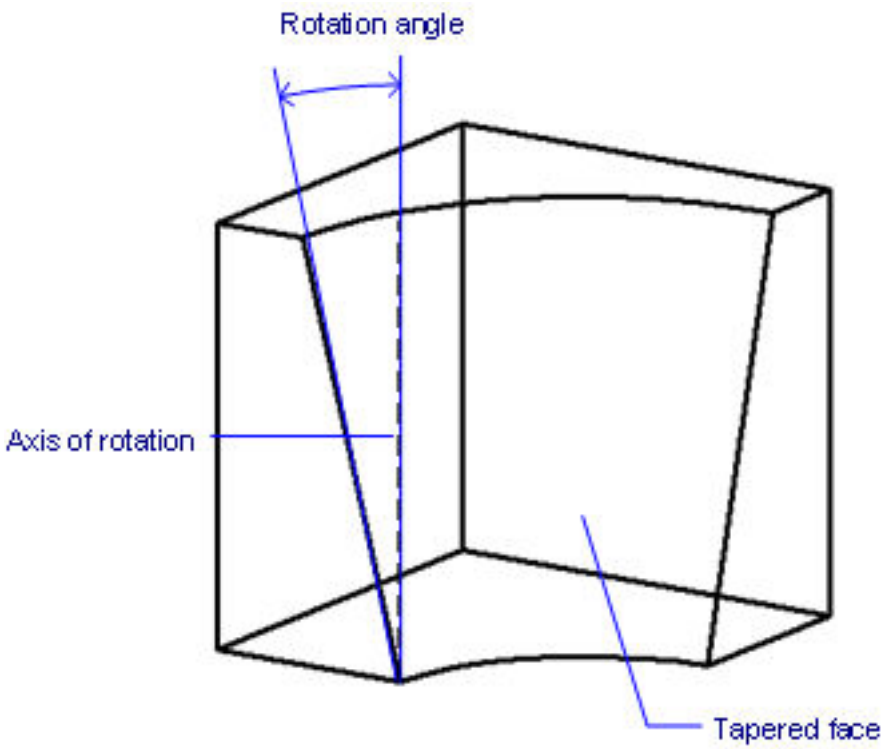


Figure 6.2133

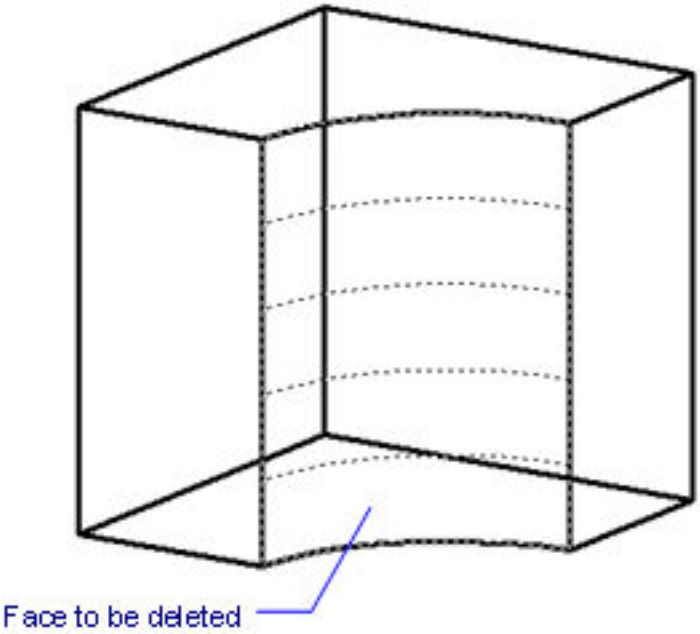


Figure 6.2134

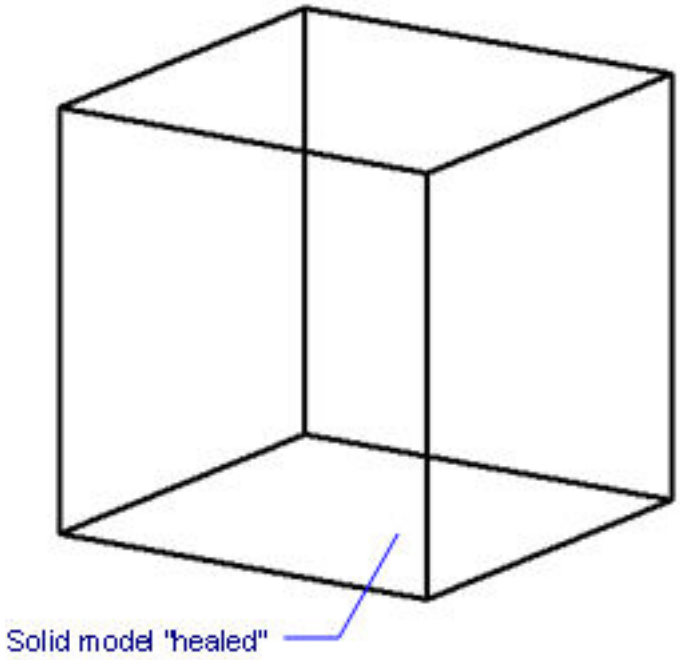


Figure 6.2135

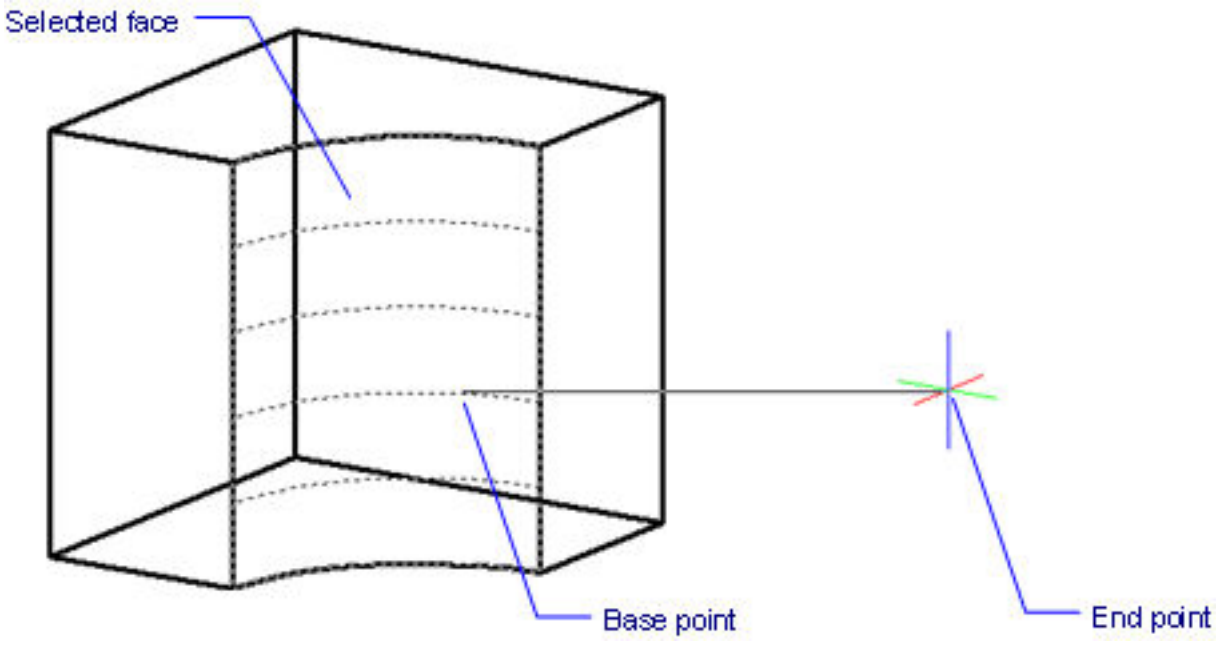


Figure 6.2136

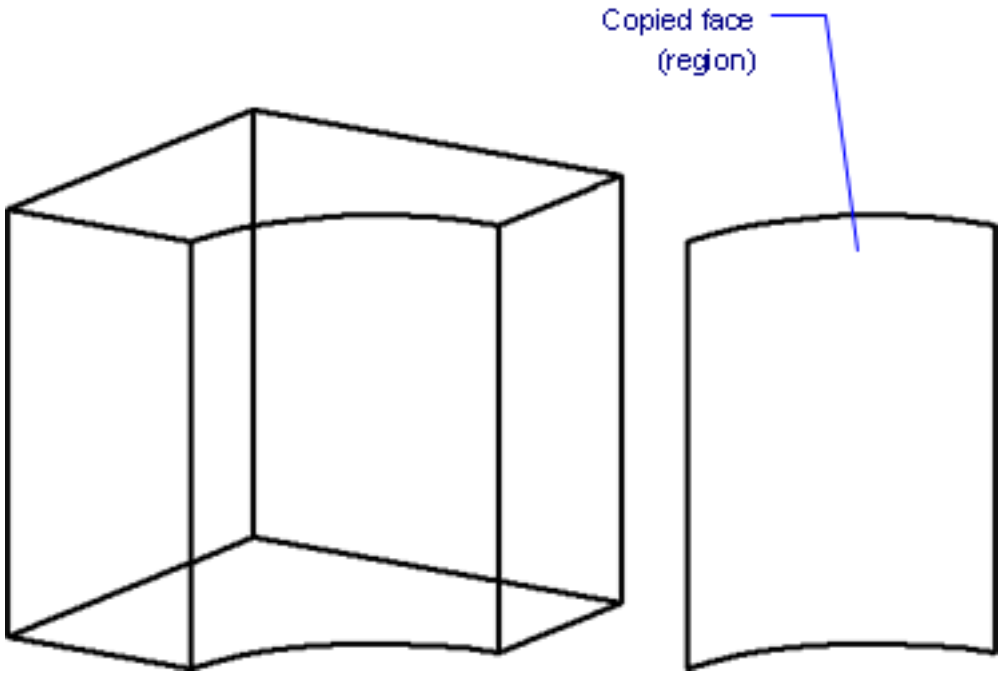


Figure 6.2137

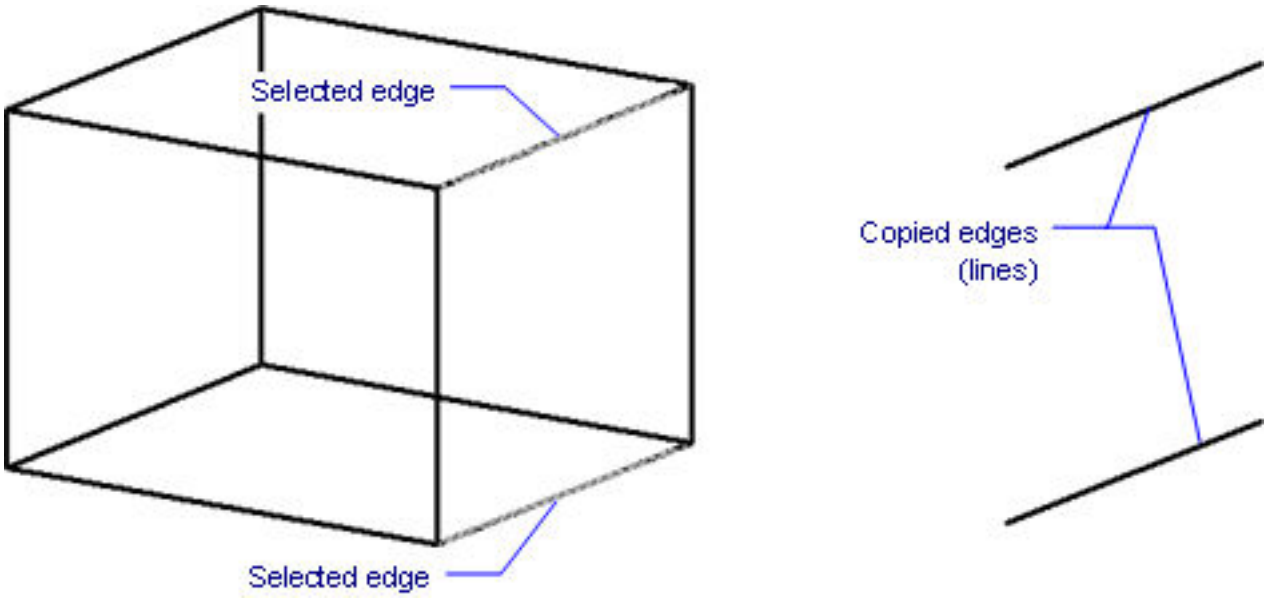


Figure 6.2138

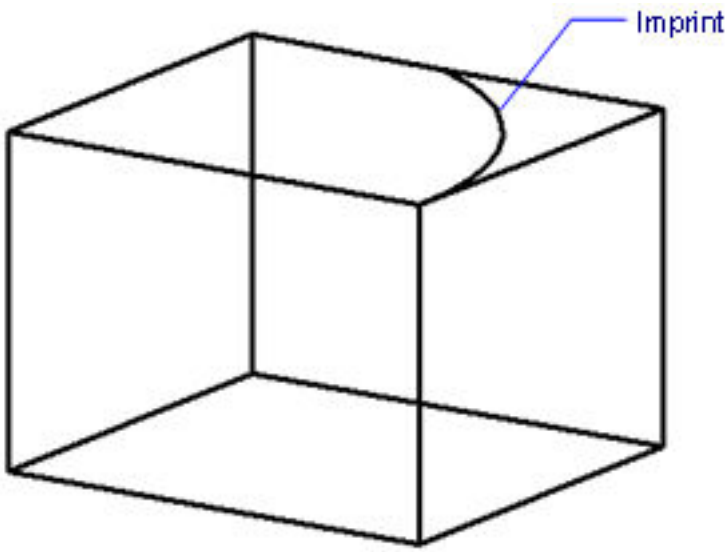


Figure 6.2139

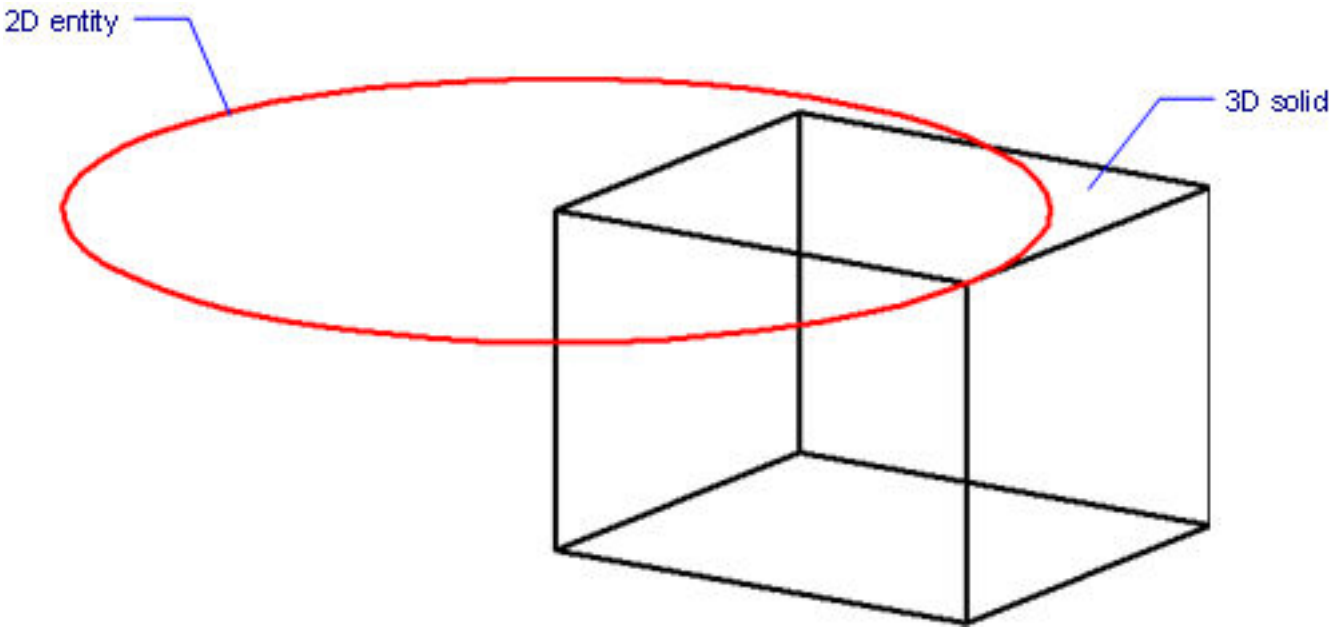


Figure 6.2140

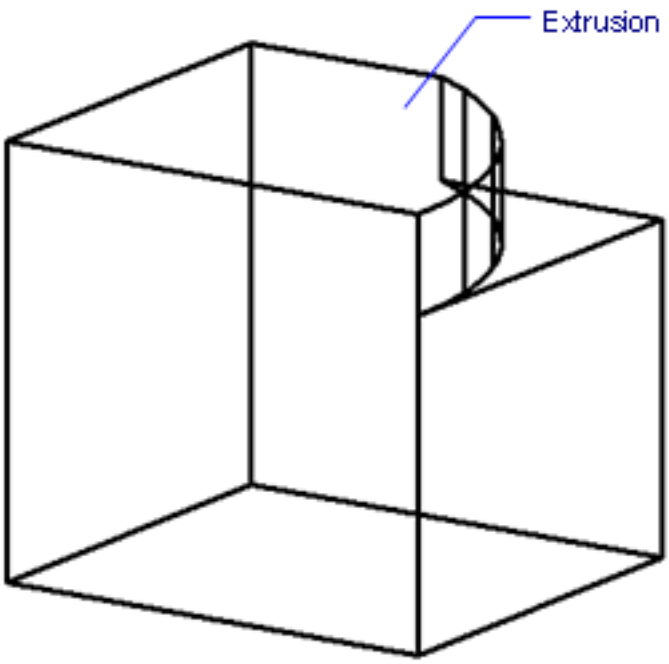


Figure 6.2141

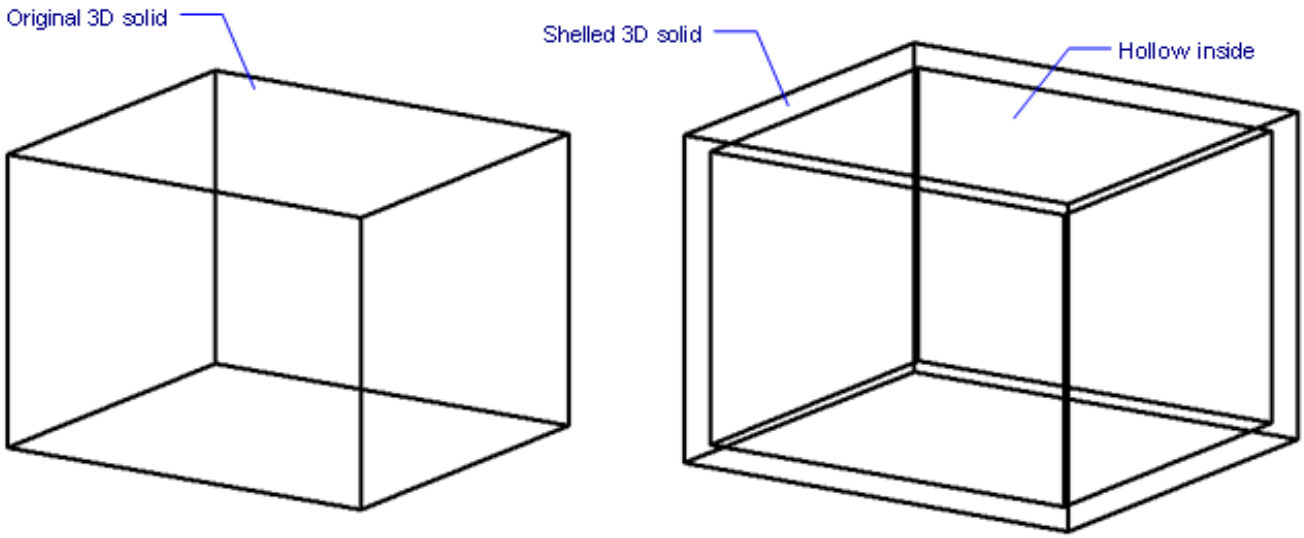


Figure 6.2142

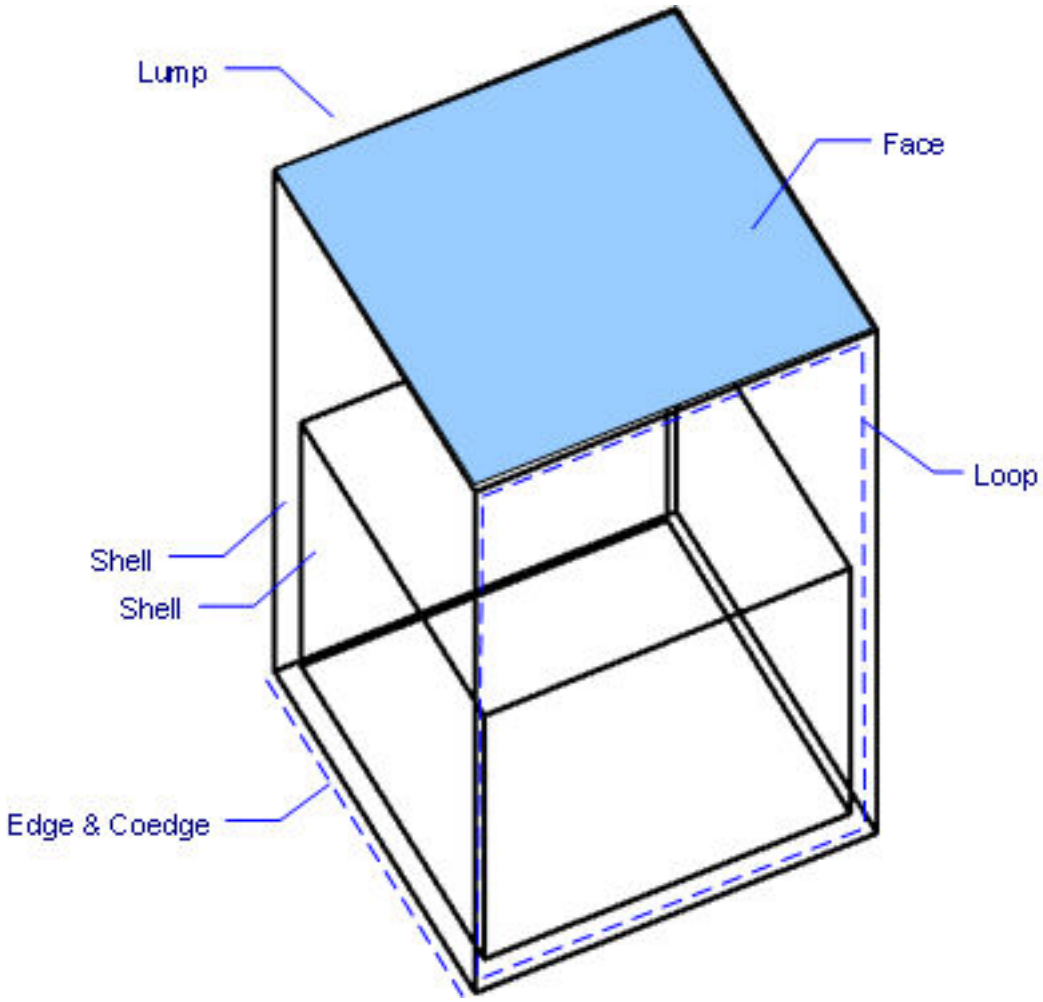


Figure 6.2143



6.27.40 SplinEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Modifies the properties of a spline; converses a spline to a polyline. **Note:** It is easier to edit the spline and its grips when Snap is turned off.

Accessing the Command

 command bar: **splinedit** transparent: **'splinedit** menu bar: **Menu| command** toolbar: **Modify |**  **: SplinEdit Prompts you in the command bar:** Select spline: **(Select a spline entity)** Edit spline [Close/Join/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>: **(Select an option)** Edit spline [Close/Join/Fit data/Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>: **(Select another option, or else press Enter to end the command)**

Command Options

Option	Description
Select spline to edit	Selects the spline to which the editing operations should apply: pick a single spline. The program shows the spline's fit points (green squares) and places a triangle at the start of the spline: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/spedit-eg.png})
Close **or** Open	Depending on whether the selected spline is open or closed, one of the options is available: • **Close** - closes an open spline connection the start and endpoint. • **Open** - opens a closed spline by removing the segment between the first and last points specified when the spline was originally created.
Join	Joins the spline with an open 2D entity that shares an endpoint with the spline (coincident endpoints); you can join lines, polylines, arcs, elliptical arcs, and other splines. Prompts you: 1. **Select any open curves to join to source - (select a connected entity.)** 2. **Entities in set: 1** 3. **Select any open curves to join to source - (select another connected entity, or else press Enter.)** The entities joined to the spline take on the properties of the spline, such as its color and linetype.

Fit Data

Edits fit point data using several options; fit points determine the curvature of the spline. Prompts you: **Edit fit data [Add/Close/Delete/Kink/Move/Purge/Tangents/toLerance/eXit] <eXit>** - **(choose an option:) Add** - adds fit point between two existing ones; prompts you:

1. **Select existing fit point <exit>** - **(select a fit point:)** the selected and the next fit point are highlighted in red: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/fitpoint-eg.png})
2. **Specify new fit point to add <exit>** - **(pick a point to add, as shown by the cursor, above.)**
3. **Specify new fit point to add <exit>** - **(pick another point to add, or else press Enter to return to the previous prompt.)**

Close/Open- closes an open spline by connecting the start and end points, or else opensa closed spline by removing the segment between the first and last points when the spline was originally created. **Delete** : Removes fit points; prompts you:

1. **Select fit point to delete <exit>** - **(select a fit point; the program removes the fit point, and adjusts the shape of the spline.)**
2. **Select fit point to delete <exit>** - **(select another point or press Enterto exit.)**

Kink - adds a knot and fit point at the specified location on the spline, which does not maintain tangent or curvature continuity at that point; prompts you:

1. **Specify a point on the spline <exit>** - **(pick a point on the spline to locate the new knot.)**
2. **Specify a point on the spline <exit>** - **(pick another point or press Enter.)**

Move - moves a fit point; prompts you: **Enter new location or [Next/Previous/Select point/eXit] <N>** - **(the program automatically selects the first fit point, the one marked by the triangle; enter an option:)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/spfitmove-eg.png})

- Pick a new point to relocated the selected fit point
- Choose an option to move to another fit point
- Press **Enter** to move the next fit point
- Press **X** to return to the previous prompt

Purge - replaces the fit data from the spline with control vertices: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/sppurge-eg.png}) **Tangents** - changes the starting and ending tangents of an open spline; prompts you:

1. **Specify start tangent or [System default] <System default>** - **(pick a point for the new starting tangency, or else press Enter to accept the system default.)** (\vref {fig:C:/Dev/Task-

Edit Vertex

Edits control frame date using several options; prompts you: **Edit vertex [Add/Delete/Elevate order/Move/Weight/eXit] <eXit>** - **(choose an option.) Add** - adds a new control vertex at the point you specify; prompts you:

1. **Specify a point on the spline <exit>** - **(pick a point on the spline to add the control vertex.)**
2. **Specify a point on the spline <exit>** - **(pick another point on the spline, or else press Enter to return to the previous prompt.)**

Delete - removes control vertex; prompts you:

1. **Select control point to delete <exit>** - **(select a vertex to delete.)**
2. **Select control point to delete <exit>** - **(pick another vertex to delete, or else press Enter to return to the previous prompt.)**

Elevate Order - increases the polynomial order (degree plus one) of the spline to increase the number of control vertices across the spline; the maximum value is 26. Prompts you: **Enter new order <xx>** - **(type an integer.) Move** - relocates control vertices one by one; prompts you:

1. **Enter new location or [Next/Previous/Select point/eXit] <N>** - **(pick a point, choose an option or press Enter to move the next fit point.)**
2. **Enter new location or [Next/Previous/Select point/eXit] <N>** - **(pick a point, choose an option or press Enter to move the next fit point or press X to stop moving vertices and return to the previous prompt.)**

Weight - changes the weight of a specified control vertex and then recalculates the spline based on the new weight value; larger weights pull the spline closer to the control vertex. In the figure below, the red control vertex has a weight of 1: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/sp weight-1-eg.png}) And here the red control vertex has a weight of 6: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/sp weight-6-eg.png}) Prompts you:

1. **Enter new weight (current = 1.00) or [Next/Previous/Select point/eXit] <N>:** - **(enter new weight or enter an option.)**
2. **Enter new weight (current = 1.00) or [Next/Previous/Select point/eXit] <N>** - **(press Enter to move to the next vertex, enter a value, or else press X to return to the previous prompt.)**

Exit - accepts the changes and returns to the previous prompt.

Convert to Polyline	Converts the spline to a polyline. The precision determines how closely the resulting polyline matches the spline: values between 0 and 99 are accepted, with higher values creating more polyline segments. Prompts you: **Specify a precision <10>** - **(enter an integer.)** The PLINECONVERTMODE (1.13.11)variable specifies the kind of polyline to convert to: • **0** - create polylines with line segments • **1** - create polylines with arc segments
Reverse	Reverses the direction of the spline; the triangle icon appears on what was formerly the spline's end point.
Undo	Undoes the previous modification.
Exit	Concludes the editing of the spline.

Related Commands

Join (6.18.1) - joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints. PEdit (6.24.20) - edits polylines, 3D polylines, and 3D meshes. Spline (6.27.65) - draws open and closed splines.

Figure 6.2144

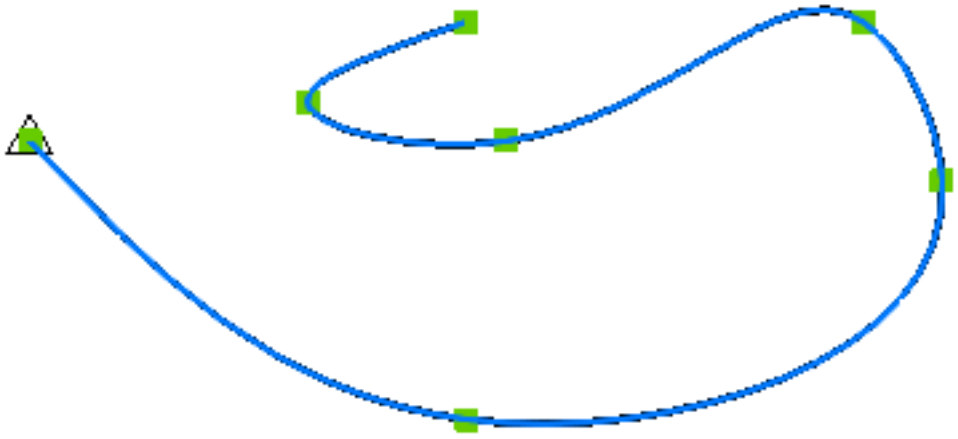


Figure 6.2145

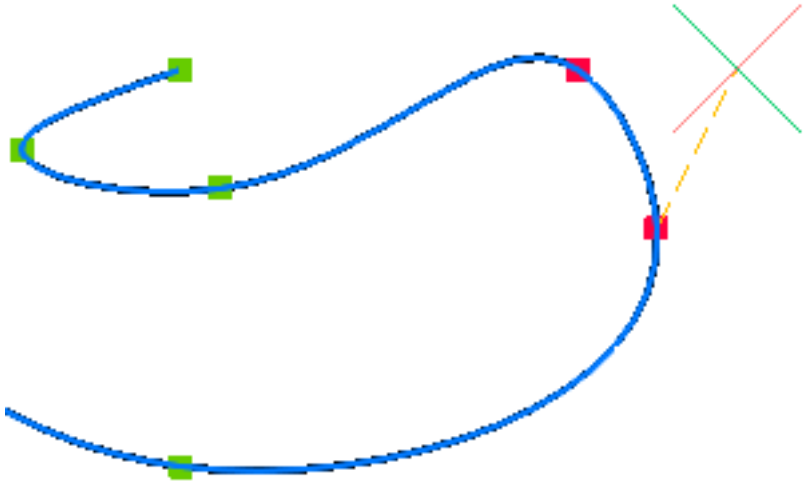


Figure 6.2146

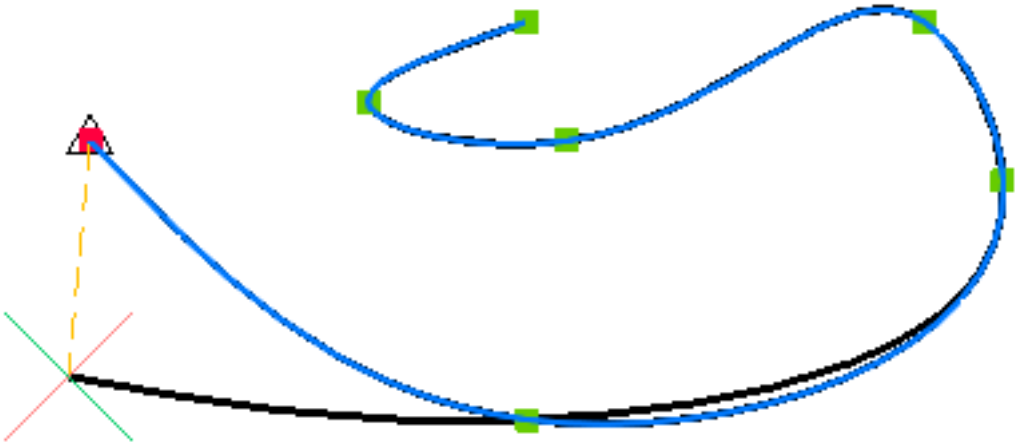


Figure 6.2147

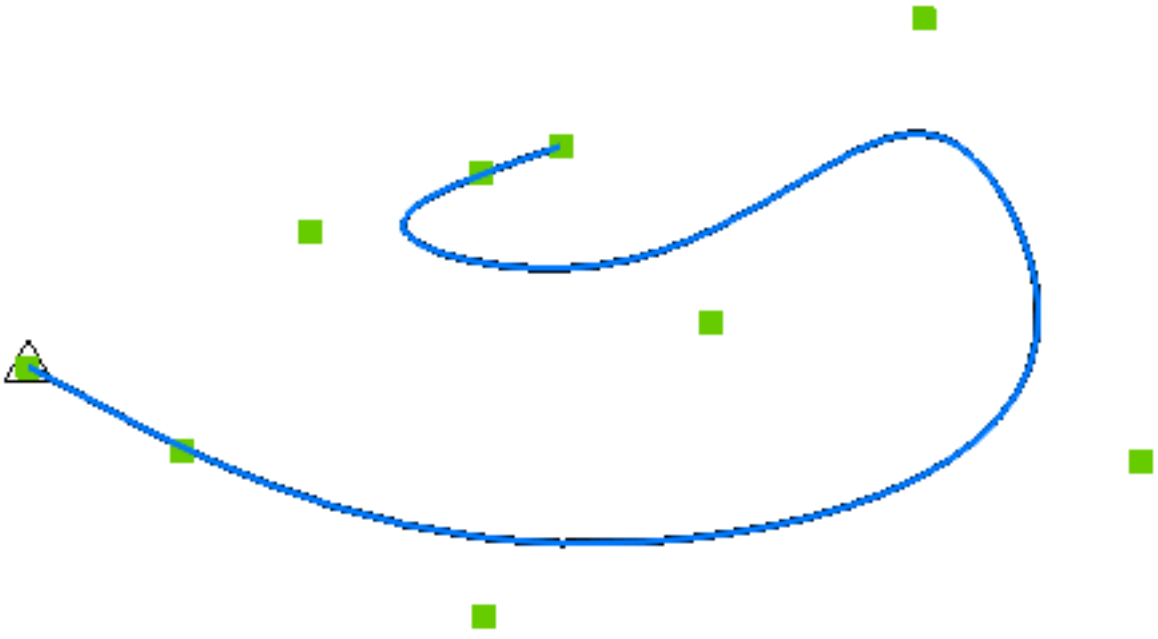


Figure 6.2148

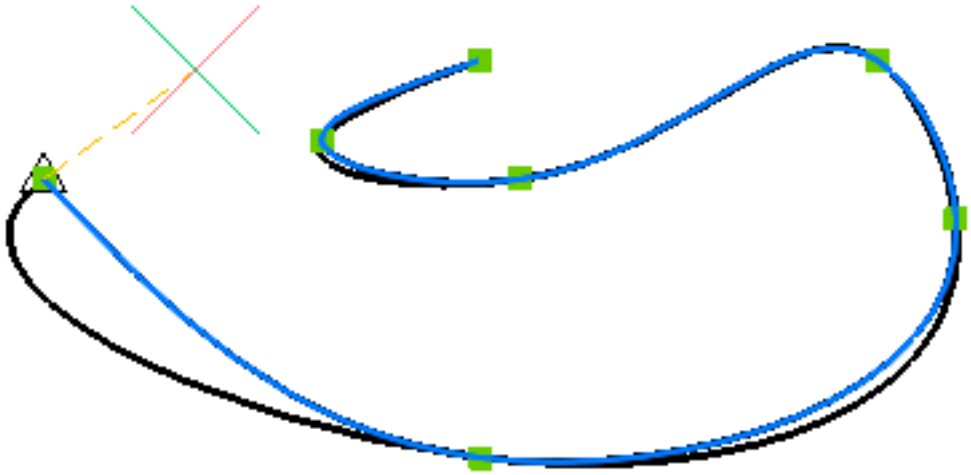


Figure 6.2149

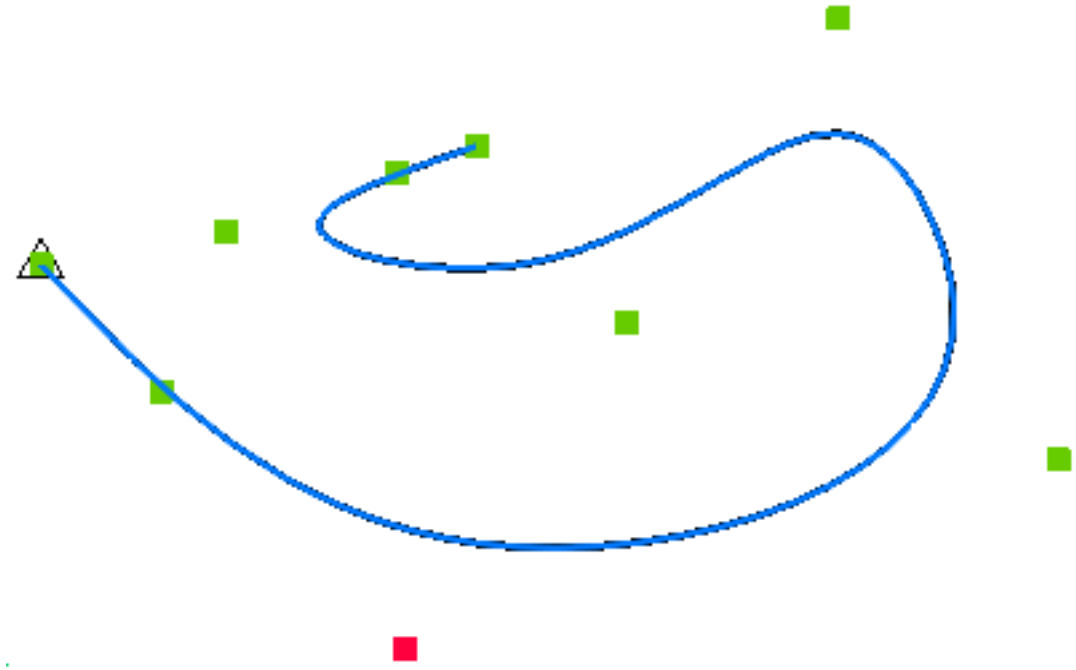


Figure 6.2150

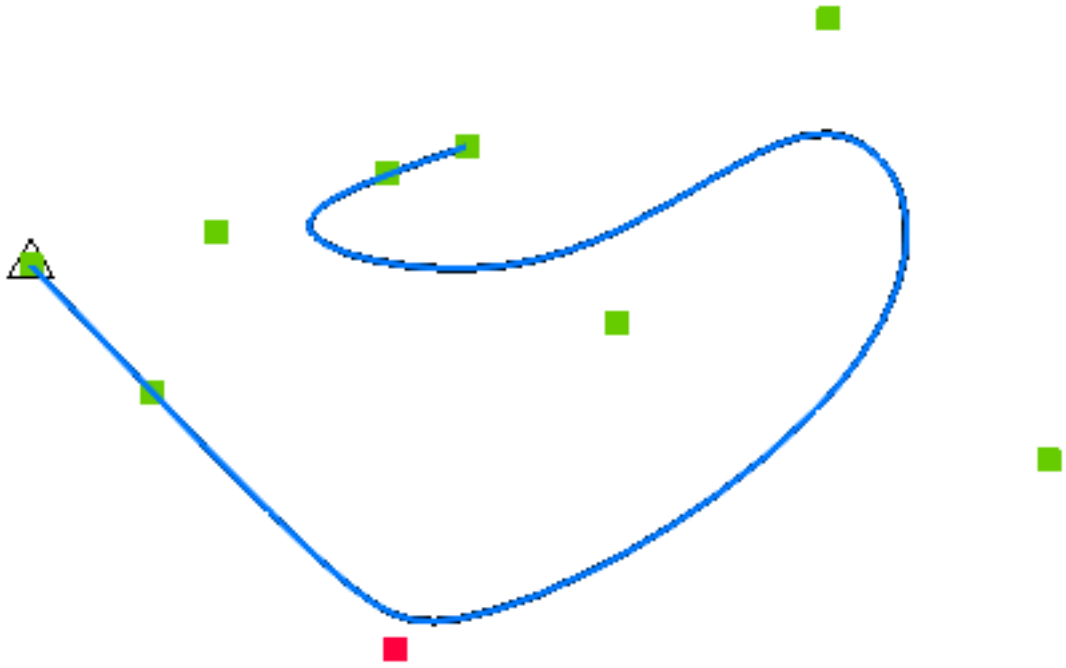


Figure 6.2151



6.27.41 Site

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Creates a topographical surface from an imported points file or placing new points.

Accessing the Command

 Keyboard: **site** Menu:**BIM | Create Site** Toolbar: **BIM Site Design** Quad Tab: **Model | Create Site** Ribbon Tab | Panel:**Data | Site | Create Site : Site Prompts you:** Select entities to create site or [Import from file/Place points]: **(Select entities or choose an option.)** By default, the create site tool is set to ”Select entities” option which allows you to create a toposurface from a selection set of points and polylines.

Command Options

Option	Description
Import from file	Allows creating a toposurface from an imported points file. Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/bimsite Choose a file and click Open.
Place points	Creates a toposurface by placing points. Prompts you: **Pick point:(Specify a point)**. **Enter elevation <0.000>:(Type a value or press Enter to accept the default)**. **Pick point:(Specify a point or press Enter to stop adding points)**. At least three points are needed to create a TIN surface (Triangulated Irregular Network).

Procedures

Creating a Toposurface from a selection set of points and polylines (1.18.8) Creating a Toposurface from an imported points file (1.18.8) Displaying the properties of a TIN surface (1.18.8) Adding linear entities as breaklines to the TIN surface (1.18.8)

Related Commands

SiteEdit (1.18.10) - allows to add/remove boundaries, breaklines, and points to a surface. Grading (6.15.24) - creates a graded surface on existing toposurfaces.

Figure 6.2152

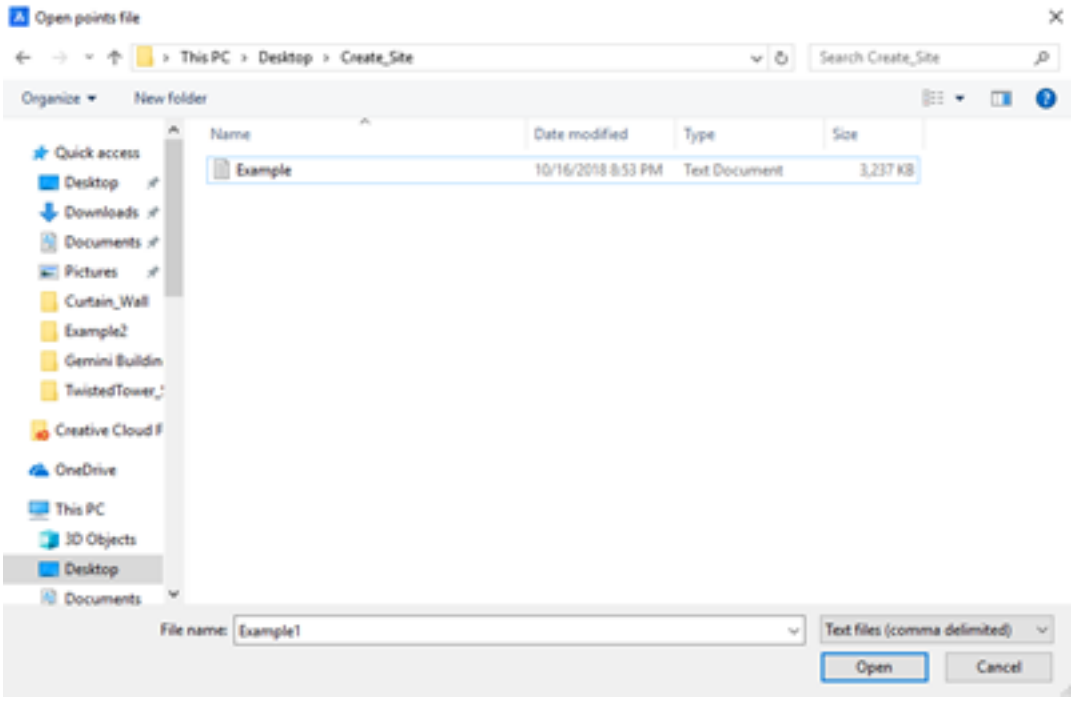
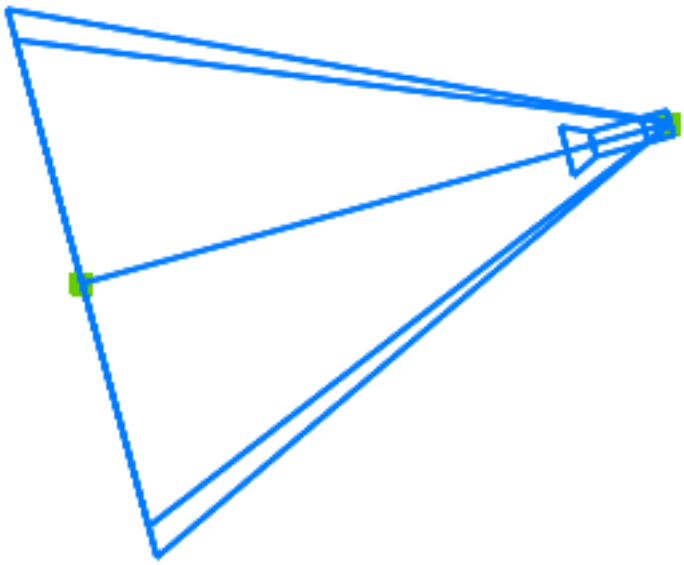


Figure 6.2153



Figure 6.2154



6.27.42 Spotlight

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates a spot light, which casts a narrow beam towards a target point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help glyph.png})

Accessing the Command


command bar: **spotlight** menu bar: **View | Rendering | Lights | New spot light** ribbon: **View | Lights | Create | New spot light** toolbar: **Lights |**  **: Spotlight Prompts you in the command bar:** Specify source position <0,0,0>: **(Pick a point in the drawing to position of the light, or enter an x,y,z coordinate)** Specify target position <0,0,-10>: **(Specify the target point for the light)**

- When LIGHTINGUNITS variable is 0 (no lighting units specified), **the following prompt is displayed next:**

Enter an option to change [Name/ Intensity/ Status/ Hotspot/ Falloff/ shadoW/ Attenuation/ Color/eXit] <eXit>: **(Enter an option, or press Enter to exit the command)**

- When LIGHTINGUNITS system variable is 1 (American lighting units) or 2 (international lighting units), **the following prompt is displayed next**:

Enter an option to change [Name/Intensity factor/Status/ Photometry/ Hotspot/ Falloff/ shadoW/ Attenuation/filterColor/eXit] <eXit>:**(Enter an option, or press Enter to exit the command)**

Command Options

Option	Description
Specify source position	Specifies the location of the light in 3D space: • Pick a point in the drawing to position of the light • Enter x,y,z coordinates • Press **Enter** to accept the default location

Specify target position	Specifies the location of the target, the point at which the spotlight is focussed: • Pick a point in the drawing to position of the target • Enter x,y,z coordinates • Press **Enter** to accept the default location
Name	Specifies a name for the spotlight; prompts you: **Enter light name <Spotlight1>** - **(enter a name or press Enter to accept the default name.)**
Intensity **or** Intensity factor	Specifies the intensity of the light; prompts you: **Enter intensity (0.00 - max float) <1.0>** - **(enter an option:)** • Enter an intensity values between 0.00 and the maximum value supported by your system (max float = maximum value of a floating point number) • Press **Enter** to accept the default value When LightingUnits variable is 0, this option is named **Intensity**; when 1 or 2, **Intensity factor**.
Status	Toggles the use of this light; prompts you: **Enter status [ON/OFf] <ON>** - **(enter an option:)** • **ON** - light is included when calculating renderings • **OFf** - light is excluded
Hotspot	**Enter hotspot angle (0.00-160.00) <45.0>:(Enter a value between 0.00 degrees and 160.00 degrees). Enter falloff angle (0.00-160.00) <50.0>: (Enter a value between 0.00 degrees and 160.00 degrees).** If yout enter a Falloff angle that is smaller than the Hotspot angle, the latter is set equal to the Falloff angle. When a surface is illuminated by a spotlight, there is an area of maximum illumination (hotspot) that is surrounded by an area of lesser intensity (falloff). • Hotspot angle: angle of the central light cone (defines the hotspot) • Falloff angle: angle of the full light cone The difference between the hotspot angle and the falloff angle defines the area of lesser light intensity. If the falloff angle and the hotspot angle are almost equal the edge of light cone is rather sharp. The greater the difference between both angles, the softer the edge of the light cone.

Photometry	(This option is available only when LightingUnits is 1 or 2.) Specifies the photometric properties of the light; prompts you: **Enter a photometric option to change [Intensity/Color/eXit] <Intensity>** - **(enter an option:)** • **Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0>**: - **(enter the intensity expressed in candela or an option:) Enter Flux (Lm)**- **(enter the luminous flux expressed in lumens.) Enter Illuminance (Lx or Fc) or enter an option [Distance] <9424.8>**- **(enter the illuminance expressed in lux or foot-candles.) Enter Distance** - **(Enter a distance expressed in drawing units.)** • **Enter color name or enter an option [?/Kelvin] <D65>**- **(enter a color name .) ?** - lists the names of available colors, such as: D65 Fluorescent Coolwhite Whitefluorescent daylightfluorescenT Incandescent Xenon Halogen Quartz Metalhalide mErcury Phosphormercury highpressureSodium Lowpressuresodium **Enter Kelvin temperature <3600.0>** - **(enter a color temperature expressed in degrees Kelvin.)**
Hotspot	Specifies the angle of the hotspot, the area of light concentration; the angle determines the size of the hotspot, irregardless the distance of target. Prompts you: **Enter hotspot angle (0.00-160.00) <45>** - **(enter an angle, or press Enter to accept the default)**
Falloff	Specifies the the angle of weaker illuminations surrounding the hotspot; out side of the falloff, there is no light. Prompts you: **Enter falloff angle (0.00-160.00) <50>**- **(enter an angle, or press Enter to accept the default.)**

shadoW

Specifies the look of shadows cast by this light; prompts you: **Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled] <Sharp>** - **(enter an option:)**

- **Off** - disables the calculation of shadows for the light.
- **Sharp** - displays shadows with sharp edges. Use this option to increase performance.
- **Soft mapped** - displays realistic shadows with soft edges. Prompts you: **Enter map size [64/128/256/512/1024/2048/4096] <256>:**
- **Soft sampled** : Displays realistic shadows with softer shadows based on extended light sources; prompts you: **Enter an option to change Shape/sAmples/Visible/<eXit>** - **(enter an option:) Enter shape [Disk/Rect] <Disk>** - **(type D or R) Enter radius <0.0>** - **(enter the radius of the disk) Enter Width <0.0>** - **(enter width of the rectangle) Enter Length <0.0>** - **(enter length of the rectangle)**
- **Enter Shadow Sample <16>** - **(specify the sampling size of the shadow)**; larger numbers are more accurate but take longer to render.
- **Enter Shape Visibility [Yes/No] <No>** - **(type Y or N to cast a shadow that represents shapes (more accurate) or is rectangular (faster to render).)**

Attenuation	Specifies how the illumination falls off with distance from the light: **Enter an option to change [attenuation Type/Use limits/attenuation start Limit/attenuation End limit/eXit] <eXit>** - **(enter an option:)** • **Enter attenuation type [None/Inverse linear/inverse Squared] <None>** - specifies the type of attenuation: **None** - No attenuation so that the distance to the light source has no influence **Inverse linear** - attenuation is the inverse of the linear distance from the light: at a distance of 2 units from the light source, light is half as strong; at a distance of 4 units, light is one quarter as strong **Inverse squared** - attenuation is the inverse of the square of the distance from the light: at a distance of 2 units, light is one quarter as strong; at a distance of 4 units, light is one sixteenth as strong • **Use limits** - toggles whether the extent of the illumination is limited; prompts you: **Limits [OFf/ON]** - **(enter an option:)** **ON**- limit the extent **OFf** - do not limit it • **attenuation start Limit** - defines the point where the light starts to shine, measured from the center of the light; prompts you: **Specify start limit offset <1.0>** - **(enter a distance.)** • **attenuation End limit** - defines the point where the light stops to shine, measured from the center of the light; prompts you: **Specify end limit offset <10.0>** - **(enter a distance.)**
Color **or** filterColor	Specifies the color of the light; prompts you: **Enter true color (R,G,B) or enter an option [Index color/Hsl]:<255,255,255>** - **(enter an option:)** • **Enter true color** - the RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The name of the model comes from the initials of the three additive primary colors, red, green, and blue. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values). • **Index color** - specifies an AutoCAD Color Index color. • **Hsl** - specifies the color using three parameters Hue, Saturation and Lightness When LightingUnits variable is 0, this option is named **Color**; when 1 or 2, **filterColor**.
Exit	Concludes the command and creates the light.

Related Commands

Distantlight (6.10.32) - creates a distant light. Light (6.20.28) - creates a new light in the drawing. Lightlist (6.20.29) - displays the Drawing Explorer - Lights dialog window. Pointlight (6.24.42) - creates a point light. Render (6.26.22) - calculates a photorealistic image of the 3D model. Sunproperties (6.27.57) - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog. Weblight (6.33.6) - creates a web light.

Figure 6.2155



6.27.43 SiteEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-bim.png}) Allows to add/remove boundaries, breaklines, and points to a surface.

Accessing the Command

BIM Toolbar:  **Quad:**  Keyboard: **siteedit** Menu:**BIM | Add to Site, Remove from Site** Toolbar: **Add to Site, Remove from Site** Quad Tab: **Model | Add boundaries, Add breaklines, Add points, Remove boundaries, Remove breaklines, and Remove points** Ribbon Tab | Panel:**Data | Site Design | Add to Site, Remove from Site : SiteEdit Prompts you:** Select a site: **(Select the toposurface.)** Pick points to add to the site or [Add Points/Add Breaklines/Add bOundaries/Remove Points/Remove Breaklines/Remove bOundaries]: **(Pick points manually or choose an option.)**

Command Options

Option	Description
Add Points	Sets points to a toposurface. Prompts you: **Select entities:(Select the entities and press Enter.)**
Add Breaklines	Sets breaklines to a toposurface. Prompts you: **Select entities:(Select the entities and press Enter.)**
Add bOundaries	Sets boundaries to a toposurface. Prompts you: **Select entities for boundary of type: [Outer/Show/Hide] <Outer>: (Choose an option)** select the bounding entity, and press Enter. **Outer:**Creates an outer surface boundary. **Show:** Shows the surface boundary. **Hide:** Hides the surface boundary.
Remove Points	Deletes points from a surface.
Remove Breaklines	Deletes breaklines from a surface.
Remove bOundaries	Deletes boundaries from a surface.

Procedures

Adding points to a Toposurface (1.18.10) Adding breaklines to a Toposurface (1.18.10) Adding boundaries to a Toposurface (1.18.10)

Related Commands

Site (6.27.41) -Creates a topographical surface from an imported points file, or placing new points. Grading (6.15.24) - creates a graded surface on existing toposurfaces.

6.27.44 Start

Run another application.

Accessing the Command

command bar: **start** transparent: **'start** : **Start Prompts you in the command bar:** Application to start: **(Enter the name of a program, such as 'otepad).** For a list of operating system commands, enter Help. To exit the window, enter Exit.

Command Options

Option	Description
Application to start	Specifies the name of the Windows program to run. Or, press **Enter** to open the Windows command prompt window. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/V

Related Commands

Shell (6.27.32) - opens the operating system command prompt window; runs other applications. Sh (6.27.28) - opens the operating system command prompt window; runs other applications. Files (6.14.5) - displays the operating system's File Manager.

Figure 6.2156

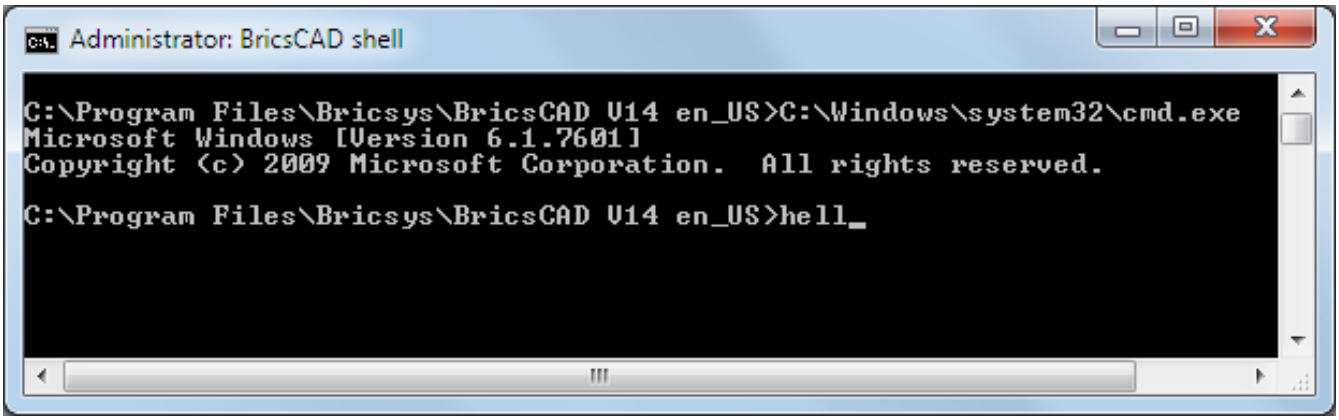


Figure 6.2157



6.27.45 StatBar

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Toggles the display of the status bar (short for "status bar").

Accessing the Command

command bar: **statbar** transparent: **'statbar** shortcut: **Shift + F3** menu bar: **View | Status Bar** : **StatBar Prompts you in the command bar:** WNDLSTAT is currently on [ON/OFf/Toggle] <On>: **(Enter an option)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/StatusBar.png}) The status bar provides quick access to information about the drawing and controls for toggling tools:

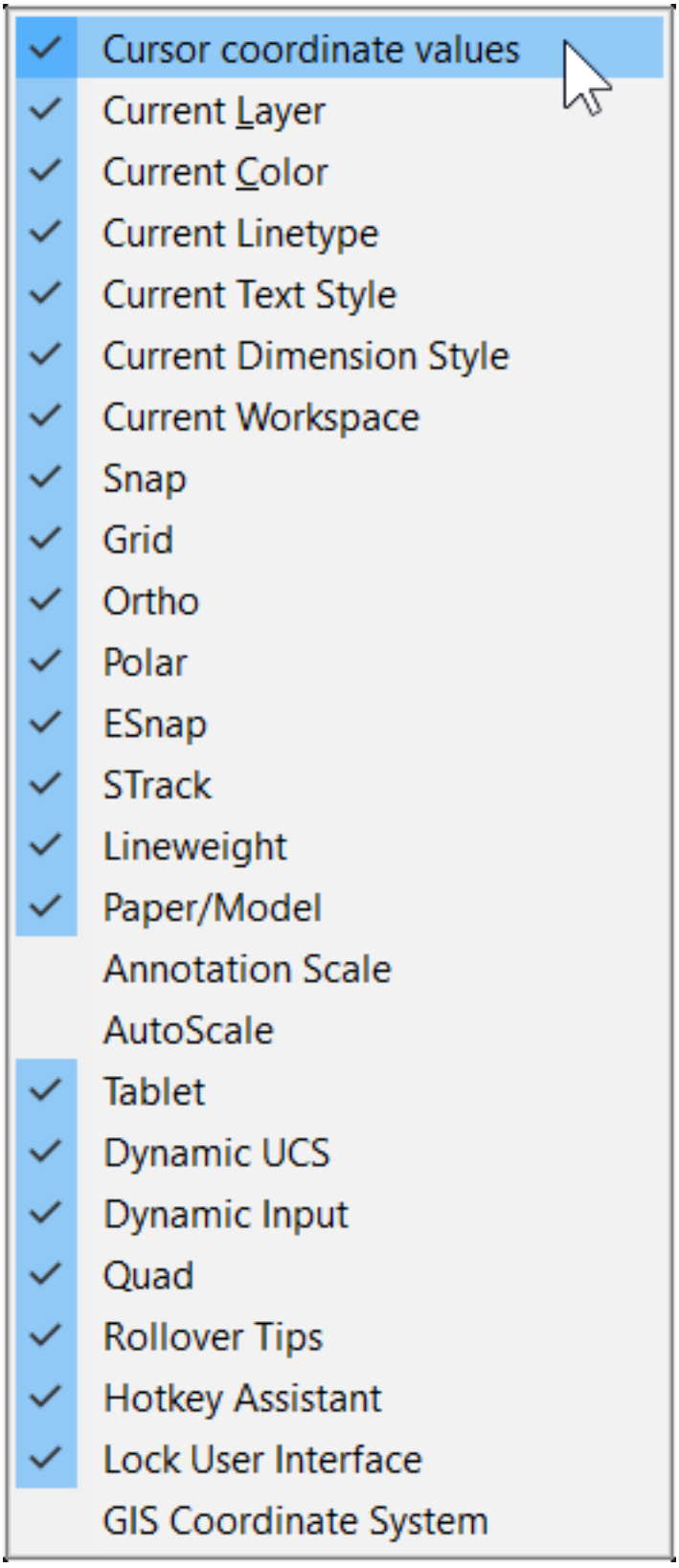
1. Status field displays a tooltip when hovering over a tool button, or displays command prompts when the command bar is closed; double click the status field to toggle the command bar on/off
2. Coordinates field
3. Drawing properties
4. Drawing aid fields
5. Status bar option button: click an option to toggle the corresponding field on and off:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/statusbar-menu.png}) The only reason to turn off the status bar is to gain a bit more drawing area. It is far more useful to leave it on.

Figure 6.2158



Figure 6.2159



Command Options

Option	Description
WNDLSTAT is currently on	Enter an option: • **ON** - turns on the status bar, when it is off • **OFf** - turns on the status bar, when it is on • **Toggle** - switches it from on to off to on WNDLSTAT is the name of the variable that stores the current status of the status bar.

Procedures

- Working with the status bar (1.12.2)

Related Commands

CleanScreenOff (6.9.21) - displays user interface elements that were hidden by theCleanScreenOn (6.9.22) command. CleanScreenOn (6.9.22) - hides elements of the user interface. CommandLine (6.9.34) - shows the command line. CommandLineHide (6.9.36) - hides the command line. GraphScr (6.15.20) - brings the graphics window to the foreground. Scrollbar (6.27.10) - toggles the display of the scroll bars. TextScr (6.29.13) - brings the text window to the foreground. Toolbar (6.29.18) - toggle the display of toolbars.

Figure 6.2160



6.27.46 Status

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Reports the status of the drawing.

Accessing the Command

 command bar: **status** transparent: **'status** menu bar: **Tools | Drawing Status** ribbon:**Tools | Inquiry | Drawing Status** toolbar: **Inquiry** |  : **Status** Displays no prompts, and displays the status report in the command bar.To view the full report, press **F2 (6.29.13)** to open the Prompt History window.A typical report looks like this: Status: Current drawing name: lsedit.dwg Drawing limits are: X= 0.0000 Y= 0.0000 X= 12.0000 Y= 9.0000 Paper space limits are: X= 0.0000 Y= 0.0000 X= 12.0000 Y= 9.0000 Screen width (pixels): 909 Screen height (pixels): 497 Insertion base is: X= 0.0000 Y= 0.0000 Z= 0.0000 Snap resolution is: X= 0.5000 Y= 0.5000 Grid spacing is: X= 0.5000 Y= 0.5000 Current layer: 0 Current color: BYLAYER Current linetype: ByLayer Current elevation: 0.0000 Current thickness: 0.0000 Fill: on Grid: off Ortho: off Snap: off Blips: off Drag: on Command echo: on Positive angle direction: Counterclockwise Angular units: Decimal degrees Dimension units: Decimal Pickbox height: 3 Entities in drawing: 136

Command Options

No options.

Figure 6.2161



6.27.47 StlOut

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Exports 3D solids and watertight meshes in a format that can be used for stereolithography (3D printing).

Accessing the Command

command bar: **stlout** : **stlout** Prompts you in the command bar: **Select 3d solids or meshes to export: (Select one or more 3D solids and/or 3D meshes) Select 3d solids or meshes to export: (Select more 3D entities, or else press Enter to continue) Create binary STL file? <Yes>/No: (Press Y or N)** Displays a Save File dialog:Type a name for the file, and click the **Save** button.

Command Options

Option	Description
Select 3d solids or meshes to export	Selects the 3D entities to be exported: • Pick one or more 3D solids and/or 3D meshes • Enter **all** to select all entities in the drawing; the program discards those not eligible for export • Press **Enter** to end entity selection
Create binary STL file?	Determines whether the output file is in binary or ASCII format: • **Yes**- writes a binary STL file, which is more compact • **No**- writes an ASCII text STL file, which is human-readable Some software might expect the file in either format.

When exporting 3D solids to STL, curved faces become faceted.Use the FACETRES variable to determine the size of the facets:

- Minimum = 0.01 for coarser curves but smaller file size
- Default = 0.5
- Maximum = 10 for very fine curves but larger file size

The MODELERPROPERTIES variable overrides Facetres to controlsmoothness by using different parameters, but I recommend you use FACETRES because it is simpler.

Related Commands

Export (6.13.15) - Exports drawings in a variety of vector and raster formats. -ModelerProperties (6.21.29) - controls the various settings of the Acis modeler through the command bar. ModelerProperties (6.21.28) - controls the various settings of the Acis modeler through the **Settings** dialog box.

Figure 6.2162



6.27.48 StopScript

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Stops the recording of scripts begun by the RecScript (6.26.5) command.

Accessing the Command

 command bar: **stopscript** menu bar: **Tools | Stop Recording** ribbon:**Tools | Action Recorder | Stop Recording** toolbar: **Tools |**  **: StopScript Displays no prompt in the command bar.** Use the RecScript (6.26.5) command to start recording scripts.For more information writing scripts, see **Customizing BricsCAD**.

Command Options

No options.

Related Commands

RecScript (6.26.5) - records scripts to SCR files. Script (6.27.9) - loads and runs a SCR script file. RScript (6.26.36) - reruns the script. Delay (6.10.23) - pauses the execution of the script for the specified number of milliseconds. Resume (6.26.28) - resumes a stopped script.

Figure 6.2163



6.27.49 Stretch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Stretches entities. **Note:** This command requires that you select entities with the Crossing or CPolygon methods.

Accessing the Command


command bar: **stretch** alias: **s** menu bar: **Modify | Stretch** ribbon: **Change | Modify | Stretch** toolbar: **Modify**


: Stretch Prompts you in the command bar: Select entities to stretch by crossing-window or crossing-polygon: **(Enter C or CP)** Opposite Corner: **(Pick another point)** Entities in set: 1 Select entities to stretch by crossing-window or crossing-polygon: **(Choose more entities, or press Enter to continue)** Base point of displacement: **(Pick a point, or enter x,y coordinates)** Second point of displacement: (**Pick another point.)** This command does not stretch text, shapes, and 3D solids.

Command Options

Option	Description
Select entities	Selects the entities to stretch; you must use Crossing or Crossing Polygon mode: • **C** - crossing mode. • **CP** - crossing polygon mode. When using the cursor, pick the second point to the left of the first: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i The selection rectangle (or polygon) determines how this command affects entities: • Entities crossing the selection rectangle are stretched. • Entities within the rectangle are moved. • Entities outside the rectangle remain in place. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Base point of displacement	Specifies the starting point of the stretch.
Second point of displacement	Specifies the stretch distance by the distance between two points, base and second point. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Procedures

- To stretch entities (1.23.23)
- To stretch entities using grips (1.23.23)

Related Commands

Lengthen (6.20.21) - changes the length of open entities. Extend (6.13.19) - extends open entities. Scale (6.27.6) - changes the size of entities. Properties (6.24.49) - changes the geometry of entities.

Figure 6.2164

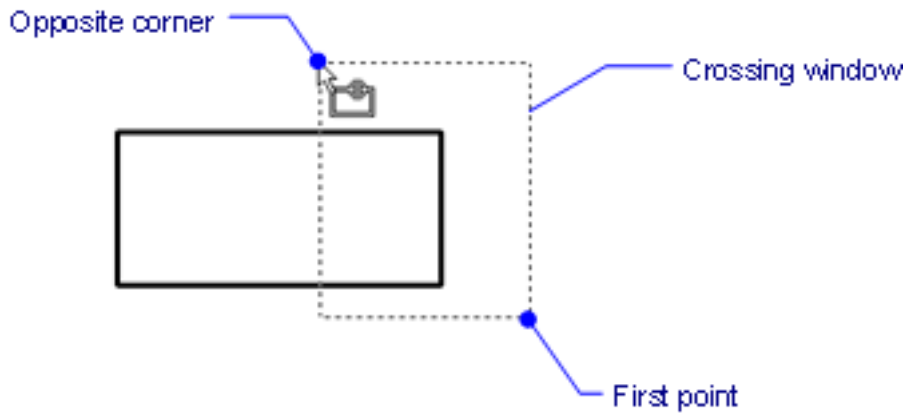


Figure 6.2165

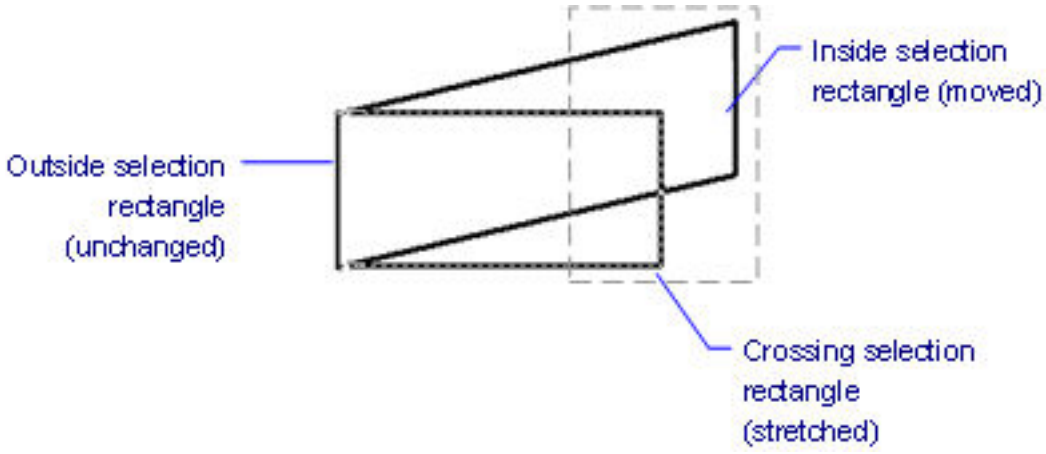


Figure 6.2166

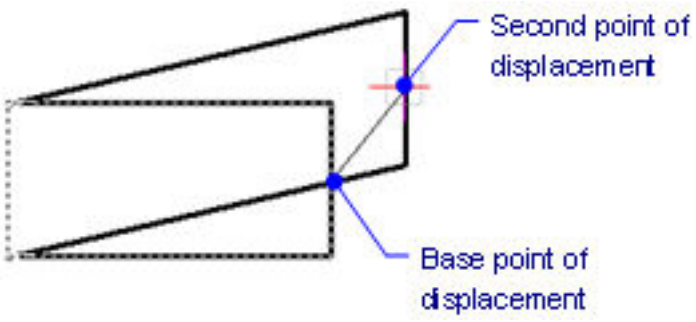


Figure 6.2167



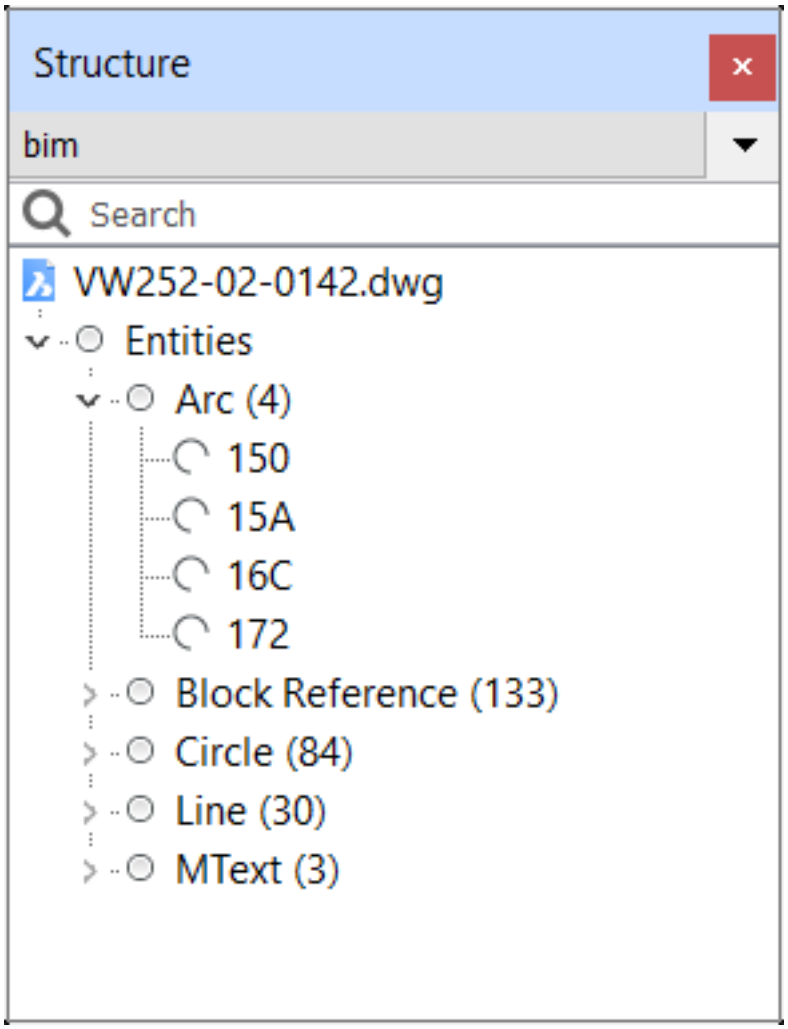
6.27.50 StructurePanel

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Structure panel, which lists entities in the drawing in a structured manner.

Accessing the Command

command bar: **structurepanel** context menu: right click any toolbar or the ribbon, then choose**Structure : StructurePanel** Displays a panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/structure-dlg.png})

Figure 6.2168

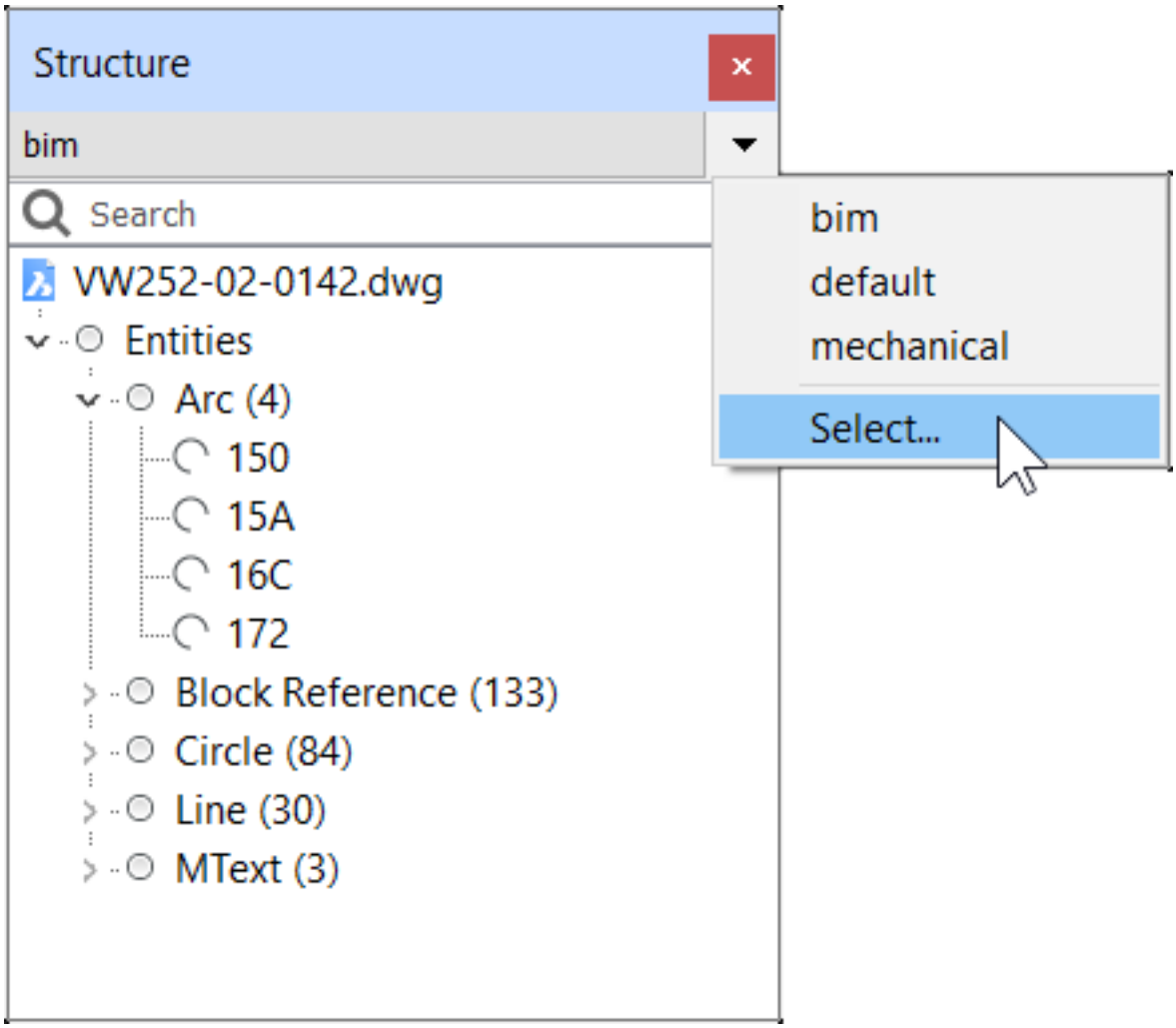


Context Menu Options

Click the down arrow to reveal the menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/menu.png})

Option	Description
BIM, default, **or** mechanical	Displays the drawing's structure in a format amenable to BIM (architecture) or mechanical drawings, or in the default format. Choose one: • bim • default • mechanical
Select	Loads a structure tree configuration file (*.cst) to change the way the content of of the Structure panel is arranged: 1. Opens the Select Structure Tree Configuration File dialog box; **(choose a *.cst configuration file from the file dialog box.)** 2. Click **OK**to open it Notice that the arrangement of information in the panel changes.

Figure 6.2169

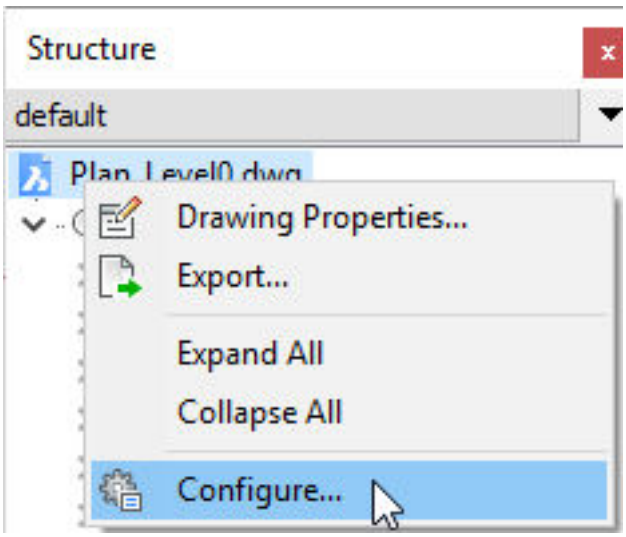


Drawing Menu Options

Right-click a drawing name to see this context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/C

Option	Description
Drawing Properties	Displays the Drawing Properties dialog box; see the DwgProps (6.10.42) command.
Export	Exports the structure and the entities in an XML file; displays the Export Structure Tree dialog box: 1. (Optional) Enter a name or accept the default 2. (Optional) Choose a folder or drive in which to save the file 3. Click **Save**. The first few lines of the XML file look like this: `<?xml version="1.0"?> <Structure config="default.cst">`
Expand All	Expands all nodes in the structure.
Collapse All	Collapses all nodes in the structure.
Configure	Opens the Configure Structure Tree dialog box for customizing the way the data is displayed; see below

Figure 6.2170

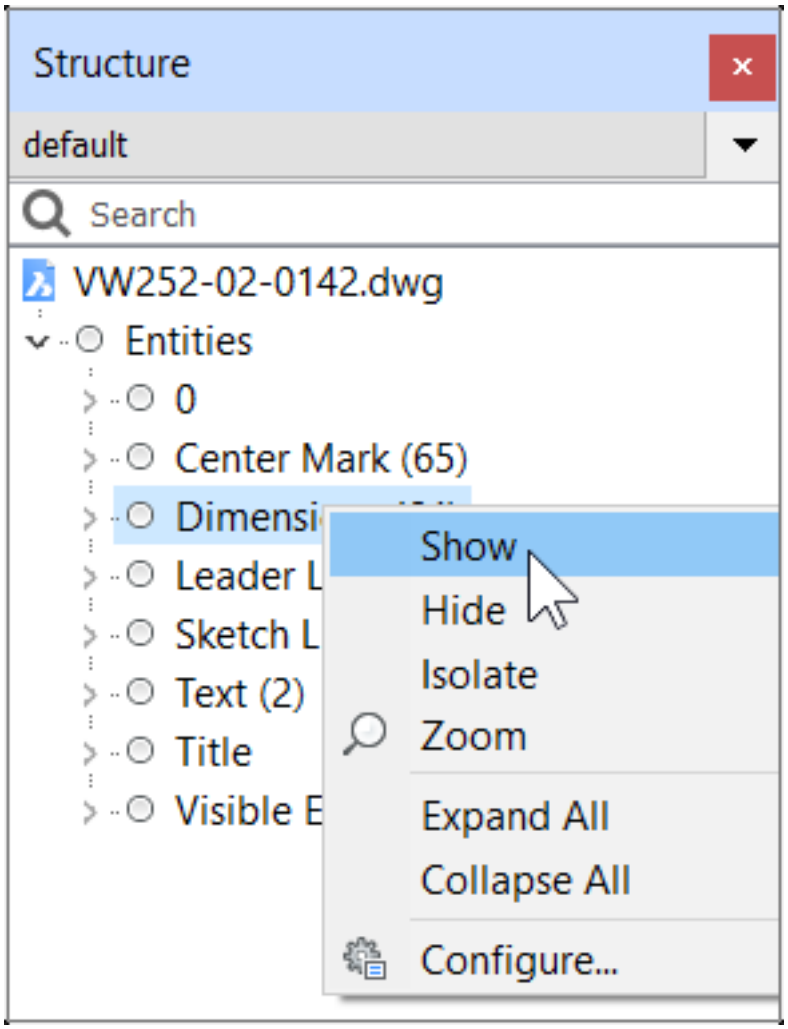


Entity Menu Options

Right-click an entity name to see this context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Cortclick.png})

Show	Shows the selection set, when it was previously hidden
Hide	Hides the selection set, and keeps all other drawing entities visible
Isolate	Hides all other entities in the drawing, except the selection set
Zoom	Zooms the view to the extents of the selected entities

Figure 6.2171

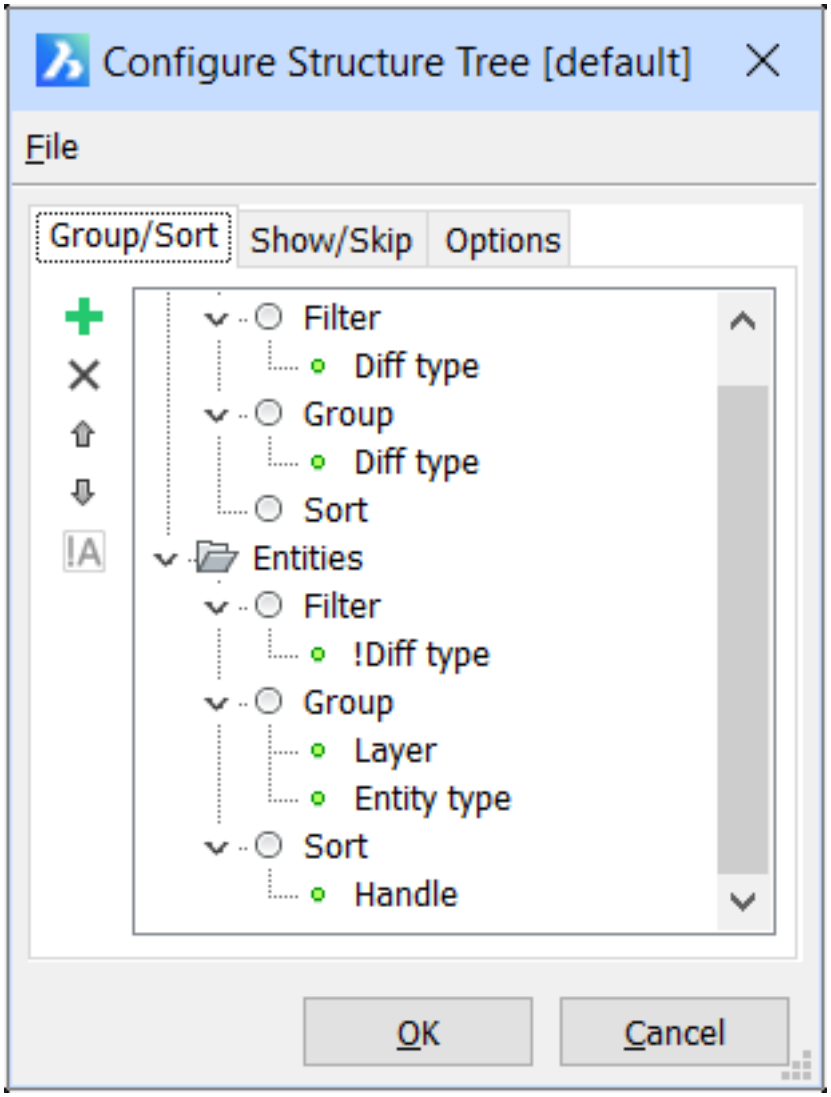


Configure Structure Tree Dialog Box

Click **Configure**option to see this dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commar
configure-GroupSort-dlg.png})

Group/Sort Tab Option	**Description**
Add Rule or Property	Adds rules and properties to display in the Structure panel. • To add a rule: 1. Select an existing rule. 2. Click **Add Rule or Property** 3. Rename the rule: right-click the new name, and then choose **Rename** 4. Create the rules: • Right-click **Filter** to add filters; see below • Right-click **Group** to add groups; see below • Right-click **Sort** to add a sorting property; see below 5. Click **OK**.
Delete Rule or Property	Removes a rule or property from the structure definition
Move Property Up	Moves the property up the list
Move Property Down	Moves the property down the list
Toggle Negation	(Applies only to filters) Switches the value of filters: • Include • Do not include
Add Filter Property, Add Group Property, **and** Add Sort Property	Filters, groups, and sorts the properties, entities, and extensions to be shown by a rule; displays the Select Property dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/str selectproperty-general.png})(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/str selectproperty-entities.png})(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/str selectproperty-extensions.png}) 1. Choose one. 2. Click **OK**.

Figure 6.2172

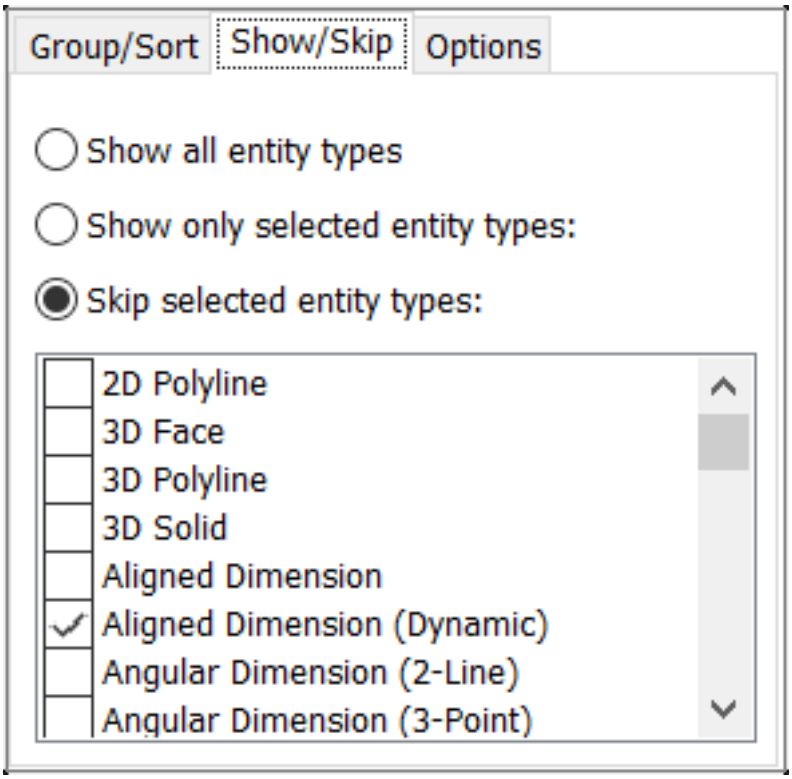


Show/Skip Tab

In theConfigure Structure Tree dialog box, click the**Show/Skip**tab to see this tab: (\vref {fig:C:/Dev/Task-2448/images/staticbric
configure-SbowSkip-tab.png})

Show/Skip Tab Option	Description
Show entity types	Determines which entities are displayed by the Structure panel • **Show all entity types** - show all entities in the Structure panel • **Show only selected entity types** - show only those checked in the list • **Skip selected entity types** - don't show those checked in the list
Entity types	Toggles whether entities are listed or not listed or not-not listed: • **Yes** - entity type is listed or not listed, depending on the choice made above • **No** - entity typeis not listed or listed, depending on the choice made above

Figure 6.2173

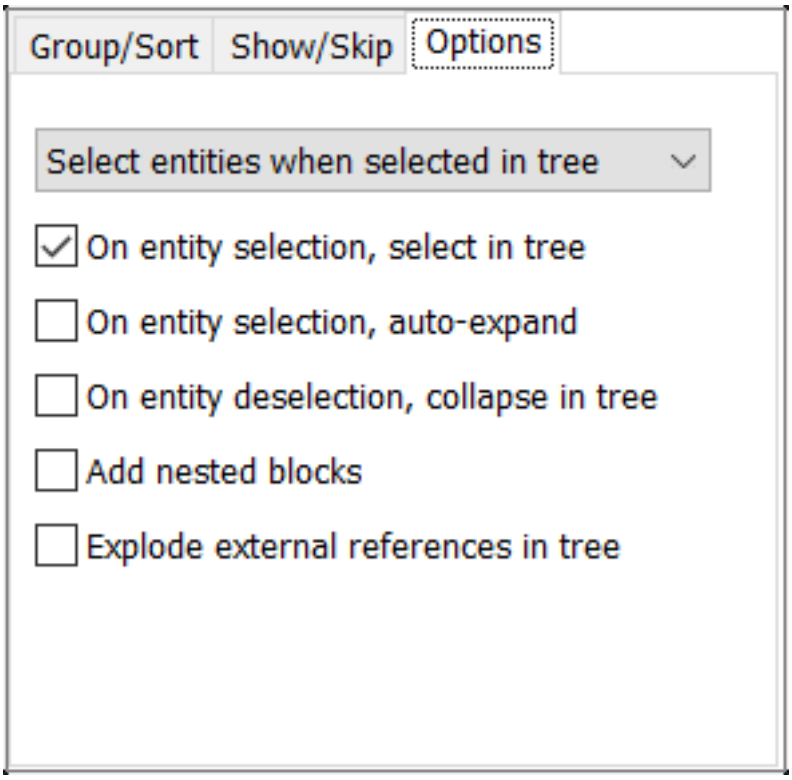


Options Tab

In theConfigure Structure Tree dialog box, click the **Options** tab to see this tab: (\vref {fig:C:/Dev/Task-2448/images/staticbricsy configure-Options-tab.png})

Options Tab Option	**Description**
Select Entities	Determines how entities are selected in the drawing when selected in the Structure panel: • **Ignore tree selection** - entities are not selected in the drawing • **Highlight entities when selected in tree** - entities in the drawing are highlighted, but not selected • **Select entities when selected in tree** - matching entities are selected in the drawing
On entity selection, select in tree	When on, selects matching entity(ies) in the structure tree when you select them in the drawing
On entity selection, auto-expand	When on, expands the node related to the entity(ies) selected in the drawing
On entity selection, collapse in tree	When on,collapses the node related to the entity(ies) selected in the drawing
Add nested blocks	When on, lists the contents of nest blocks in the structure tree
Explode external references in tree	When on, lists the contents of externally-referenced drawings in the structure tree

Figure 6.2174



Procedures

- To open the Structure panel (1.23.8)
- To configure the tree-structure on the Structure panel (1.23.8)
- To open or close a panel (1.12.7)
- To dock a panel (1.12.7)

Related Commands

StructurePanelClose (6.27.52) - closes the Structure panel. +StructurePanel (6.27.51) -opens a *.cst structure tree configuration file, and then loads it into the Structure panel.

Figure 6.2175

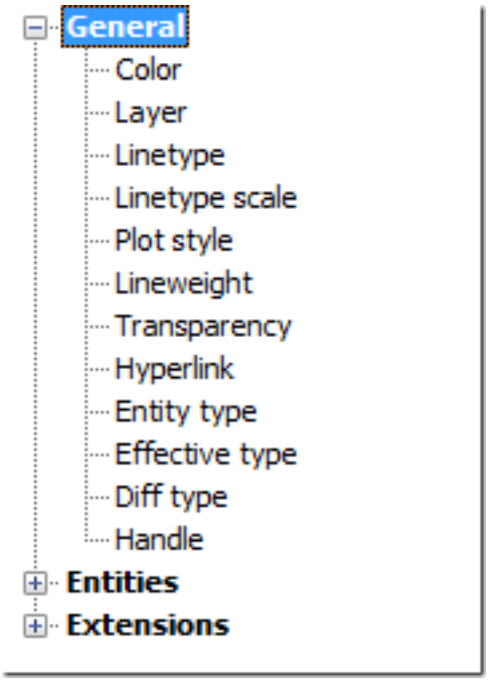


Figure 6.2176

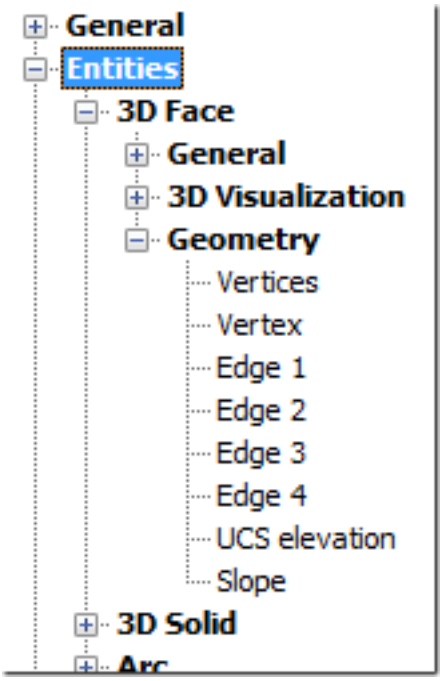


Figure 6.2177

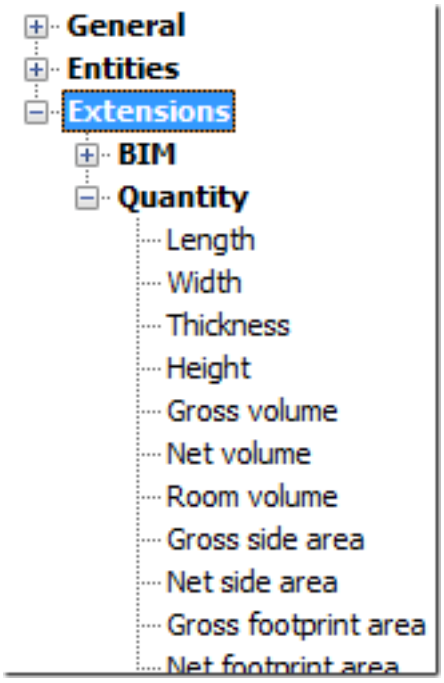


Figure 6.2178



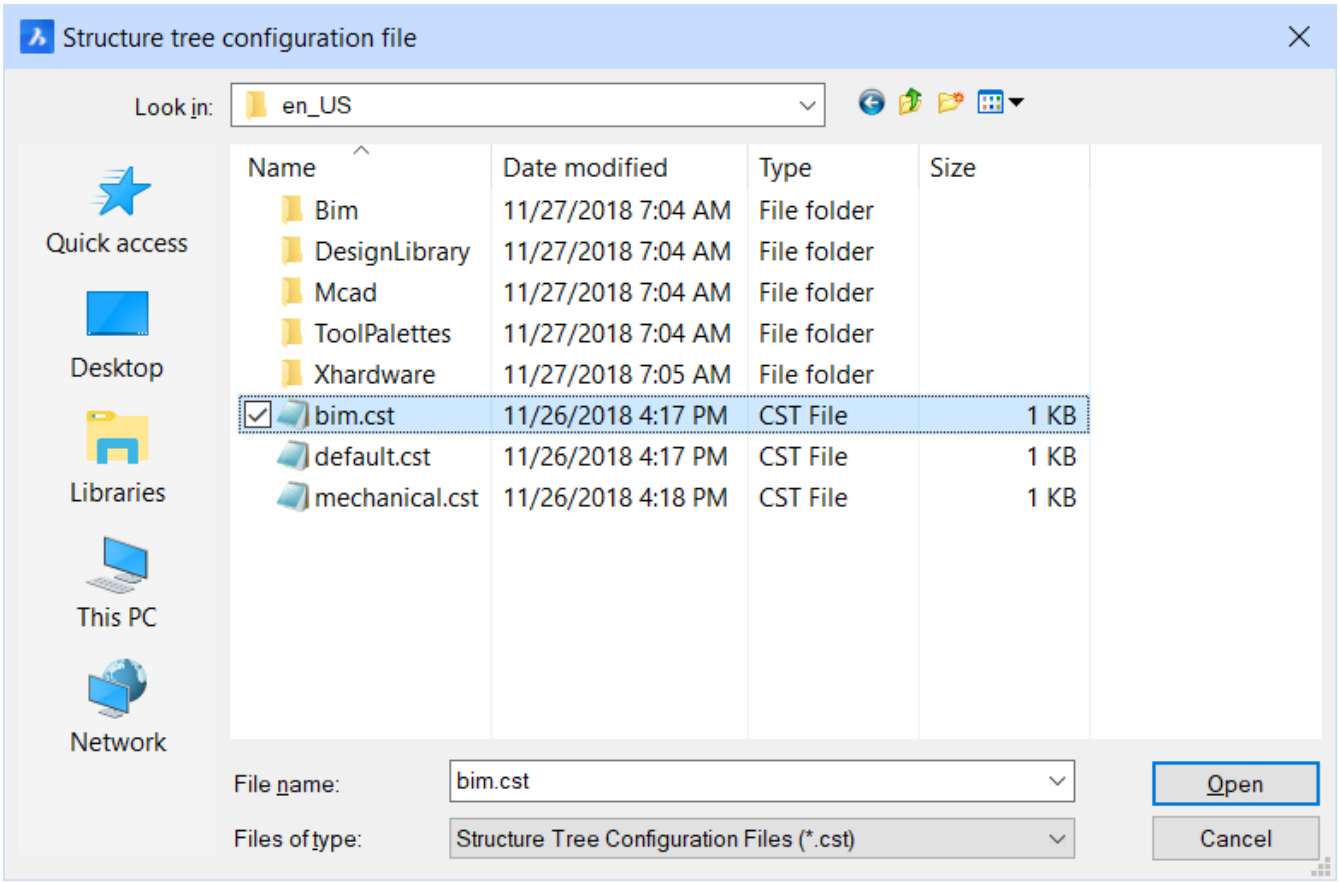
6.27.51 +StructurePanel

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads a CST structure tree configuration file into the Structure panel. **Note:**Loading different CST files changes the way that the StructurePanel (6.27.50) command presents drawing data.

Accessing the Command

Keyboard: **+structurepanel :+StructurePanel** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/he
StructurePanel-dlg.png})

Figure 6.2179



Dialog Box Options

Option	Description
File name	Specifies the name of the CST file to load.Loading different CST files changes the way that the Structure panel presents drawing data. The program includes three CST files (short for "configurationstructure tree"): • **bim.cst** - meant for BIM drawings • **default.cst** - meant for all other kinds of drawings • **mechanical.cst** - meant for mechanical drawings The CST files are found in theC:\Program Files\Bricsys\BricsCAD V19en_US\UserDataCache\Support\en_US folder
Open	Opens the CST file and then loads it into the Structure panel
Cancel	Cancels the command

Related Commands

StructurePanel (6.27.50) - opens the Structure panel.

Figure 6.2180



6.27.52 StructurePanelClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes the Structure panel.

Accessing the Command

command bar: **structurepanelclose** context menu: right click any toolbar or the ribbon, then choose**Structure** : **structurepanelclose** The Structure panel is closed. If the Structure panel is currently docked, the Structure Panel tab is removed.

Command Options

No command options.

Related Commands

StructurePanel (6.27.50) - opens the Structure panel.

Figure 6.2181



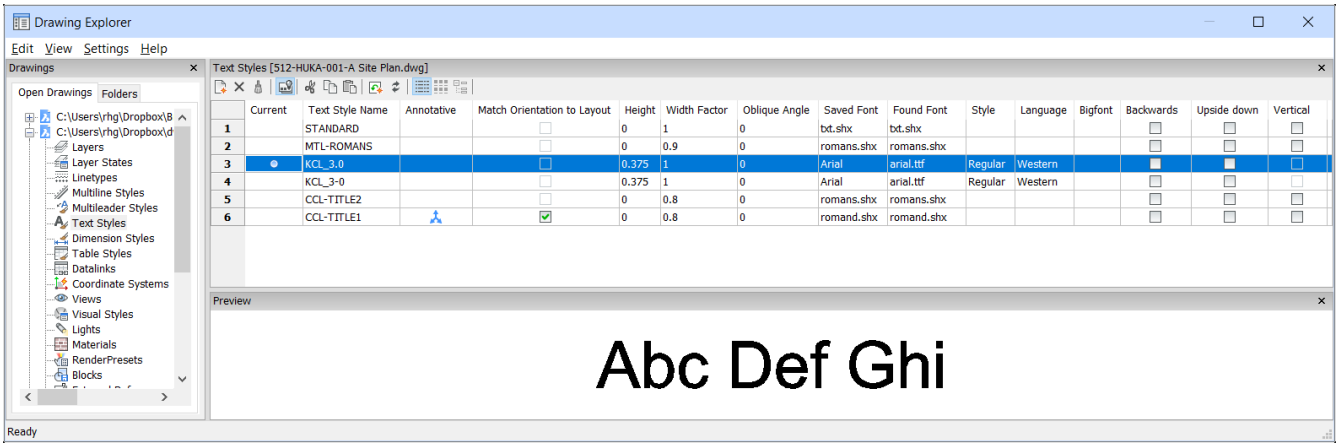
6.27.53 Style

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and modifies text style with the Drawing Explorer, for use by text-related commands. **Note:** Styles preset the look of text, such as the font to use, its height, and whether it uses an annotative scale factor.

Accessing the Command

command bar: **style** aliases: **ddstyle expfonts expstyle expstyles st** menu bar: **Tools | Drawing Explorer | Text Styles** ribbon:**Home | Settings | Drawing Explorer | Text Styles** status bar: **Standard : Styles** Displays the Text Styles section of the Drawing Explorer: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/style-dlg.png}) Make changes to styles, or create new ones, and then click **X** to exit. Press **F1** to access help.

Figure 6.2182

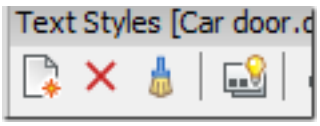


Toolbar Options

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/style-toolbar.png})

Toolbar Option	Description
New	Creates a new text style with the generic name New-Styl1; to rename it, click the name twice, slowly.
Delete	Erases the selected text style(s).The current style and the Standard style cannot be deleted. Whenthe style is in use, a dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ Choose an option. • **Delete** - **caution**! Erases the style, as well as all text that uses the style! • **Change** - selects another style before erasing it. Entities using the doomed style are given the alternative that you select. Displays list of available styles: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/Cm 1. Choose style name 2. Click **Change** The style is erased and text takes on the selected style • **Cancel** - cancels the operation; the style is not erased
Purge	Removes unused styles from the drawing. The following styles cannot be purged: • Standard text style • The current style • Styles used in the drawing
Hide Xref Symbols On/Off	Toggles the display of text styles used by xrefs.

Figure 6.2183



Text Style Options

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/style-headers.png})

Option	Description
Current	Designates the current text style. This is the style used by default by the Text (6.29.12) and MText (6.21.35) commands.
Text Style Name	Specifies the style name: • To rename, click the name twice, and then rename • To create a new text style, click the **New** button to create a new text style; the new text style takes on the properties of the current style.
Annotative	Sets the Annotative property of the text entities created with this style. When the text style is an annotative one, the Height property specifies the height in paper space, while in model space the height depends on the current annotation scale (specified by the CANNOSCALE and CANNOSCALEVALUE variables).
Match Orientation	(This option is for annotative text styles only.) Matches the orientation of the text to the orientation of the pa-perspace layout.
Height	Specifies the height of the text. Enter 0 or a number larger than 0: • **0** - does not set the height; TEXT command prompts for height. • Not 0 - specifies the height of text; TEXT command does not ask for height.
Width Factor	Specifies the width factor of the text. For example, entering 2 makes the text twice as wide. Enter a number larger or smaller than 1: • Less than 1 - text is squeezed narrower. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • More than 1 - text is spread wider. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Figure 6.2184

	Current	Text Style Name	Annotative	Match Orientation to Layout	Height	Width Factor	Oblique Angle	Saved Font	Found Font	Style	Language	Bigfont	Backwards	Upside down	Vertical
1	●	DYN_DIM		☐	0	1	0	Arial	arial.ttf	Regular	Western		☐	☐	☐
2		Standard		☐	0	1	0	Arial	arial.ttf	Regular	Western		☐	☐	☐
3		NewStyle1		☐	0	1	0	Arial	arial.ttf	Regular	Western		☐	☐	☐

Oblique Angle

Specifies the obliquing angle for the text. For example, entering 15 slants the text forward by 15 degrees. Enter a positive or negative number:

- Negative number - text slants to the left.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

- **0** - text is upright.
- Positive number - text slants to the right.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Saved Font

Specifies the SHX or TTF font file to use; choose a name from the droplist: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

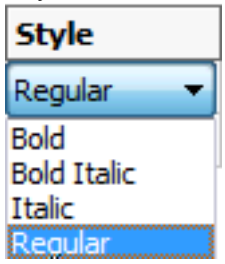
- **SHX** - compiled shape files created for AutoCAD and BricsCAD
- **TTF** - TrueType font files created for Windows and Macintosh

The program can use any SHX and TTF font found on your computer.

Found Font

Shows the name of the font file found.If the font defined in by **Saved Font** is not found .shx fonts are replaced by the font defined by the FONTALT (1.13.11) (Alternative font) system variable. TTF fonts are replaced according to the replacement table defined by the FONTMAP system variable. By default this is the**Default.fmp**file stored in the **Support** subfolder of the roamable root folder (e.g. **C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V18**
Fonts that have been replaced display in Red . (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

Style

Specifies style for TTF fonts; choose a style from the droplist: 

- **Regular** - applies neither bold nor italics
- **Bold** - applies boldface
- **Italic** - applies italics
- **Bold Italic** - applies bold and italics

This property is not available for SHX fonts.

Language

Specifies the language for TTF fonts; choose one from the droplist: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Available languages may vary for each font. This property affects accents and other attributes specific to languages. This property is not available for SHX fonts; use Bigfont property instead.

Bigfont	Lists the names of SHX bigfont files; choose one from the droplist: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i "Bigfonts" is the name given to large SHX files that contain characters for multiple languages. This property is not available for TTF fonts; use Language property instead.
Backwards*	Toggles backwards text. Backwards text is useful when text is plotted on the back of mylar or other clear media. • **Yes** - text is drawn backwards (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **No** - text is drawn forwards
Upside down*	Toggles upside-down text. • **Yes** - text is drawn upside down (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **No** - text is drawn rightside up
Vertical	Toggles vertical text. Only some SHX fonts support vertical text; this option is not available for TTF fonts. • **Yes** - draws text vertically. The Text command sets the rotation angle to 270 degrees automatically (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **No** - draws text horizontally
Preview	Shows what the current style looks like: the font with the properties applied.

* This setting applies to Text entities only, not to MText (6.21.35) entities.

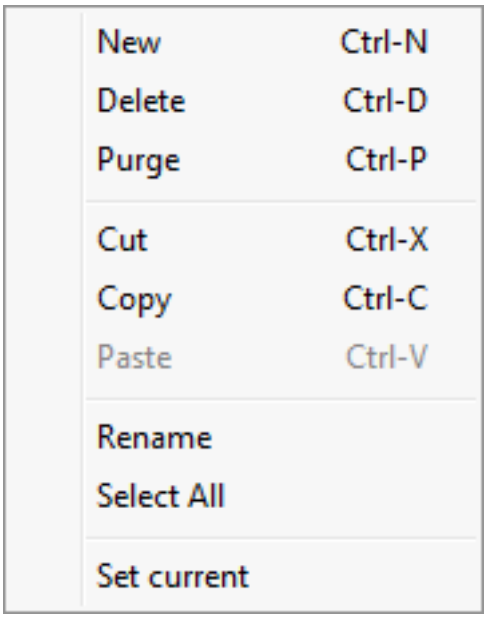
Shortcut Menu Options

Right-click a layer name to see this shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cm

Option	Keystroke	Description
New	Ctrl+N	Creates new text styles.
Delete	Ctrl+D	Deletes styles from Drawing Explorer. The following styles cannot be deleted: • Standard • Current style

Purge	Ctrl+P	Purges styles from the current drawing. The following styles cannot be purged: • Standard • Current style • Any style used by text
Cut	Ctrl+X	Cuts the style and its properties to the Clipboard.
Copy	Ctrl+C	Copies the style and its properties to the Clipboard.
Paste	Ctrl+V	Pastes the style from the Clipboard into Drawing Explorer.
Rename		Renames the selected style.
Select All		Selects all styles.
Set Current		Specifies the current working style. Only one style can be current at a time.

Figure 6.2185



Status Bar Menu

Right-click Standard (or other style name) on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticb
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image522.jpg})

Shortcut Menu	Description
Standard	Chooses the default text style. When the drawing contains additional text styles, their names are listed here.
Properties	Opens the Text Styles section of the Drawing Explorer.

Figure 6.2186

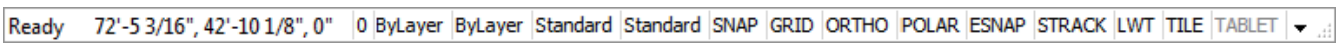
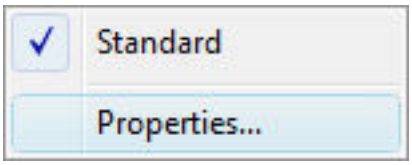


Figure 6.2187



Procedures

- To open the Text Style Explorer dialog (1.21.18)
- To create a text style (1.21.20)
- To edit a text style (1.21.18)
- To make a text style current (1.21.18)
- To delete a text style (1.21.18)

Related Commands

-Style (6.27.54) - creates and modifies text styles through the command bar. Text (6.29.12) - places single-lines of text in drawings. -Text (6.29.12) - places text and evaluates LISP expressions. MText (6.21.35) and -MText (6.21.33) - places paragraph text. AttDef (6.4.27) - defines attribute text. DdEdit (6.10.14) - edits text. Change (6.9.10) - changes the position of text. TxtExp (6.29.27) - explodes text into polylines. Spell (6.27.63) - checks the spelling of text in drawings. Properties (6.24.49) - modifies the properties of text. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.2188

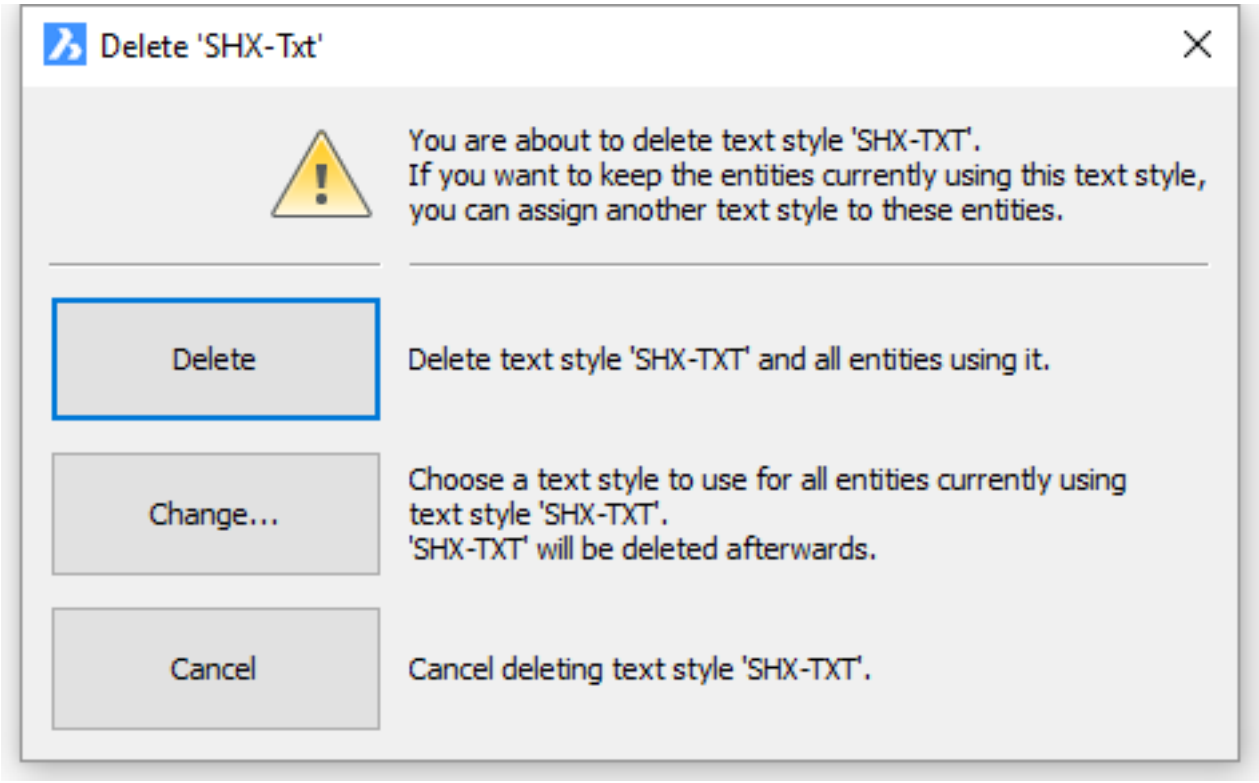


Figure 6.2189

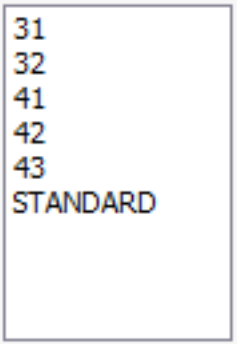


Figure 6.2190



Figure 6.2191



Figure 6.2192



Figure 6.2193



Figure 6.2194

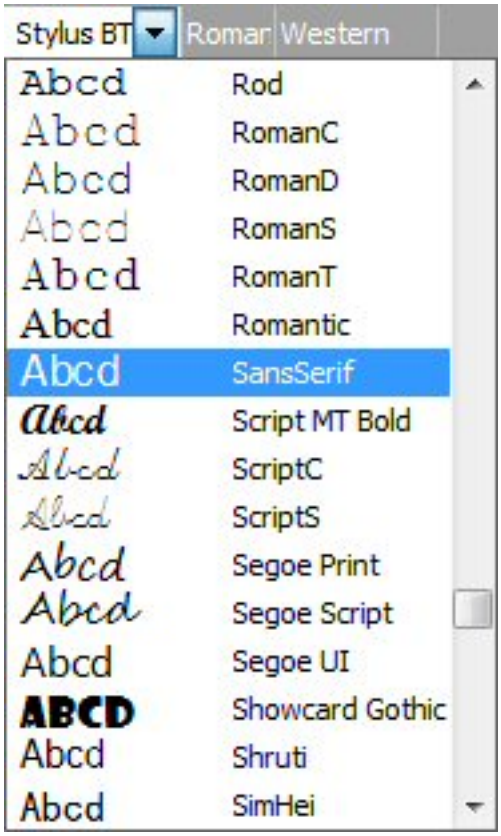


Figure 6.2195

Saved Font	Found Font
Arial	arial.ttf
Arial	arial.ttf
romans.shx	simplex.shx
Arial	arial.ttf

Figure 6.2196

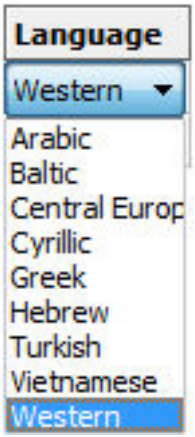


Figure 6.2197

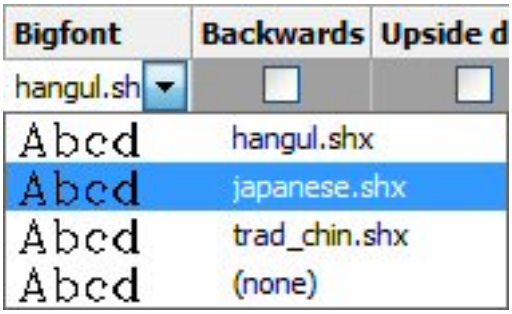


Figure 6.2198



Figure 6.2199



Figure 6.2200

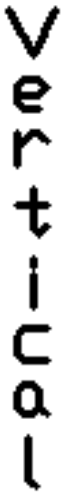


Figure 6.2201



6.27.54 -Style

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and modifies text styles in the command bar, for use by text-related commands. **Note:** Styles preset the look of text, such as the font to use, its height, and whether it uses an annotative scale factor.

Accessing the Command

command bar: **-style** transparent: **'-style** alias: **font** : **-Style Prompts you in the command bar:** ? to list existing styles/<Text style to create or modify> <STANDARD>: **(Enter a name, or press ?)** Font <arial.ttf>: **(Enter the name of a SHX or TTF font file, or else press Enter to accept the default file name.)** font: arial.ttf bigfont: is selected Text height or [Annotative] <0">: **(Enter a number, press A, or else press Enter to accept the default height)** Width factor <1.0000>: **(Enter a number, or else press Enter to accept the default width)** Obliquing angle <0">: **(Enter a number, or else press Enter to accept the default angle)** Draw text backwards? [Yes-backwards text/No-normal text] <No>:**(Press Y or N)** Draw text upside-down? [Yes-upside-down text/No-right side up] <No>:**(Press Y or N)** The additional prompt appears when an SHX font file was specified: Draw text vertically? <No>: **(Press Y or N)**

Command Options

Option	Description
Text style to create or modify	Specifies the style name: • **Create** - to create a a new text style, enter a new name, such as 'MyNewStyle' • **Modify** - to modify an existing text style, enter its name, such as 'Standard'

? to list existing styles	Lists style names in the drawing; prompts you: **Text style(s) to list** - **(specifies which styles to list; enter an option:)**
	• ***** - to list all style names in the drawing
	• **name*** - to list style names that begin with 'name'
	• **name** - lists information about the text style named 'name'
	The list for the 'Standard' text style looks like this:
	`Text styles:`
	`Style name: Standard Font typeface: Arial Height: 0`
	`Current text style: Standard`

Font	Specifies the SHX or TTF font file the style should use: • **SHX** - font files created for AutoCAD and BricsCAD • **TTF** - font files created for Windows and Macintosh BricsCAD can use any SHX and TTF font found on your computer.

Text height	Specifies the height of the text; enter 0 or a number larger than 0: • **0** - has a special meaning in that it does not set the height of text; the Text (6.29.12) command will prompt you for for height • Not 0 - sets the text height, such as 0.25 units; Text command will not ask you for a height as it will use this one
Annotative	Sets the Annotative property of the text entities created with this style. Prompts you: 1. **Create annotative text style[Yes/No] <Yes>** - **(enter an option:)** • **Yes**- makes the text style annotative, so that its height matches the current annotation scale factor automatically • **No** - keeps the text height fixed **Match text orientation to layout?[Yes/No] <No>** - **(enter an option:)** • **Yes**- makes the text rotate automatically to make the rotation angle of the layout (in paper space) • **No** - keeps the text angle fixed 3. **Text height or [Annotative] <0>** - **(press Enter to continue.)** When the text style is an annotative one, the Height property specifies the height in paper space, while in model space the height depends on the current annotation scale (specified by the CANNOSCALE and CANNOSCALEVALUE variables).
Width factor	Specifies the width factor of the text.For example, entering 2 makes the text twice as wide.Enter a number larger or smaller than 1: • Less than 1 - squeezes text narrower (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • More than 1 - spreads text wider (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Obliquing angle	Specifies the obliquing angle for the text.For example, entering 15 slants the text forward by 15 degrees.Enter a positive or negative number: • Negative number - slants text to the left (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **0** - text is upright • Positive number - slants text to the right (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Draw text backwards?	Toggles backwards text.Backwards text is useful when text is plotted on the back of mylar or other clear media. **(Enter Y or N:)** • **Yes** - draws text backwards. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **No** - draws text forwards.
Draw text upside-down?	Toggles upside-down text. **(Enter Y or N:)** • **Yes** - draws text upside down. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **No** - draws text right side up.
Draw text vertically?	Toggles vertical text.Only some SHX fonts support vertical text; no TTF fonts can be drawn vertically, and so this prompt is not displayed for TTF fonts.The Text (6.29.12) command sets the rotation angle to 270 degrees automatically. **(Enter Y or N:)** • **Yes** - draws text vertically (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **No** - draws text horizontally

Status Bar Menu

Right-click Standard (or other style name) on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticb (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image522.jpg})

Shortcut Menu	Description
Standard	Chooses the default text style. When the drawing contains additional text styles, their names are listed here.
Properties	Opens the Text Styles section of the Drawing Explorer. See **Style** command.

Figure 6.2202

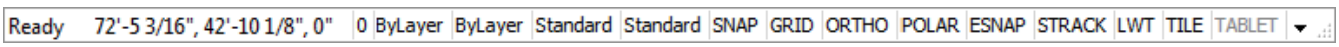
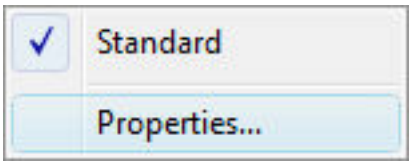


Figure 6.2203



Procedures

To open the Text Style Explorer dialog (1.21.18) To create a text style (1.21.18) To edit a text style (1.21.18) To make a text style current (1.21.18) To delete a text style (1.21.18)

Related Commands

Style (6.27.53) - creates and modifies text styles through the Drawing Explorer. Text (6.29.12) - places single-lines of text in drawings. -Text (6.29.12) - places text and evaluates LISP expressions. MText (6.21.35) and -MText (6.21.33) - places paragraph text. AttDef (6.4.27) - defines attribute text. DdEdit (6.10.14) - edits text. Change (6.9.10) - changes the position of text. TxtExp (6.29.27) - explodes text into polylines. Spell (6.27.63) - checks the spelling of text in drawings. Properties (6.24.49) - modifies the properties of text.

Figure 6.2204



Figure 6.2205



Figure 6.2206



Figure 6.2207



Figure 6.2208



Figure 6.2209



Figure 6.2210

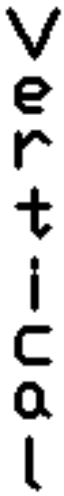


Figure 6.2211



6.27.55 StylesManager

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and edits plot style files. **Note:** Plot styles fine-tune how drawings are printed, but are not necessary for printing. The program supports two groups of plot styleto control the printed output:

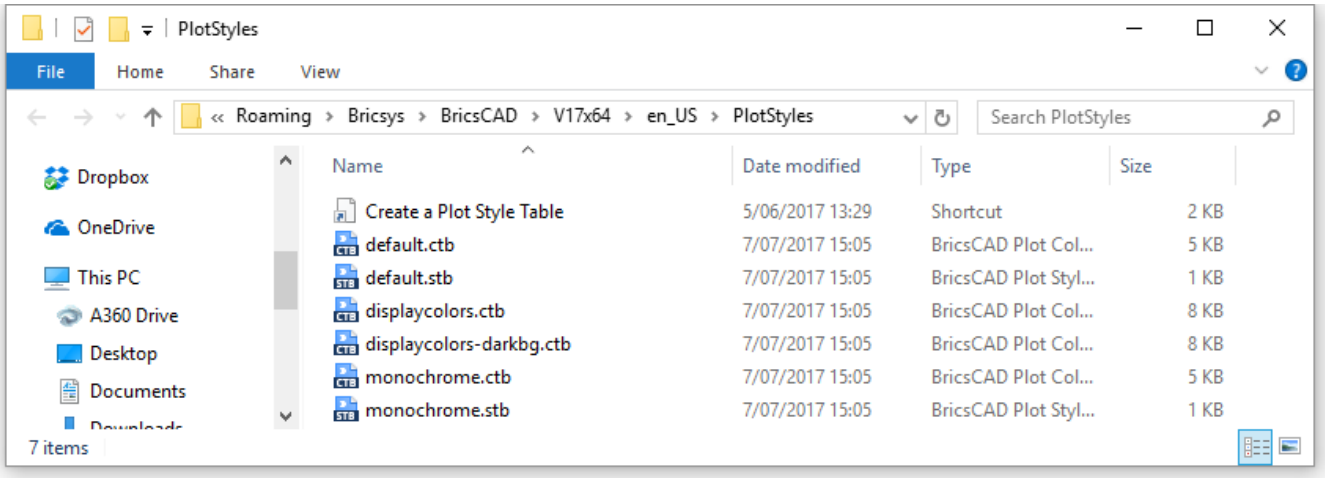
- CTB color tables simply use colors
- STB style tables use an extensive range of properties

Use the PageSetup (6.24.1) command to assign a plot style to a drawing's model and layout tabs.

Accessing the Command

command bar: **stylesmanager** menu bar: **File | Plotstyles Manager : StylesManager** Displays a window: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/StylesManagerDialog.png}) Double-click a CTB or STB file to open the Plot Style Table Editor; see thePlotStyle (6.24.29) command. Double-click Create a Plot Style Table to launch the Add Plot Style Table wizard; the XXXXXXXXXXXXX command

Figure 6.2212

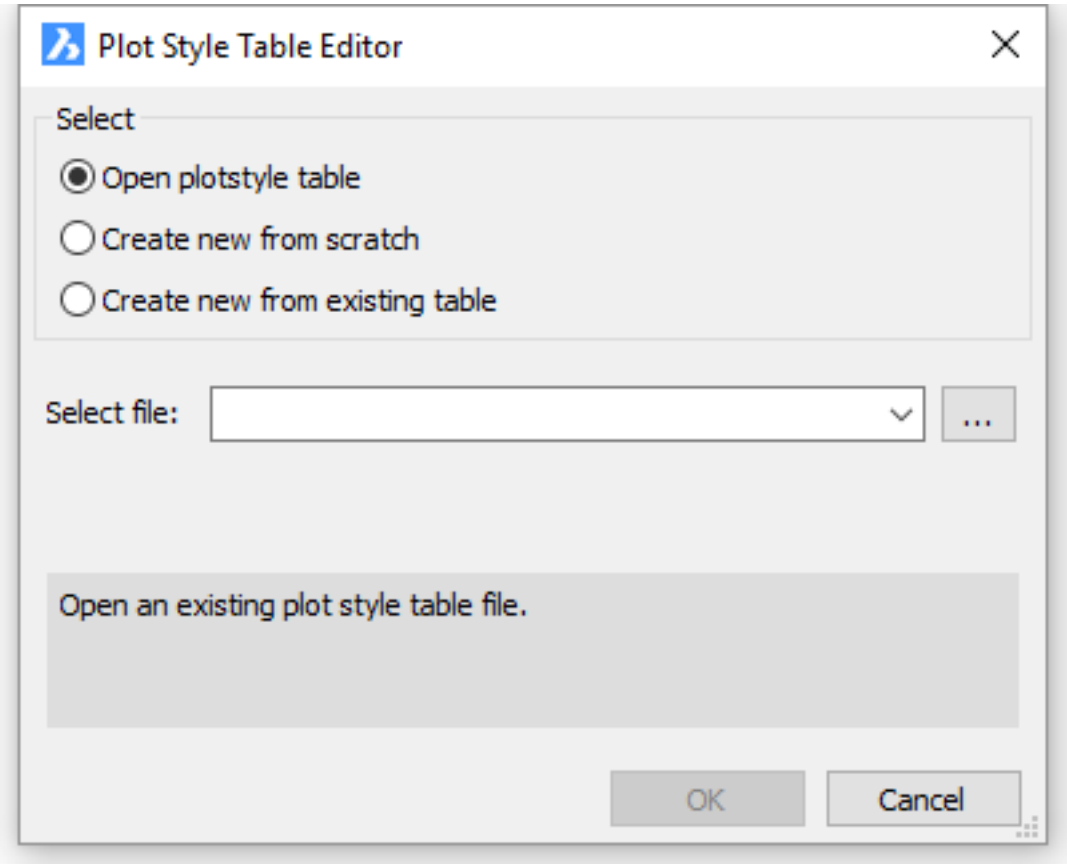


Plot Style Table Editor Dialog Box Options

Double-click**Create a Plot Style Table** to see this dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS

Option	Description
Open plotstyle table	Opens an existing plot style table; to choose the table file: 1. Click**Browse** 2. Choose a CTB or STB file 3. Click **Open** 4. Click **OK** open the Plot Style Table Editor (1.25.3) dialog box.
Create from scratch	Creates a new plotstyle table: 1. Select a table type from the droplist: • **CTB** (color-dependent plot style table) - controls prints with entity colors. Plotting parameters, such as pen thickness and linetype, are matched to the color of entities. This system is older and less flexible than named plot styles, but is the default in all new drawings. Creates a CTB file. • **STB** (named plot style table) - controls prints with plot styles. Plotting parameters are given to every entity and every layer. This system is newer and more flexible, but may be incompatible with other software that reads DWG files. Creates an STB file. 2. Type a name in the **Table name** field. 3. Click**OK**to open the Plot Style Table Editor (1.25.3) dialog box.
Create new from existing table	Create a new plot style from an existing file: 1. Select an existing file from the **Select file** droplist or else click**Browse**, then choose an existing CTB or STB file and click**OK**. 2. Type a name for the new plot style in the **Table name** field. 3. Click**OK**to open the Plot Style Table Editor (1.25.3) dialog box.

Figure 6.2213



Procedures

- To open the Plot Styles Manager (1.25.5)
- To create a new plot style table (1.25.5)
- To create a new plot style table in the Print dialog (1.25.5)
- To edit a plot style table (1.25.5)
- To edit a plot style table in the Print dialog (1.25.5)

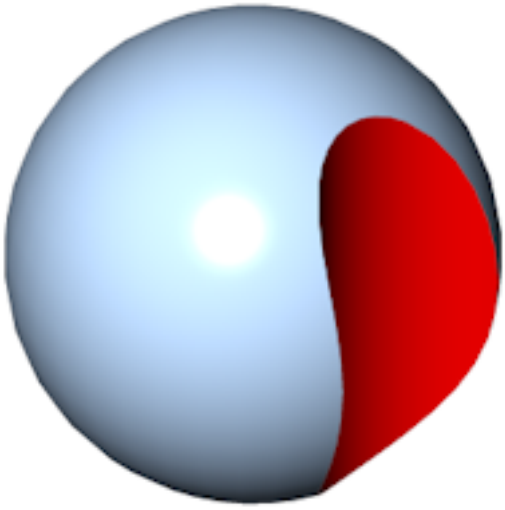
Related Commands

ConvertCtb (6.9.42) - converts CTB plot style files to STB files. ConvertPStyles (6.9.46) - converts drawings from CTB to STB plot styles. Layer (6.20.2) - assigns plot styles to layers. PageSetup (6.24.1) - assigns plot style to drawings. PlotStyle (6.24.29) - sets the current plot style. Print (6.25.3) - plots drawings with plot styles. Properties (6.24.49) - changes plot styles assigned to entities.

Figure 6.2214



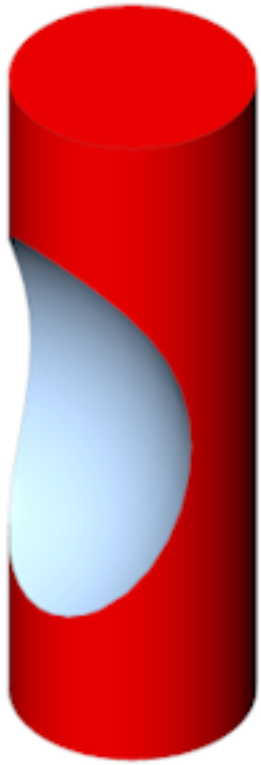
Figure 6.2215



6.27.56 Subtract

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Performs Boolean subtraction operations on 3D solids and 2D regions by removing the overlapping portions of one from another. **Note:** This result of this command is sensitive to the order in which you select entities. Here, a cylinder was subtracted from a sphere (left) and vice versa (right). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SubtractSolidsEnd.png}) (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/SubtractSolidsEnd2.png})

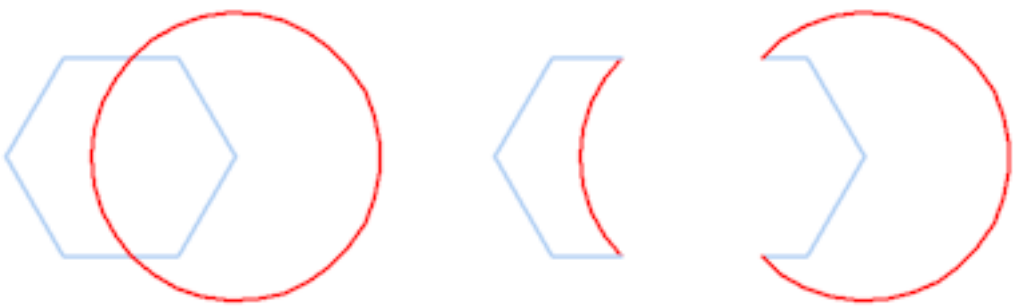
Figure 6.2216



Accessing the Command


command bar: **subtract** alias: **su** menu bar: **Model | 3D Solid Editing | Subtract** ribbon: **Solid | Edit | Boolean | Subtract** toolbar: **Solid Editing |**  **: Subtract Prompts you in the command bar:** Select ACIS object to subtract from: **(Choose one or more 3D solids and/or 2D regions)** Entities in set: 1 Select ACIS object to subtract from: **(Choose more ACIS entities, or else press Enter to continue)** Select ACIS objects to subtract: **(Choose the second set of one or more 3D solids and/or 2D regions)** Entities in set: 1 Select ACIS objects to subtract: **(Choose more ACIS entities, or else press Enter to continue)** The DELETETOOL (1.13.11)variable determines whether the source entities are deleted or retained. The order in which you select entities also applies to regions: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SubtractRegions.png})

Figure 6.2217



Command Options

Option	Description
Select ACIS object to subtract from	Specifies the ACIS entities that will have a portion removed: • Choose one or more 3D solids • Choose one or more 2D regions • Press **Enter** to continue
Select ACIS objects to subtract	Specifies the ACIS entities that will doe the removing: • Choose one or more 3D solids • Choose one or more 2D regions • Press **Enter** to perform the subtraction

Related Commands

Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. Revolve (6.26.30) - revolves 2D entities into 3D solid models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Union (6.30.6) - performs Boolean union operations on 3D solids and 2D regions; adds one to another to form a single entity.

Figure 6.2218



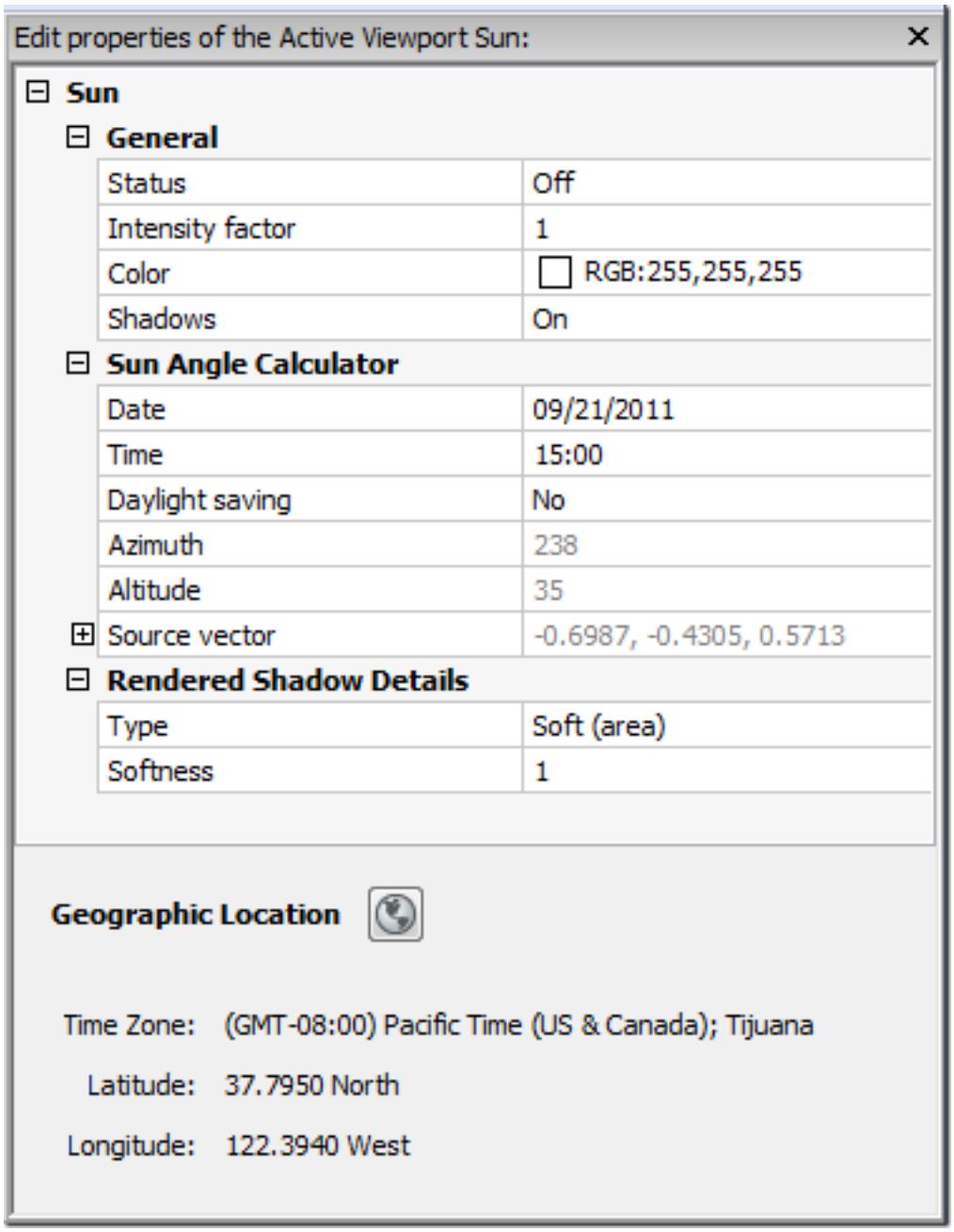
6.27.57 SunProperties

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edit the properties of the sun light through the Drawing Explorer. **Note:** Sun is a light that mimics the lighting effects of the sun, specifically according to the time and date. You do not place the sun, only turn it on and of.

Accessing the Command

 command bar: **sunproperties** alias: **sun** menu bar: **Tools | Drawing Explorer | Lights View | Rendering | Lights | Sun Properties** ribbonL **View | Sun | Sun Properties** toolbar: **Lights |** **: SunProperties** Displays the Lights section of the Drawing Editor dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/

Figure 6.2219



Dialog Box Options

Options	Description
General	**Status** - toggles the sun on and off: • **On** - displays the sun as a light in the model • **Off** - does not display the light **Intensity factor** -specifies the intensity or brightness of the sun; the higher the number, the brighter the light. (This option is not available when LIGHTINGUNITS (1.13.11) = 1 or 2) **Color** - sets the color of the sun light: select a color in the droplist or else choose **Select Color** to open the Select Color (6.9.33) dialog box. **Shadows** -toggles the display of shadows by the sun light: • **On** - displays shadows according to the properties in the Rendered Shadow Details section, below • **Off** - does not display shadows, which increases rendering speed
Sun Angle Calculator	**Date** - displays the current date; edit the date to show the sun at a different time of year **Time** - displays the current time; edit the time to show the sun at a different time of day **Daylight saving** - toggles the effect of daylight saving time on the time property **Azimuth** - reports the horizontal angle of the sun measured clockwise from the north; this setting cannot be changed as the program calculates it from the other settings in this section. (Read-only)**Altitude** - reports the vertical angle of the sun, measured from the horizon; this setting cannot be changed as the program calculates it from the other settings in this section. (Read-only) **Source vector** - reports the direction of the sunlight; this setting cannot be changed as the program calculates it from the other settings in this section.
Rendered Shadow Details	**Type** -specifies the type of shadow type; choose one from the droplist: • **Sharp** - displays shadows with sharp edges; use this option to increase performance over the soft shadow options • **Soft (mapped)** - displays realistic shadows with soft edges, according to the Map Size property • **Soft (area)**- displays realistic shadows with softer shadows based on extended light sources, according to the Softness property (For Soft-mapped option only) **Map size** -applies 'Soft (mapped)' shadow type: select a value from the droplist;the larger the value, the more realistic the shadow but the longer the rendering takes **Softness** - determines the sampling size of shadows; larger numbers are more accurate but take longer to render
Geographic Location	Reports location of the drawing on earth; to change the location, click the **Edit** button to access the Geographic Location dialog box; see GeographicLocation (6.15.15) command.

Related Commands

Light (6.20.28) - creates a new light. Geographiclocation (6.15.15) - sets the geographic location of the drawing. Lightlist (6.20.29) - opens the Drawing Explorer - Lights dialog. Render (6.26.22) - calculates a photorealistic image of the 3D model.

Figure 6.2220



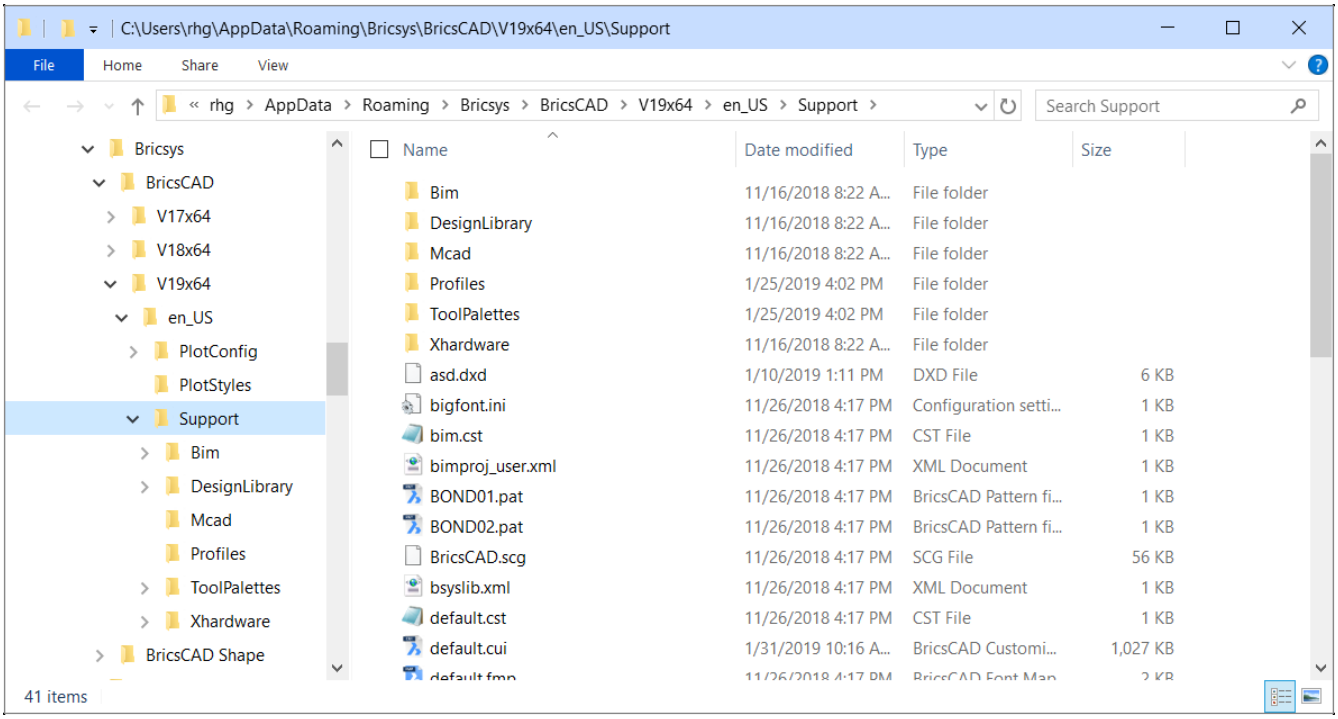
6.27.58 SupportFolder

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Opens the operating system's file manager at the Support folder. **Note:** The support folder holds many files that drawings and commands need, such as linetype patterns and tool palette libraries.

Accessing the Command

command bar: **supportfolder** : **SupportFolder** Displays a file manager dialog box showing the contents of the support folder: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/supportfolder-dlg.png}) Support folder paths are stored in theSRCHPATH (1.13.11)variable, and the default path is: **C:\Users\<user_name>\A**

Figure 6.2221



Command Options

No Options

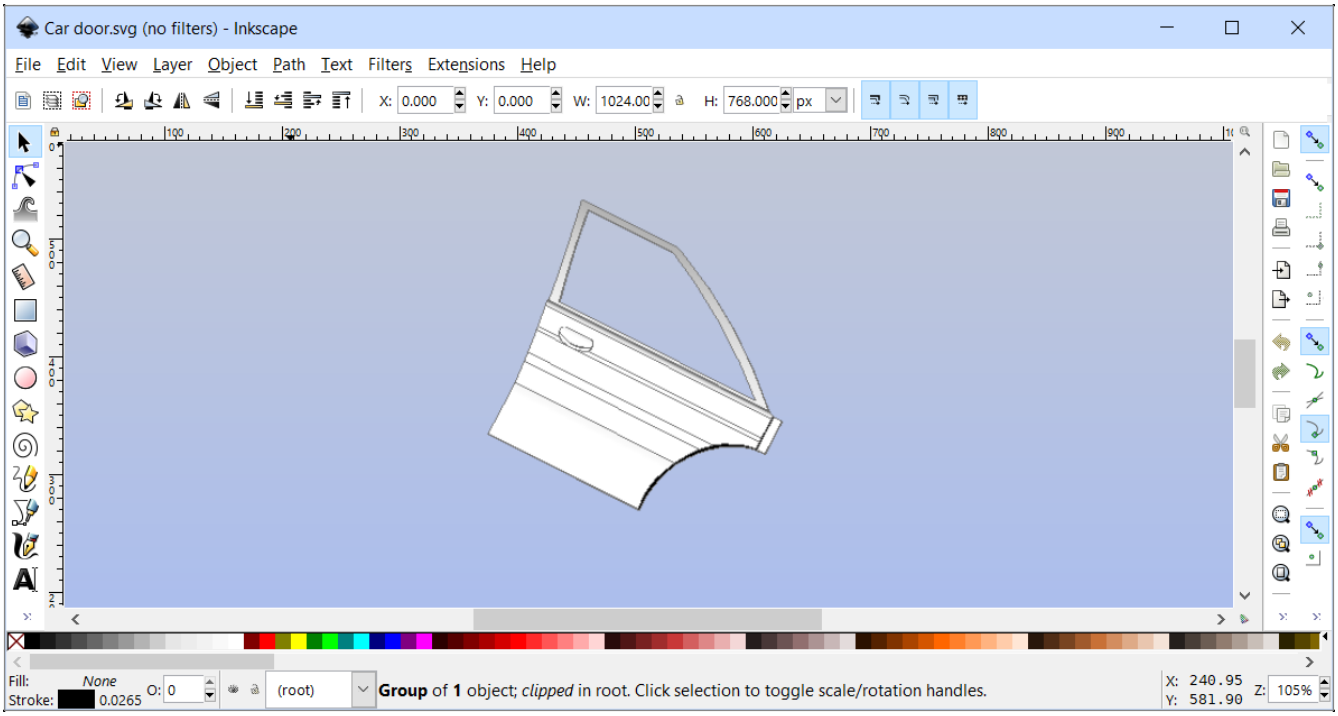
Related Commands

TemplateFolder (6.29.11) - opens the Template folder in a File dialog box.

Figure 6.2222



Figure 6.2223



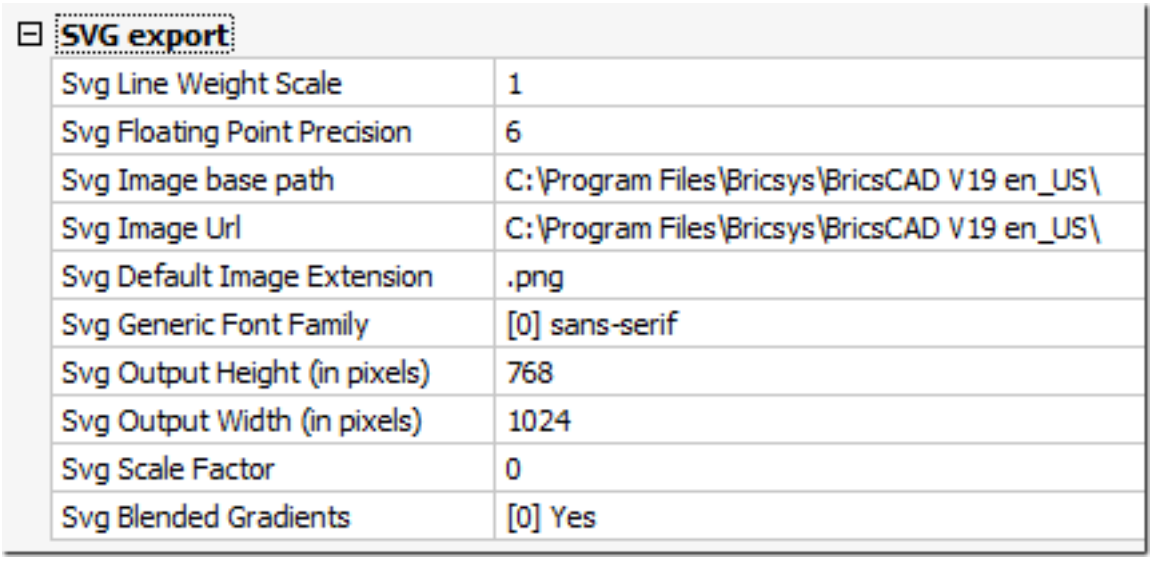
6.27.59 SvgOptions

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the SVG Export section of the Settings dialog box; short for "scalable vector graphics." **Note:** These options are used by the Export (6.13.15) command when you choose SVG as the file type. SVG files can be viewed and edited by the freeware Inkscape program, shown below: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/svgoptions-eg.png})

Accessing the Command

command bar: **svgoptions** transparent: **'svgoptions** : **SvgOptions** Displays the SVG Options section of the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/svgoptions-dlg.png})

Figure 6.2224



Settings Options

Option	Description
Svg Line Weight Scale	Specifies the scale factor at which to size of pixel widths of lines
Svg Floating Point Precision	Specifies the precision in terms of decimal places
Svg Image base path	Reports the path to where image files are kept; click Browse to choose another path
Svg Image Url	Specifies the prefix to append to images stored in SVG files
Svg Default Image Extension	Specifies the default extension for images
Svg Generic Font Family	Specifies the class of font to use; choose from: • **Sans-serif** - fonts without serifs, like Arial • **Serif** - fonts with serifs, like Times Roman • **Cursive** - fonts that look handwritten • **Fantasy** - unusual fonts • **Monospace** - fonts where each character takes up the same space (non-proportional spacing), such as Courier
Svg Output Height	Specifies the height of the SVG output file, in pixels
Svg Output Width	Specifies the width of the SVG output file, in pixels
Svg Scale Factor	Specifies the scale factor: • **0** - the current view is scaled to fit the page size specified by Svg Output Height and Width • Positive number - scales up the size of the output page
Svg Blended Gradients	Toggles the use of blended gradients in areas filled with gradients; choose from: • **Yes** - uses blended gradients; the file size may be larger • **No** - does not use blended gradients

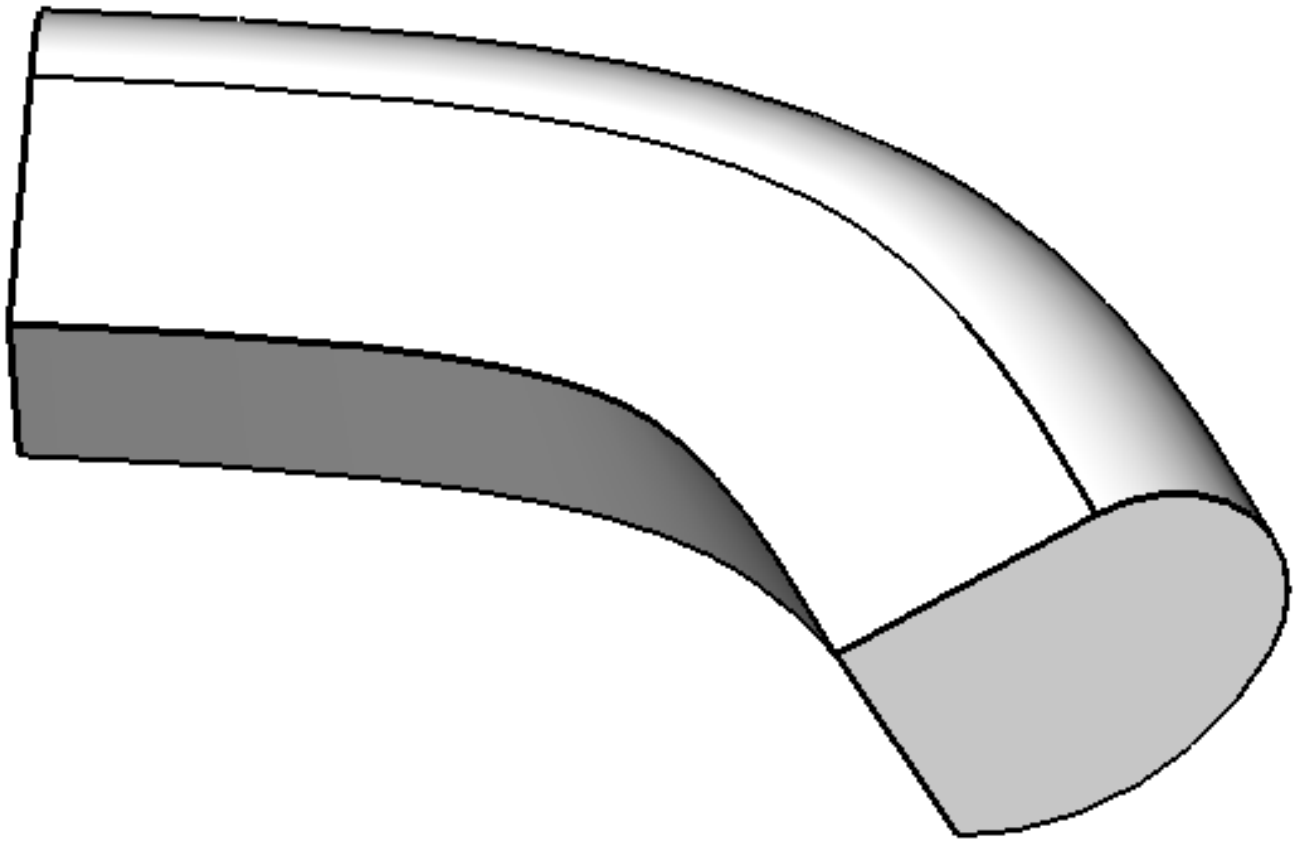
Related Commands

Export (6.13.15) - exports drawings in a variety of vector and raster formats.

Figure 6.2225



Figure 6.2226



6.27.60 Sweep

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Creates 3D solids or surfaces by sweeping profiles (open or closed 2D entities or 3D subentities) along an open or closed path. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands eg-1.png})

Accessing the Command

 command bar: **sweep** menu bar: **Model| 3D Solids | Sweep** ribbon: **Solid | Create | Sweep** toolbar: **3D Solids|**  **: Sweep Prompts you in the command bar**: Select profile entities or [MOde]: **(Select one or more profile* entities, or enter MO)** Entities in set: 1 Select profile entities or [MOde]: **(Select more profile entities, or else press Enter to continue)** Select sweep path or [Alignment/Base point/Twist]: **(Select a path** entity, or else enter an option)** The selected 2D entities are swept along the path. Depending on the value of the DELOBJ (1.18.1) (Delete object) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities. Delete defining entities? [Yes/No] <Yes>: **(Choose Yes or No)** * Entities accepted as profile entities: 2D splines, 3D splines, 2D polylines, 2D solids, 3D solid faces, 3D solid edges, arcs, circles, ellipses, elliptical arcs, lines, regions, surface and mesh edges, traces. ** Entities accepted as a path entity:2D splines, 3D splines,2D polylines,3D polylines, edges of 3D solids, surfaces or meshes, helices, arcs, circles, ellipses, elliptical arcs, lines.

Command Options

Option	Description
Select profile entities	Select one or more profile entities. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/sweep-2.png}) This command will take the profile entity and then sweep it along the path entity.
Mode	Sets the 3D entity type this command will create; prompts you: **Choose type of created entity [SOlid/SUrface] <SOlid>** - **(enter an option:)** • **Solid** - constructs 3D solids • **Surface** - constructs surfaces This option applies only to closed source entities; open entities always construct 3D surfaces.
Select sweep path	Selects an entity to define the path. This command will take the profile entity and sweep it along the path entity.
Alignment	Determines the angle at which the profile 2D entities are swept along the paths; see figures below. Prompts you: **Align sweep entity perpendicular to path before sweep? [Yes/No]** - **(enter an option:)** • **Yes** - aligns the sweep entity perpendicularly to the sweep path entity • **No** - the sweep entity maintains its original orientation, and so its not rotated along the path (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S At left, the alignment is on and so the circle is placed perpendicularly to the sweep path.At right, the alignment is off and so the circle is not rotated.
Base point	Defines the point on the sweep entity that follows the path; prompts you: **Specify base point** - **(pick a point in the drawing.)** This option allows offset sweep paths.
Twist	Specifies a twist angle for the swept entity or entities. The twist angle is the amount of rotation along the entire length of the sweep path.Prompts you: **Enter twist angle or [allow Banking for a non-planar sweep path (Bank)]** <0> - **(enter an option:)** • Type a value for the angle • **Bank**determines whether the swept entities will bank (rotate) naturally along a 3D sweep path,such as those made from a 3D polyline, spline, or helix

Grips Editing

Sweep solids can only be moved using grips.

Related Commands

dmExtrude (6.12.13) - creates a 3D solids or surfaces by extruding open or closed 2D entities or regions. dmRevolve (6.12.22) - creates 3D solids or surfaces by revolution of open or closed 2D entities or regions about an axis. dmThicken (6.12.31) - creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, faces of 3D solids and wire entities such as lines, arcs, polylines, ... Extrude (6.13.21) - extrudes 2D entities to 3D solids or surfaces. Loft (6.20.37) - creates 3D solids passing through two or more cross sections. Revolve (6.26.30) - revolves open or closed 2D entities to turn them into 3D solids or surfaces. RevSurf (6.26.32) - creates a 3D mesh surface by revolving a linear entity around a line.

Figure 6.2227



Figure 6.2228

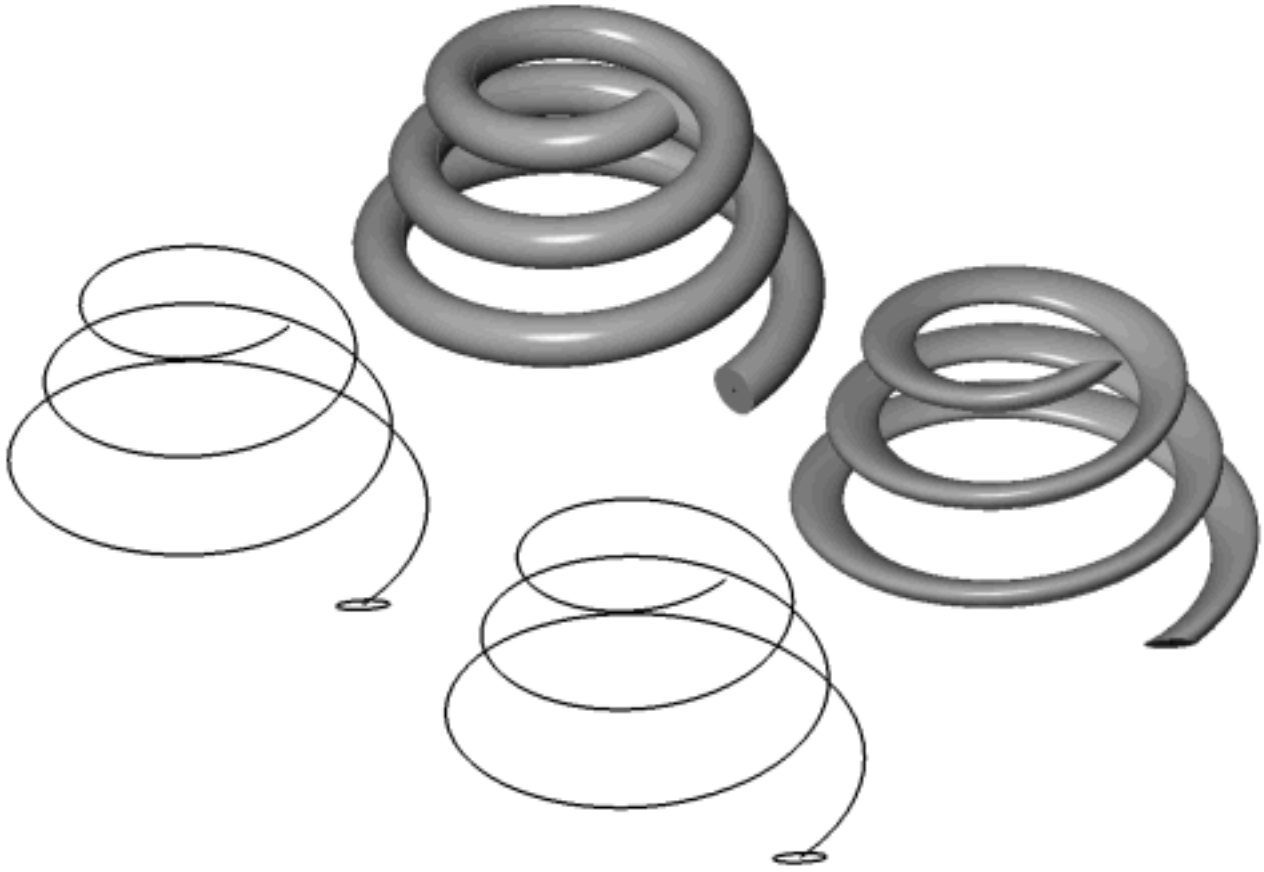
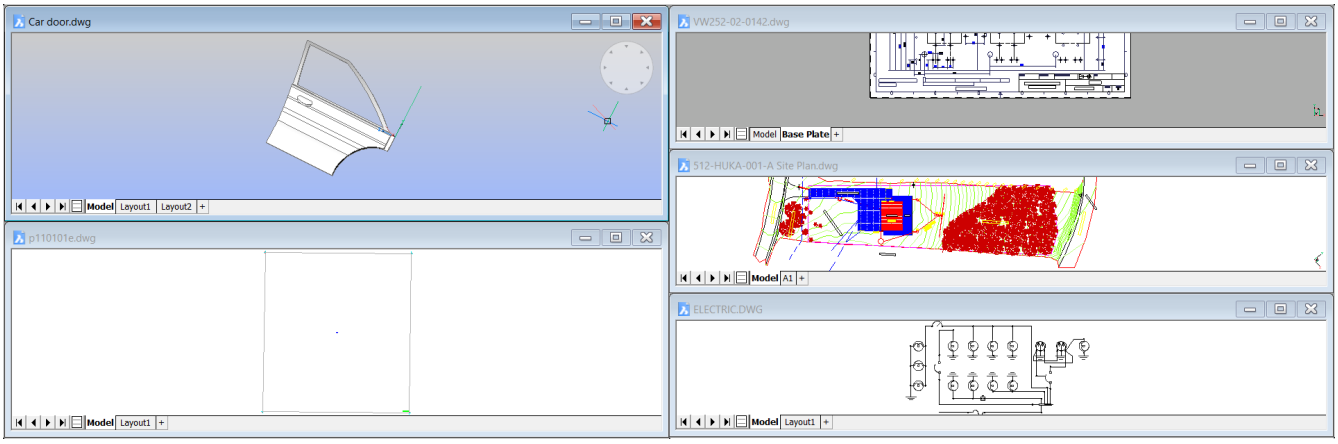


Figure 6.2229



Figure 6.2230



6.27.61 SysWindows

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Arranges the program's windows in tiles or cascaded formations (short for "system windows"). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/syswindows-eg.png})

Accessing the Command

command bar: **syswindows** menu bar: **Window | Cascade Window | Tile Vertically Window | Tile Horizontally : SysWindows Prompts you in the command bar:** Arrange MDI windows [Cascade/tile Horizontally/tile Vertically/Arrange icons] <Cascade>: **(Enter an option)**

Command Options

Option	Description
Cascade	Overlaps windows in a cascading arrangements; see the WCascade (6.33.3) command
Horizontally	Arranges windows horizontally; see the WhTile (6.33.9) command.
Vertically	Arranges windows vertically; see the WvTile (6.33.19) command.
Arrange icons	Arranges icons of minimized windows at the bottom of the graphical screen; see theWiArrange (6.33.10) command.

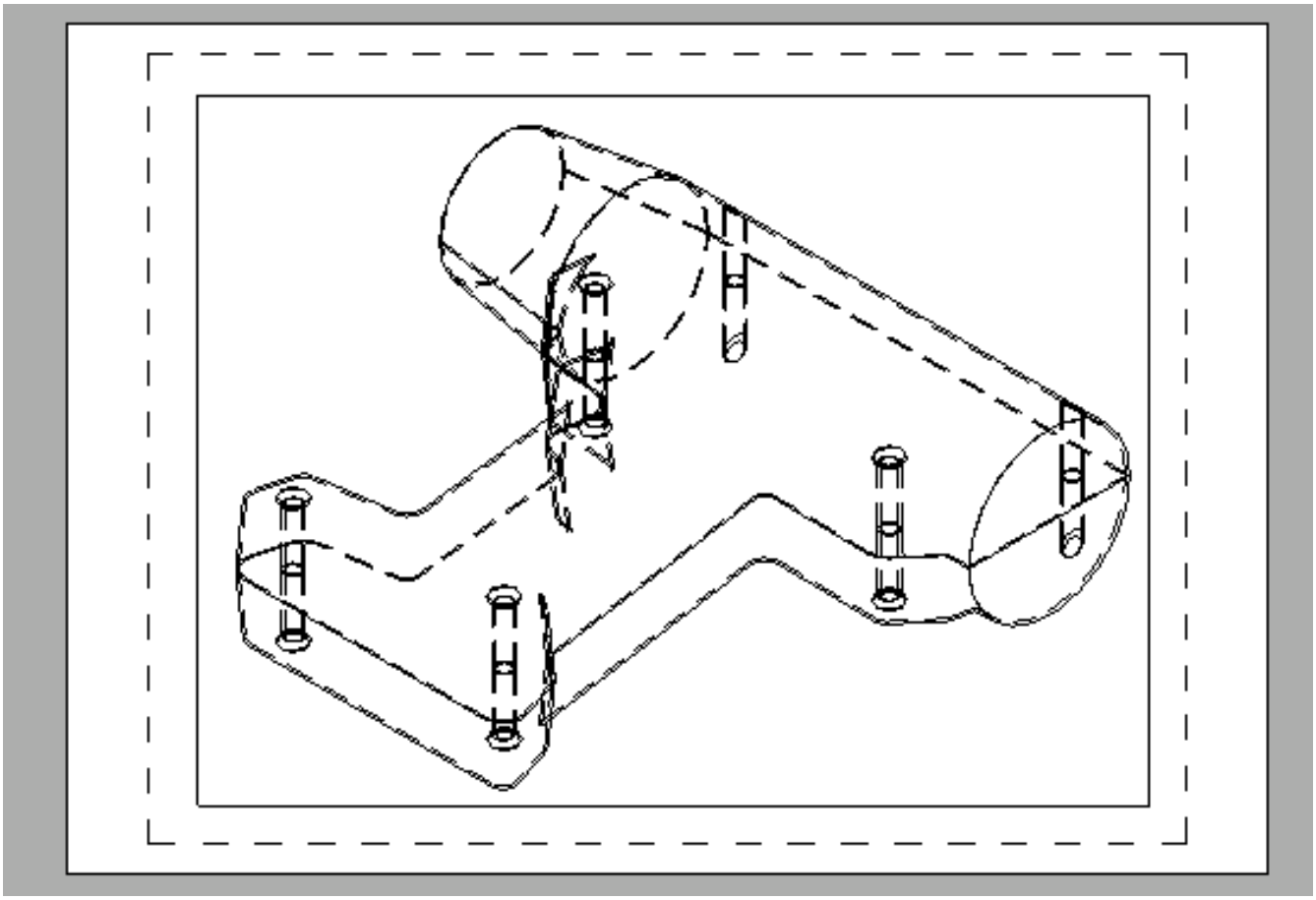
Related Commands

WCascade (6.33.3) - cascades the windows. WClose (6.33.4) - closes the current window. WCloseAll (6.33.5) - closes all windows. WhTile (6.33.9) - tiles windows horizontally. WiArrange (6.33.10) - arranges iconized windows. WvTile (6.33.19) - tiles windows vertically.

Figure 6.2231



Figure 6.2232



6.27.62 Solprof

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates hidden line representations of 3D solids in layout viewports (short for "solid profiles"); the representations are blocks. **Note:** This command operates only in model space of a layout tab viewport. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/solprof-5-eg.png})

Accessing the Command

 command bar: **solprof** menu bar: **Model | Solprof** ribbon:**Model | Sections | Section Plane | Solprof** toolbar: **Sections |**  : **Solprof** In a layout tab, double-click inside a viewport. **Prompts you in the command bar:** Select entities: **(Select one or more 3D solids) Select entities: (Select more 3D solids, or else press Enter to stop selecting entities)** Use separate layer for hidden lines? [Yes/No] <Yes>: **(Press Y or N)** Project results onto plane? [Yes/No] <Yes>: **(Press Y or N)** Hide tangential edges? [Yes/No] <Yes>: **(Press Y or N)**

Command Options

Option	Description

Select entities

Select one or more 3D solids; this command does not work with any other kind of 3D entity, or with 3D solids referenced externally (xrefs):

- Select one or more entities in the viewport
- Enter **all** to select all entities; the program filters out ineligible ones
- Press **Enter** to end entity selection

(This command operates only in the model space of a layout tab viewport.)

Use separate layer for hidden lines?

Determines if hidden lines are placed on a separate layer from the visible lines:

- **Yes** - creates two 2D blocks, one for the visible lines placed on layer PV-**<viewport handle>** (left, in the figure below)and another block for hidden lines on layer PH-**<viewport handle>** (at right) **(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/so 4-eg.png})**
- **No**- creates one block of both hidden and visible lines on layer PV-**<viewport handle>**

The blocks and layers are named by the program, such as block *U3 and layers PH-88 and PV-88:

- ***U** identifies blocks created by the program
- **PH-** is short for ”profile hidden”
- **PV-** is short for ”profile visible”

Project results onto plane?

Determines if the profiles are projected, which is code for, ”Are the resulting blocks 2D?

- **Yes** - profile lines are projected onto the viewing plane. The 3D solids (shown in yellow below) are projected onto a plane parallel with the layout’s viewport (i.e., your point of view), creating a block in 2D (shown by the black lines below). In the figure, the viewpoint has been rotated to show that the block(s) are indeed 2D.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/so 2-eg.png})

- **No** - no projection; the blocks are made of 3D entities, as shown below

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/so 3-eg.png})

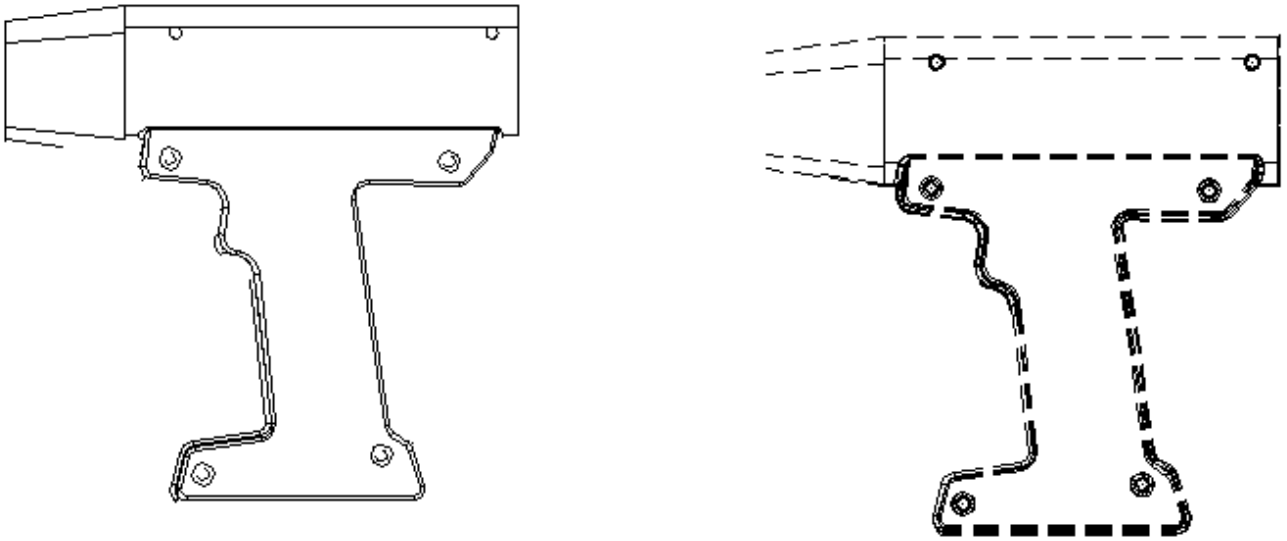
Hide tangential edges?

Toggles the display of tangential lines:

- **Yes** - hides tangential lines
- **No** - displays tangential lines

Tangential lines are used to show the transition between two faces that are curved. In a 3D shaded model, they are not needed as highlights and shadows indicate the transition; but in 2D wireframe mode it can be useful to show them. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

Figure 6.2233



Related Commands

Flatshot (6.14.9) - creates a hidden line representation of 3D solids in model space. Hide (6.16.10) - removes hidden lines from 3D entities. Shademode (6.27.29) - sets the shading style for 3D drawings. ViewExport (6.31.7) - exports the content of drafting views and sections obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing.

Figure 6.2234

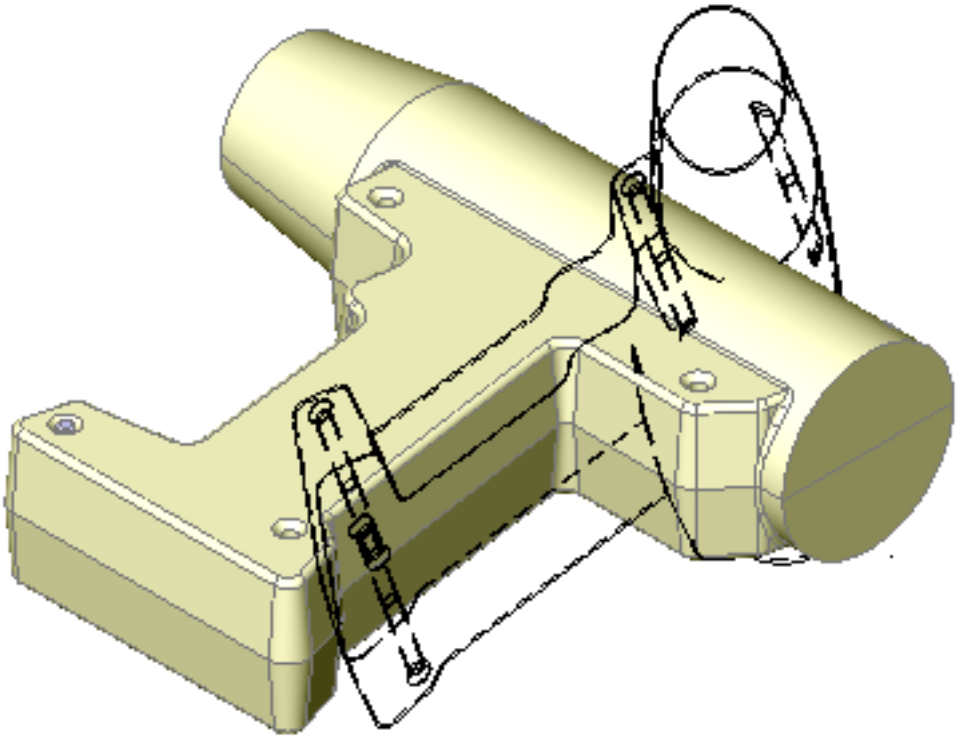


Figure 6.2235

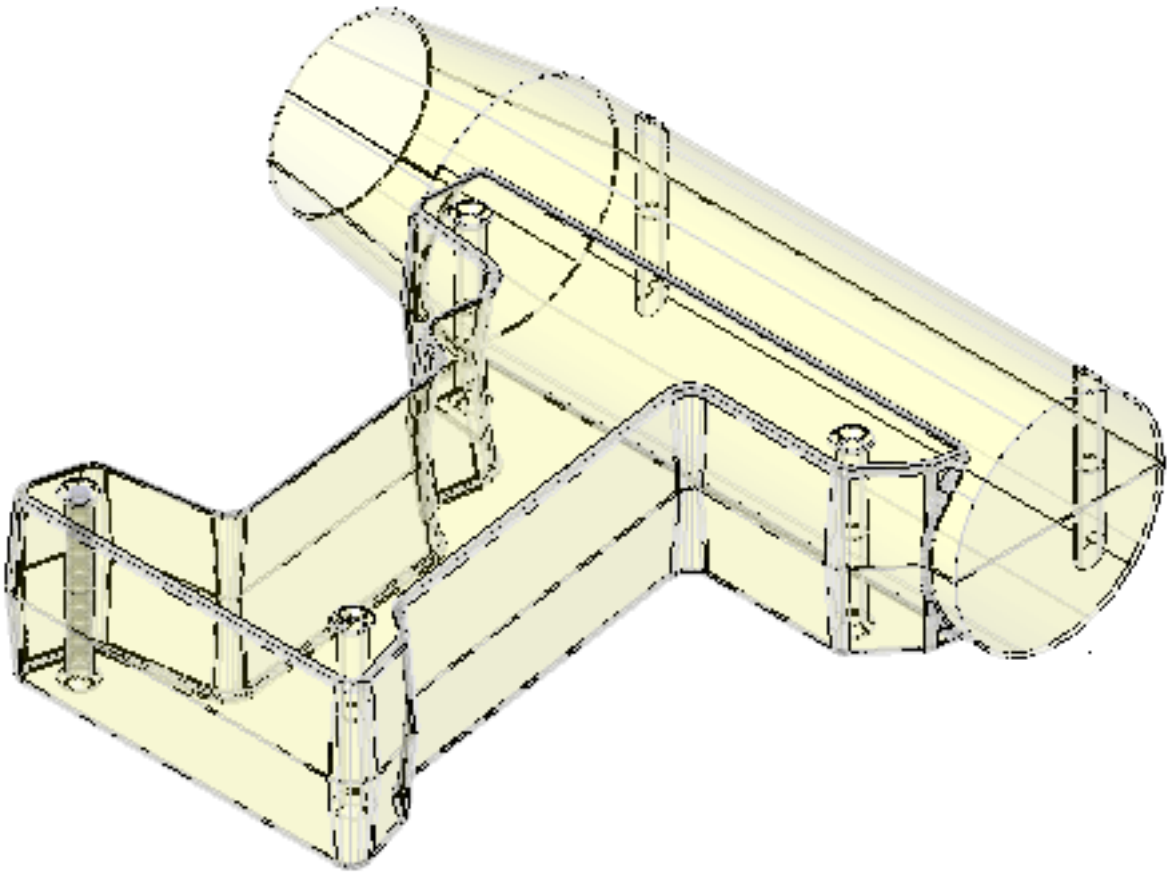


Figure 6.2236

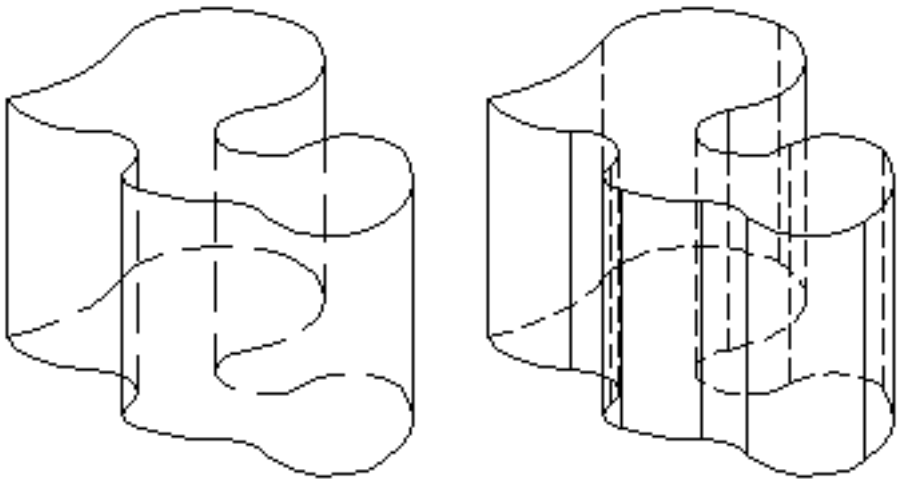


Figure 6.2237



6.27.63 Spell

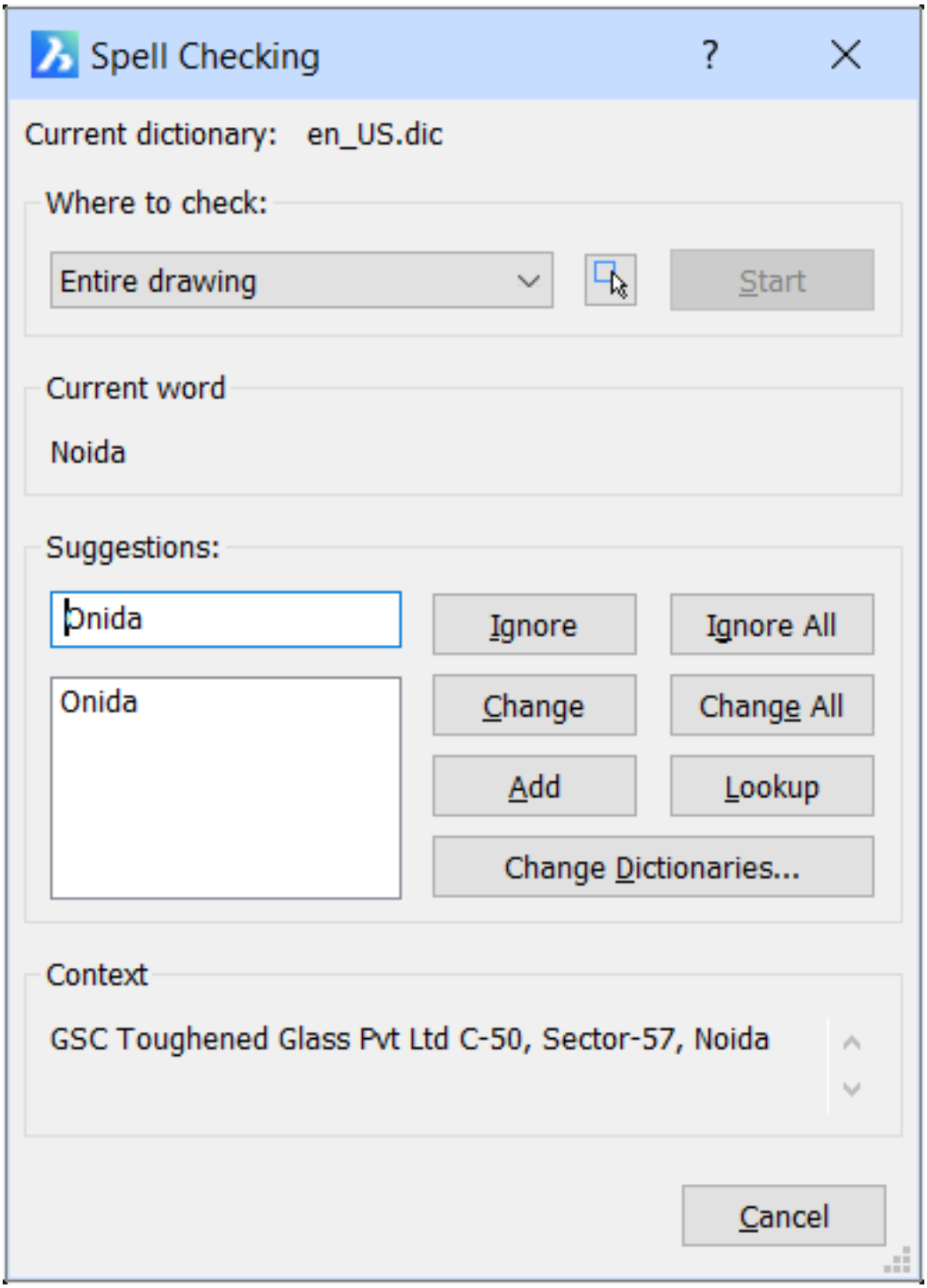
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Checks the spelling of words in drawings, includingfields in texts, mtexts, leaders, mleaders, tables, and block attributes.

Accessing the Command



command bar: **spell** transparent:'**spell** alias: **sp** menu bar: **Tools | Check Spelling : Spell** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/spell-dlg.png}) **Click Start.**

Figure 6.2238



Dialog Box Options

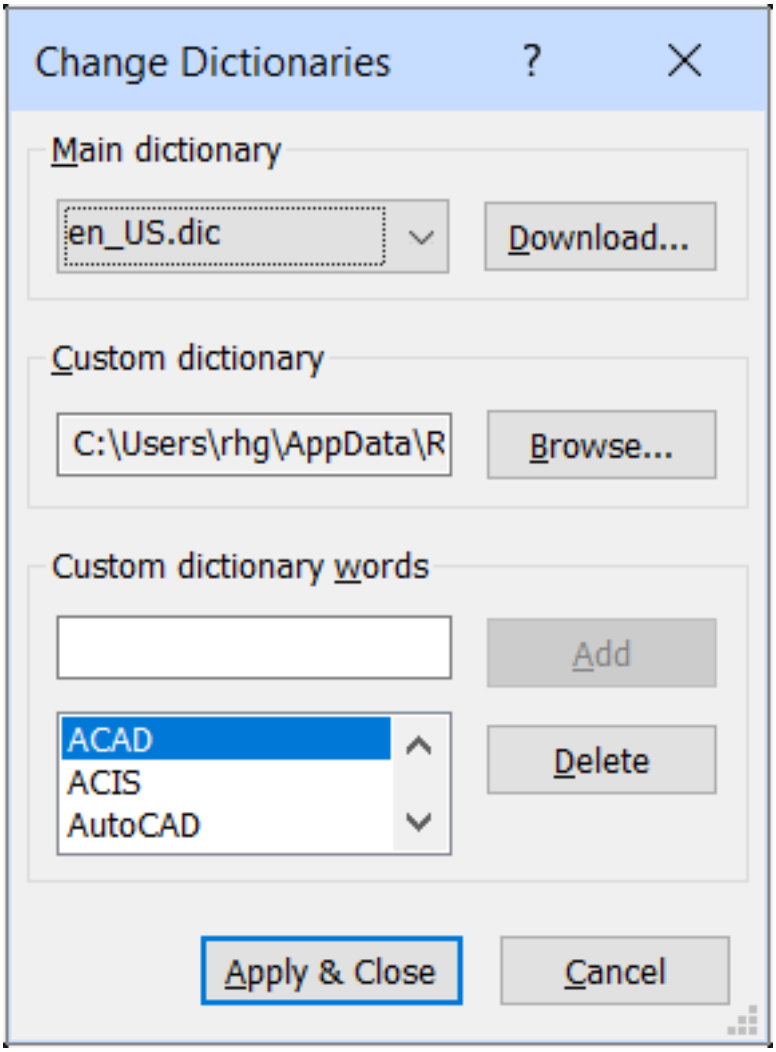
Option	Description
Where to check	Checks text in: • **Entire drawing** - checks all text in the drawing, except for text on frozen layers • **Selected entities** - checks selected text; click the Select Entities button to choose text
 Select entities	Selects the text to be checked; prompts you: **Select entities** - **(choose one or more pieces of text; press Enter to return to the dialog box.)**
Start	Starts checking the spelling of words.
Suggestions	Lists the suggested replacements for misspelled words.Choose a replacement word, and then click **Change** or **Change All**.
Ignore	Ignores the spelling of the word.
Ignore All	Ignores all instances of the word.
Change	Changes the word to the one selected from the Suggestions list.
Change All	Changes all instances of the word to the one selected from the Suggestions list.
Add	Adds the word to the word list.
Lookup	Provides additional alternative spellings for the word selected in the Suggestions list.
Change Dictionaries	Changes the dictionary used for spell checking; displays the Change Dictionaries dialog box.

Change Dictionaries Dialog Box Options

Click **Change Dictionaries** to arrive at this dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/C
changedictionaries-dlg.png})

Change Dictionaries	Description
Main Dictionary	Specifies the dictionary to use for spell checking. To change dictionaries: 1. Click the droplist 2. Choose another dictionary
Download	Accesses additional dictionaries from the OpenOffice.org Wiki. Opens the Web browser at **https://extensions.openoffice.org/dictionaries**
Custom dictionary	Specifies the name of a custom dictionary. Custom dictionaries are used for discipline-specific words, such as medical or mechanical.
Browse	Selects a CUS custom dictionary file. Opens the Select Custom Dictionary dialog box: 1. Choose a CUS file 2. Click **Open**.
Custom dictionary words	Enters words to be stored in the custom dictionary.
Add	Adds the word to the custom dictionary. Words are also added when Add is clicked on the Spell Checking dialog box.
Delete	Removes the word from the custom dictionary.

Figure 6.2239



Procedures

- To check the spelling (1.21.24)
- To change dictionaries (1.21.24)
- To add a new main dictionary (1.21.24)
- To add a new custom dictionary (1.21.24)

Related Commands

Text (1.21.20) - places single-lines of text in drawings. -Text (6.29.12) - places text and evaluates LISP expressions. MText (6.21.35) and -MText (6.21.33) - places paragraph text. AttDef (6.4.27) - defines attribute text. DdEdit (6.10.14) - edits text. Find (6.14.8) - finds and replaces text. Properties (6.24.49) - modifies the properties of text.

Figure 6.2240



6.27.64 Sphere

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws 3D spheres:

- Classic: sphere is drawn from a 3D polyface mesh.
- Shape, Pro and Platinum: sphere is drawn as a 3D solid.

Accessing the Command

 command bar: **sphere** menu bar: **Model | 3D Solids | Sphere** ribbon:**Solid | Create | Primitive | Sphere** toolbar: **3D Solids |**  **: Sphere Prompts you in the command bar:** Center of sphere: **(Pick a point, or enter coordinates)** Set radius of sphere or [Diameter]: **(Specify a distance, or press D)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco IfDynamic Dimensions (1.15.9)are active, you can enter the radius in the dynamic dimension entry field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SphereDynDim.png})

Figure 6.2241

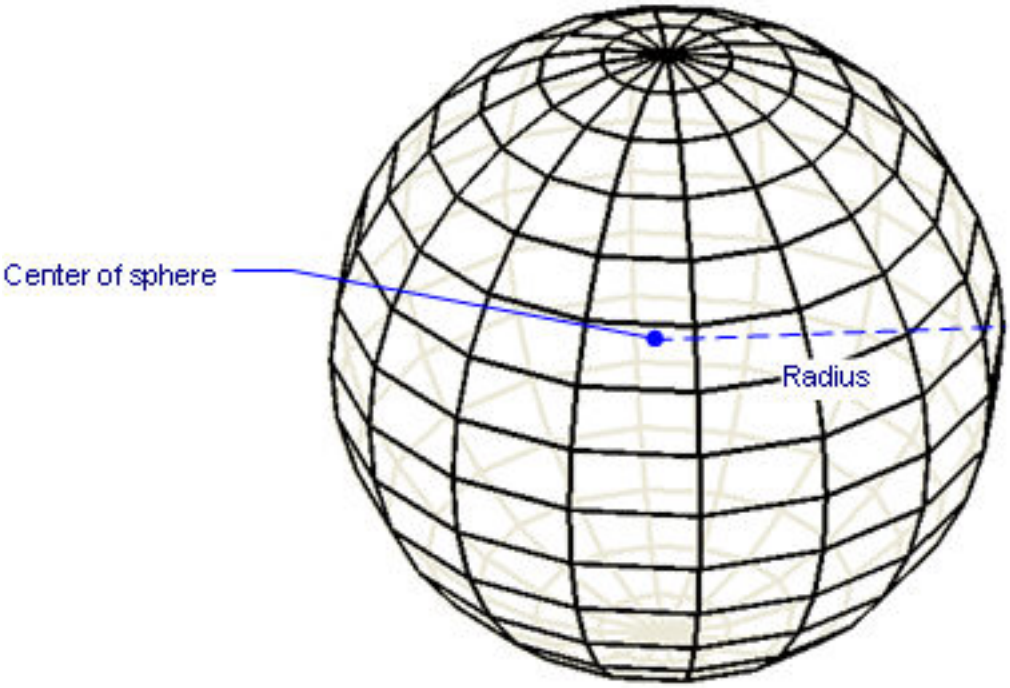
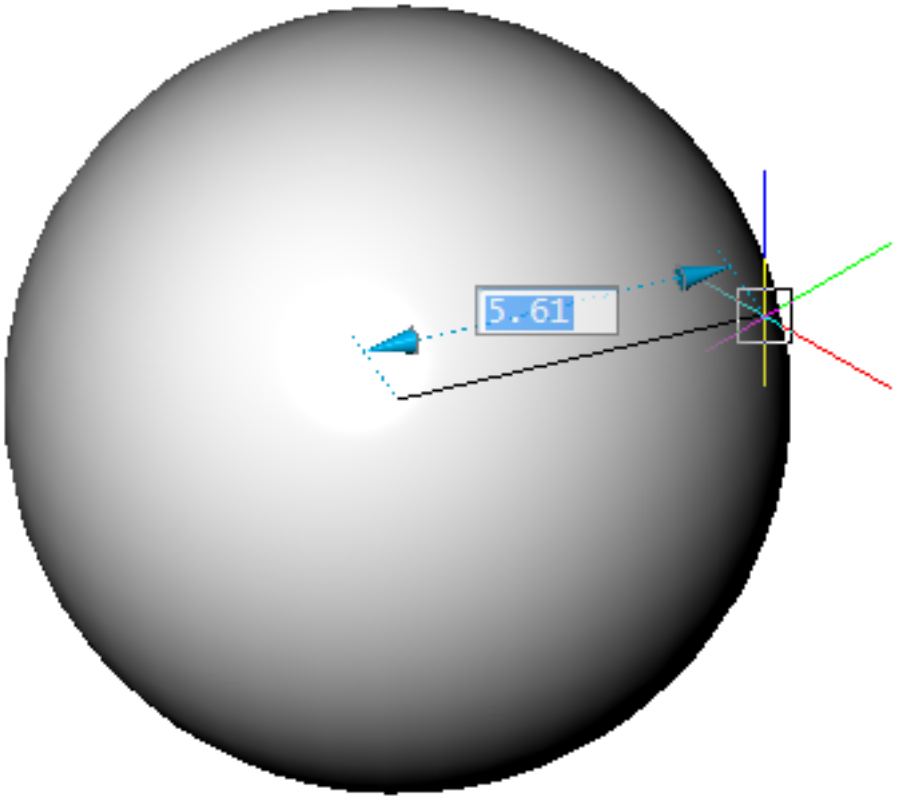


Figure 6.2242



Command Options

Option	Description
Center of sphere	Specifies the center point of the sphere; enter x,y- or x,y,z-coordinates, or pick a point in the drawing.
Radius of sphere	Specifies the radius.
Diameter	Specifies the diameter.

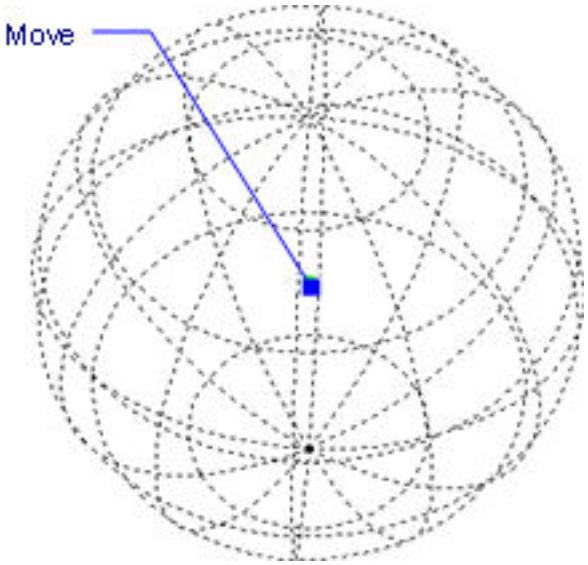
Grips Editing

Spheres can be edited directly through grips:

1. Select the sphere. Notice that it has one grip.
2. Drag the grip to move the sphere.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image441.jpg})

Figure 6.2243



Related Commands

Box (6.6.17) - creates 3D boxes as solid models. Cone (6.9.63) - creates 3D cones as solid models. Cylinder (6.9.60) - creates 3D cylinders as solid models. Dish (6.10.30) - creates the lower half of 3D balls as solid models. Dome (6.10.34) - creates the upper half of 3D balls as solid models. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Pyramid (6.24.58) - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic) Revolve (6.26.30) - revolves 2D entities into 3D solid models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. **Subtract (6.27.56)** - removes one solid model from a second. Torus (6.29.22) - creates 3D donuts as solid models. Union (6.30.6) - joins two or more solid models into a single entity. Wedge (6.33.7) - creates 3D wedges as solid models.

Figure 6.2244



Figure 6.2245



6.27.65 Spline

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws open and closed splines in 2D and 3D. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyeg.png})

Accessing the Command

 command bar: **spline** alias: **spl** menu bar: **Draw | Spline** toolbar: **Draw |**  **: Spline Prompts you in the command bar:** First point for spline: **(Pick a point, or enter x,y,z coordinates)** Second point: **(Pick another point)** Set next point or [Close/Fit tolerance/Undo]: **(Pick another point, or enter an option)** Set next point or [Close/Fit tolerance/Undo]: **(Pick another point, enter an option, or press Enter to end drawing the spline and move on to specifying the tangent points)** Select starting tangent point: **(Pick a point, or press Enter for no starting tangent)** Enter tangent for ending point: **(Pick a point, or press Enter for no ending tangent)**

Command Options

Option	Description
First point for spline	Specifies the start of the tangent: • Pick a point • Enter x,y,z coordinates
Second point	Specifies the first vertex of the spline: • Pick another point • Enter x,y,z coordinates
Next point	Specifies the next vertex. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i This prompt repeats until you press **Enter**.

Close	Closes the spline; the end and start points are joined. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i To open or close an existing spline, follow these steps: 1. Select the spline 2. Change the **Close** property in the Properties (6.24.49) panel
Undo	Undraws the last portion drawn.
Fit tolerance	Specifies how closely the spline matches its vertices (also known as "fit points"); prompts you: **Enter fit tolerance (in drawing units)**- **(enter an value:)** • **0** - spline passes through fit points • More than 0 - spline passes further away from fit points
Select starting tangent point	Locates a tangency for the starting point; press **Enter** to ignore this option
Enter tangent for ending point	Locates a tangency for the ending point; press **Enter** to ignore this option

Grips Editing

Open and closed splines can be edited directly through grips. Splines can be grip edited either with fit points or control vertices. Choose either **Fit points** or **Control vertices** from the **Method** property options of a spline in the Properties Panel (6.24.49). The **CV Frame** property of the spline controls whether the control vertices frame displays or not. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Image4801.png})
Method = Fit points (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Image4802.png}
Method = Control Vertices; CV Frame = Off (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sc
Method = Control Vertices; CV Frame = ON

1. Select the spline. Notice that it has one grip at each fit point or control vertex.
2. Drag a grip to stretch the spline.

Figure 6.2246



Figure 6.2247

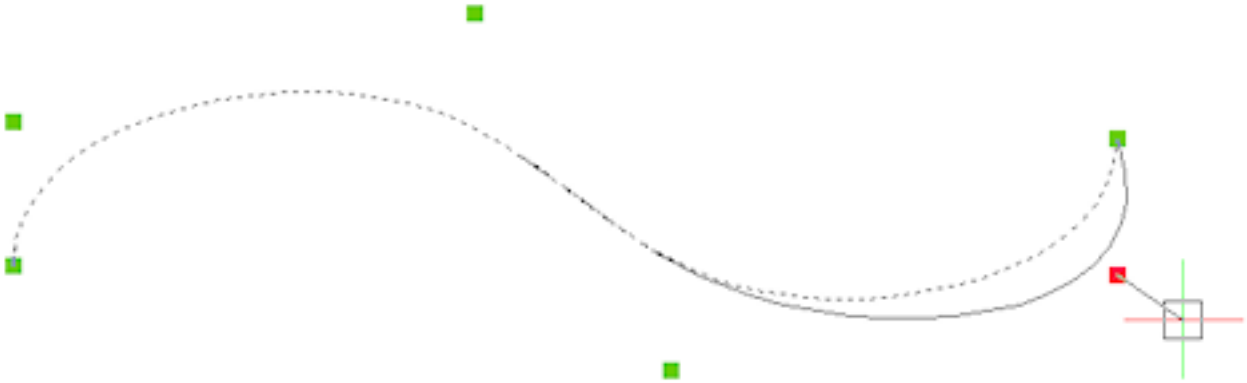
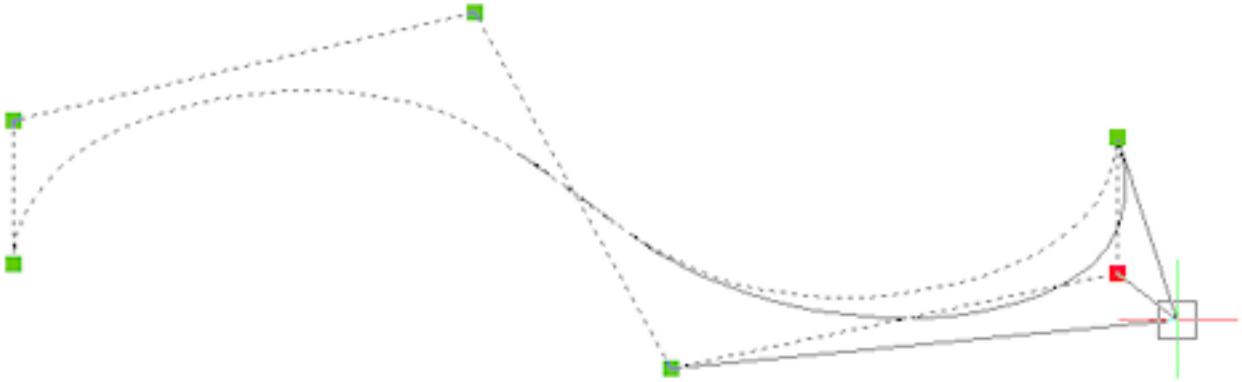


Figure 6.2248



Procedures

- Drawing splines (1.17.13)

Related Command

PEdit (6.24.20) - turns polylines into splines. Splinedit (6.27.40) - modifies the properties of a spline or converses a spline to a polyline. Helix (6.16.7) - Creates a 2D spiral or a 3D helix.

Figure 6.2249

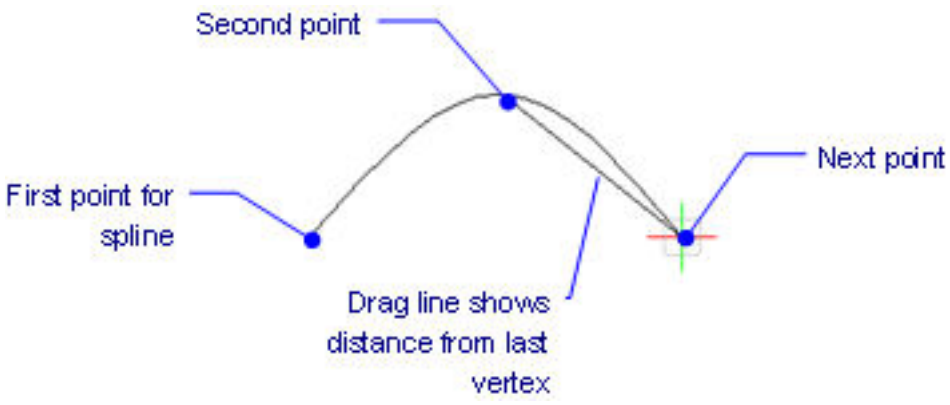
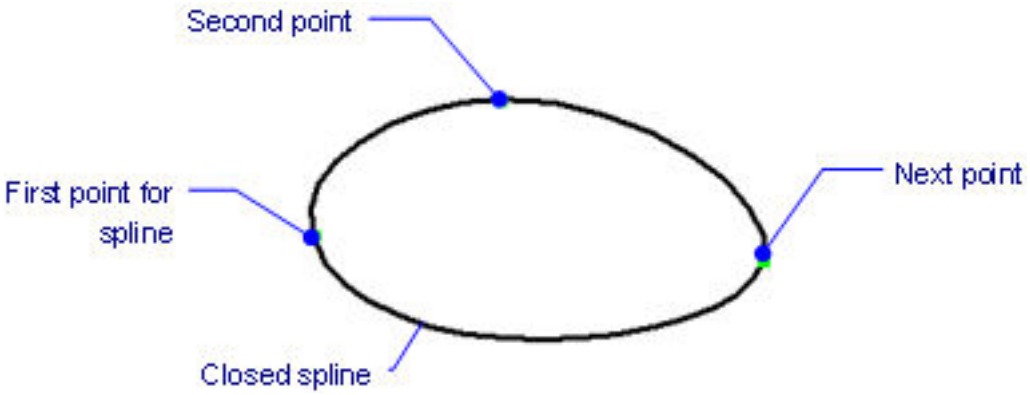


Figure 6.2250



6.28 Sm (Sheet Metal)

6.28.1 SmAssemblyExport

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Traverses the Assembly Structure of the current drawing, searches for 3D solids in its component, and applies the SmConvert (6.28.4) and SmExport2d (6.28.7) commands. The resulting .dxf files with unfolding information are saved in a user defined output folder and sorted by thickness. Parts which were processed with command warnings or errors are put to special folders. An HTML report listing all the solids in the assembly is generated, indicating their status and links to .dwg and .dxf files. Each solid is classified:

- ”Sheet Metal” – solid is a sheet metal part;
- ”Poor Sheet Metal” – solid looks like a sheet metal design, user assistance, and rework are needed;
- ”Not Sheet Metal” – solid is not recognized as sheet metal design.

The command allows to quickly sort out non-sheet metal parts, therefore it can be applied to mixed assemblies containing both sheet metal and non-sheet metal parts. The command can be applied to simple drawings with solids, to assemblies created in BricsCAD and to drawings imported by Communicator (4.1.1). It is recommended to set the ImportProductStructure (1.13.11) Communicator user preference to 2 (= As mechanical components), else mechanical components will be exploded to solids, which increases the number of entities being processed by SmAssemblyExport, resulting in an increased processing time and multiple repeats in the output. In order to improve performance the Visual Style (6.31.13) of the active viewport is best set to **2dWireframe**.

Figure 6.2251



Accessing the Command



command bar: **smassemblyexport** menu bar: **Sheet Metal | Unfold | Export Assembly to DXF** toolbar: **Sheet Metal | Export Assembly to DXF** ribbon: **Sheet Metal | Flatten | Export Assembly to DXF** : **SmAssemblyExport Prompts you in the command bar:** Select solids, component instances or [Output folder/bend Table/Dxf version] <Entire model>: **(Select the to be processed or press Enter to include all the entities in assembly. Note that a component will be skipped only if it is not referred by any instance.)** Unless defined differently in the command options, the results are saved in the folder of the processed drawing.

Command Options

Option	Description
Set Output Folder	Allows to set the output folder: Displays a File dialog to set the output folder.
Set Bend Table	Allows to assign a bend table Displays a File dialog to set the bend table to be used during SmExport2d (6.28.7) calls in the output process.
DXF version	Allows to specify the DXF version: Select DXF version [R11/R12/R13/LT2/R14/LT95/2000/2004/2007/2010/2013/2018] <2013> - choose a DXF versi

Related Commands

SmConvert (6.28.4) - automatically recognizes flanges and bends in a 3D solid. SmExport2d (6.28.7) - exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.

Figure 6.2252



6.28.2 SmBendCreate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Converts hard edges (sharp edges between flange faces) or junctions into bends.

Accessing the Command

 command bar: **smbendcreate** menu bar: **Sheet Metal | Bend | Create Bend** toolbar: **Sheet Metal |Create Bend** ribbon: **Sheet Metal | Modify |Create Bend** Quad cursor menu: **Sheet Metal |Create Bend : SmBendCreate Prompts you in the command bar:** Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>: **(Select a face or a hard edge or 3D solid or press Enter to include all sheet metal parts)** Entities in set: 1 Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>: **(Select another face or hard edge or 3D solid or press Enter)*** * All selected items must be of the same kind: faces or hard edges.

Related Commands

SmBendSwitch (6.28.3) - converts bends to lofted bends. SmConvert (6.28.4) - automatically recognizes flanges and bends in a 3D solid. SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmDissolve (6.28.6) - removes sheet metal data from the selected faces. SmFlangeConnect (6.28.12) - closes gaps between two arbitrarily oriented flanges. SmJunctionCreate (6.28.18) - converts hard edges (sharp edges between flange faces) and bends into junctions. SmReliefCreate (6.28.22) - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge .

Figure 6.2253



6.28.3 SmBendSwitch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Converts bends to lofted bends.

Accessing the Command

 command bar: **smbendswitch** menu bar: **Sheet Metal | Bend | Switch Bend** toolbar: **Sheet Metal | Switch Bend** ribbon: **Sheet Metal | Modify | Switch Bend** Quad cursor menu: **Sheet Metal | Switch Bend : SmBendSwitch Prompts you in the command bar:** Select bend faces/<Entire model>: **(Select a bend or press Enter to select all bends in the model.)** Entities in set: 1 Select bend faces/<Entire model>: **(Select a bend or press Enter to convert the selected bends.)**

Command Options

No options.

Related Commands

SmBendCreate (6.28.2) - converts hard edges (sharp edges between flange faces) into bends SmLoft (6.28.20) - creates sheet metal part with lofted bends and flanges from two non-coplanar curves. SmRepair (6.28.24) - restores the 3D solid model of a sheet metal part by thickening one of its sides: all thickness faces become perpendicular to flange faces. Fixes problems specific to parts with lofted bend features: merges adjacent bends and provides tangential connection with flanges.

Figure 6.2254



6.28.4 SmConvert

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Automatically recognizes flanges, bends, lofted bends, ribs, form features and holes for selected 3D solids.

Accessing the Command



command bar: **smconvert** menu bar: **Sheet Metal | Convert to Sheet Metal** toolbar: **Sheet Metal | Convert to Sheet Metal** ribbon: **Sheet Metal | Create | Convert to Sheet Metal** Quad cursor menu: **Sheet Metal | Convert to Sheet Metal : SmConvert Prompts you in the command bar:** Select 3D solids or [Entire model] <Entire model>: **(Select a 3D solid or press Enter to include the entire model.)** Entities in set: 1 Select 3D solids or [Entire model] <Entire model>: **(Select more 3D solids or right click.)**

Command Options

Option	Description
Entire Model	All solids in the entire model are analyzed and, if possible, converted to Sheet Metal bodies. The command recognizes sheet metal features (such as flanges and bends) in the input geometry. The body thickness of the sheet metal drawing is the criterium in case multiple solids are selected. Therefore selected bodies must have the same thickness, else an error message displays in the command window: "Could not convert the 3D solid to sheet metal. Recognized thickness is not equal the thickness of document: 5.000000 != 7.000000."
Settings	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee • Prefer form features: Controls preferable way to recognize ambigious geometry - as single form feature or as a set of flanges and bends. • Recognize holes: If enabled, holes will be recognized as features. • Recognize rib control curves: Controls recognition of control curve for rib feature. • Deviation value of wrong feature thickness: Determines allowed deviation between model thickness and thickness of given wrong feature. • Type of deviation of wrong feature thickness: Deviationcan be treated as absolute value or ratio to model thickness.

Related Commands

SmAssemblyExport (6.28.1) - analyzes the assembly structure of the current drawing, executes the SmConvert (6.28.4) and SmExport2d (6.28.7) commands, and saves the resulting .dxf files with unfolding information in an output folder. SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. SmDissolve (6.28.6) - removes sheet metal data from the selected faces. SmFlangeBase (6.28.10) - creates a base (initial) flange of a sheet metal part from a closed planar profile. SmForm (6.28.16) - converts a selected set of faces to a form feature. SmParametrize (6.28.21) - creates a consistent set of 3D constraints for a selected sheet metal part. SmRepair (6.28.24) - restores the 3D solid model of a sheet metal part by t[illegible] one of its sides (all thickness faces become perpendicular to flange faces).

Figure 6.2255

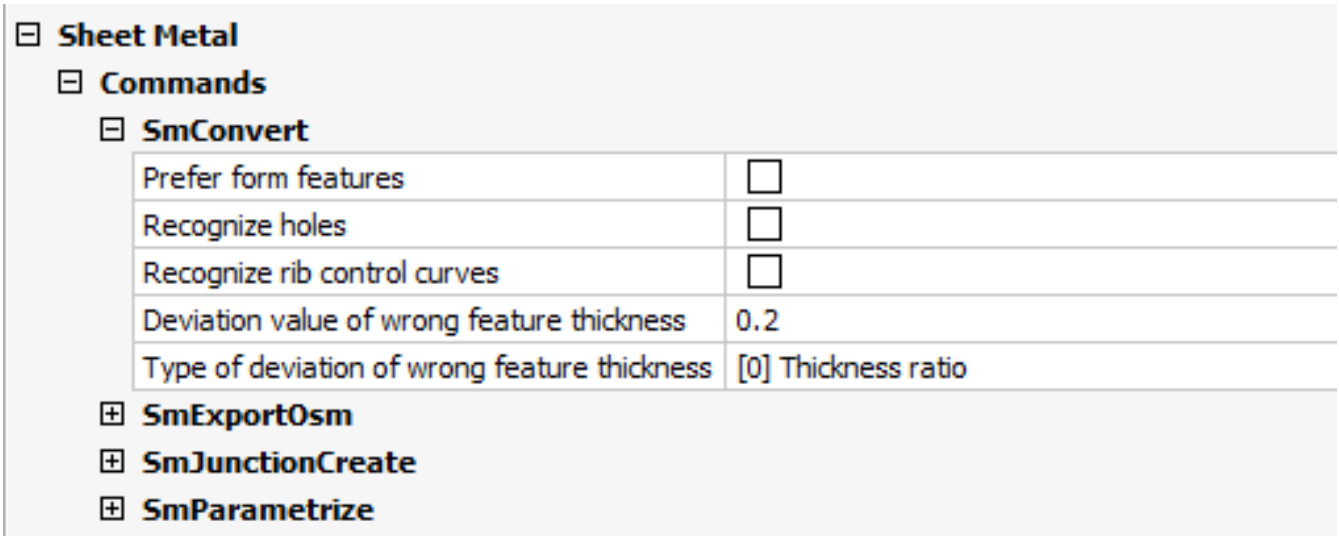


Figure 6.2256



6.28.5 SmDelete

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature.

Accessing the Command

 command bar: **smdelete** menu bar: **Sheet Metal | Delete Feature** toolbar: **Sheet Metal | Delete Feature** ribbon: **Sheet Metal | Modify | Delete Feature** Quad cursor menu: **Sheet Metal | Delete Feature : SmDelete Prompts you in the command bar:** Select feature faces or thickness faces: **(Select an item.)** Entities in set: 1 Select feature faces or thickness faces: **(Select another item or right click.)** Example of bend feature deletion:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/staticbricsys1.png)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee
Selected bend faces	The selected bends are replaced by hard edges

Example of deletion by using thickness faces without junction feature:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/staticbricsys2.png)	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee
Selected thickness faces	The selected faces are replaced by hard edge

Command Options

No options.

Related Commands

SmBendCreate (6.28.2) - converts hard edges (sharp edges between flange faces) and junctions into bends. SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmDissolve (6.28.6) - removes sheet metal data from the selected faces or entities. SmExtrude (6.28.9) - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline. SmFlangeConnect (6.28.12) - closes gaps between two arbitrarily oriented flanges. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange. SmSplit (6.28.28) - splits a flange along a polyline drawn on its face. SmJunctionCreate (6.28.18) - converts hard edges (sharp edges between flange faces) and bends into junctions. SmReliefCreate (6.28.22) - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.

Figure 6.2257

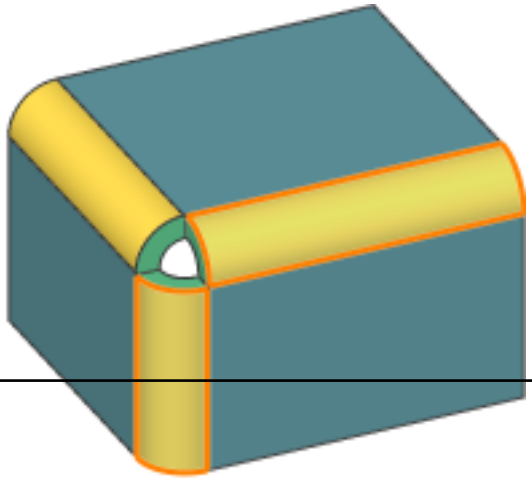


Figure 6.2258

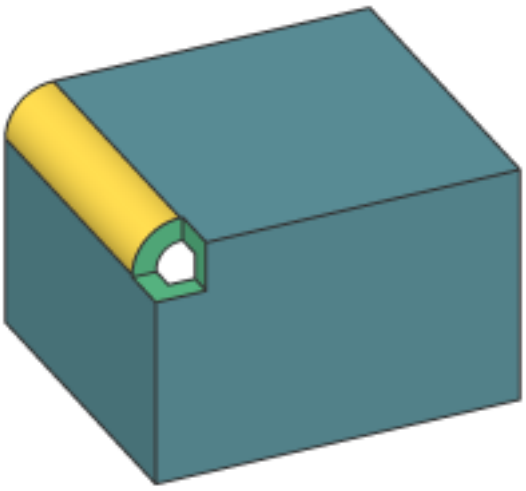


Figure 6.2259

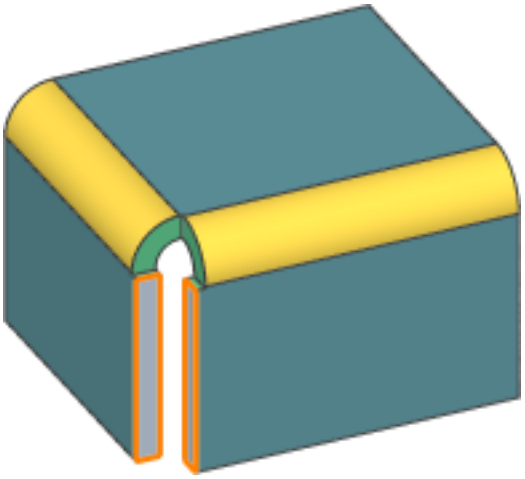


Figure 6.2260

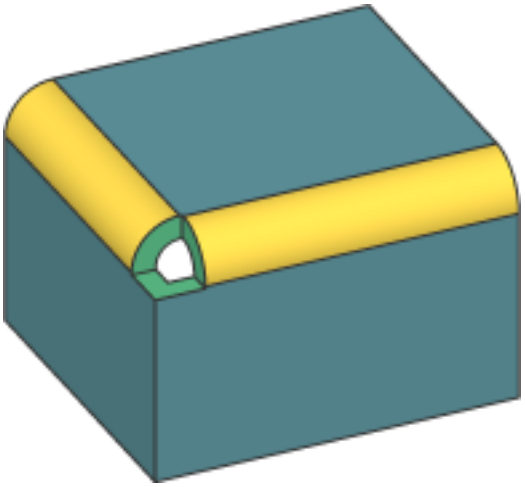


Figure 6.2261



6.28.6 SmDissolve

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Removes sheet metal data from the selected features. The command can dissolve sheet metal features on 3D solid faces. Dissolved features are no longer listed in the Mechanical Browser (3.2.2). Faces of dissolved features can be moved, rotated or push-pulled freely.

Accessing the Command


command bar: **smdissolve** menu bar: **Sheet Metal | Dissolve** toolbar: **Sheet Metal | Dissolve** ribbon: **Sheet Metal | Modify | Dissolve** Quad cursor menu: **Sheet Metal | Dissolve : SmDissolve Prompts you in the command bar:** Select faces on models or [Entire model] <Entire model>: **(Select a face, or entities, or right click to select the entire model.)** Entities in set: 1 Select faces on models: **(Select a face or right click to stop.)**

Command Options

Option	Description
Entire model	Selects all entities in the model.

Related Commands

SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmExtrude (6.28.9) - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline. SmFlangeBase (6.28.10) - creates a base (initial) flange of a sheet metal part from a closed planar profile. SmForm (6.28.16) - converts a selected set of faces to a form feature. SmLoft (6.28.20) - creates sheet metal part with lofted bends and flanges from two non-coplanar curves.

Figure 6.2262



6.28.7 SmExport2d

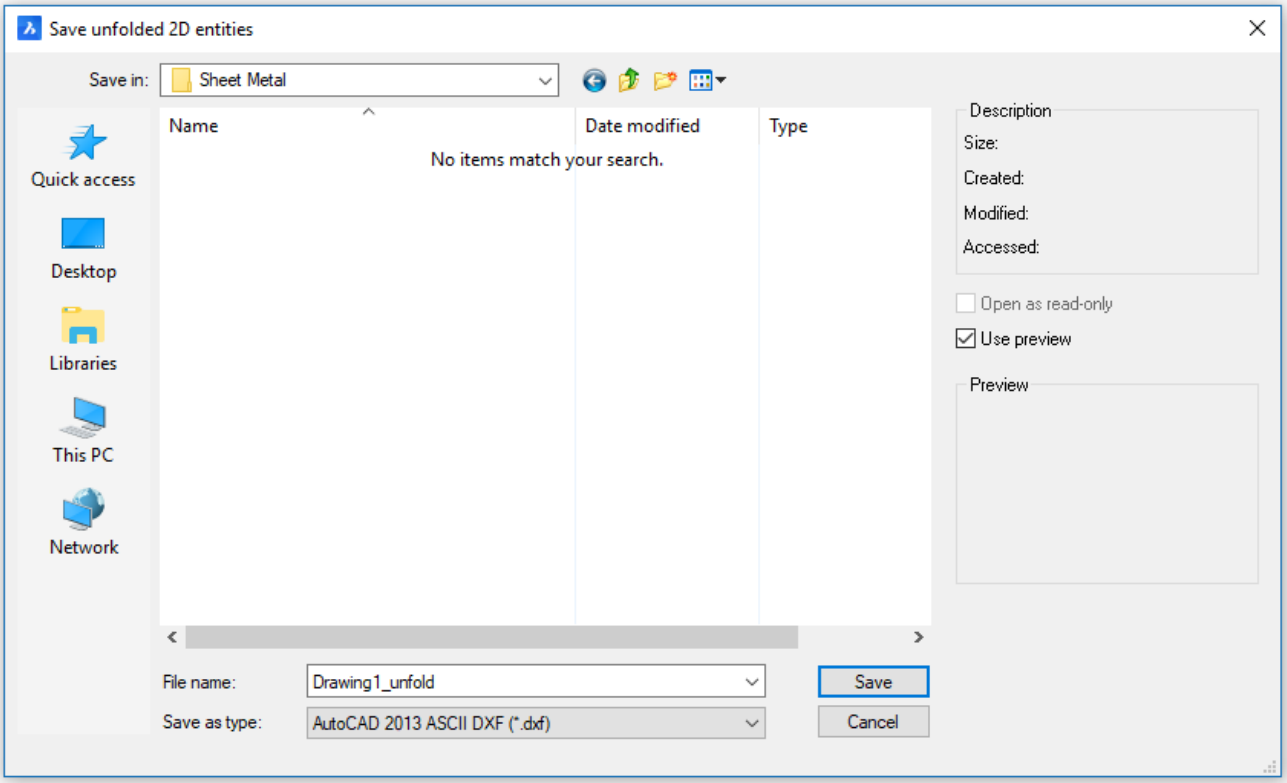
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png})
Exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.

Accessing the Command



command bar: **smexport2d** menu bar: **Sheet Metal | Unfold | Export to DXF** toolbar: **Sheet Metal | Export to DXF** ribbon: **Sheet Metal | Flatten | Export to DXF** Quad cursor menu:**Sheet Metal | Export to DXF : SmeExport2d**
Prompts you in the command bar: Select a flange or lofted bend face to start exporting: **(Identify the start face.)**
Displays a file dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/sm
Select the file format: dwg or dxf.

Figure 6.2263



Command Options

No options.

Related Commands

SmAssemblyExport (6.28.1) - analyzes the assembly structure of the current drawing, executes the SmConvert (6.28.4) and SmExport2d (6.28.7) commands, and saves the resulting .dxf files with unfolding information in an output folder. SmExportOSM (6.28.8) - exports a sheet metal solid to the .osm (Open Sheet Metal) file format (native for CADMAN-B CAM system). SmUnfold (6.28.29) - creates an unfolded 2D or 3D representation of a sheet metal part.

Figure 6.2264



6.28.8 SmExportOSM

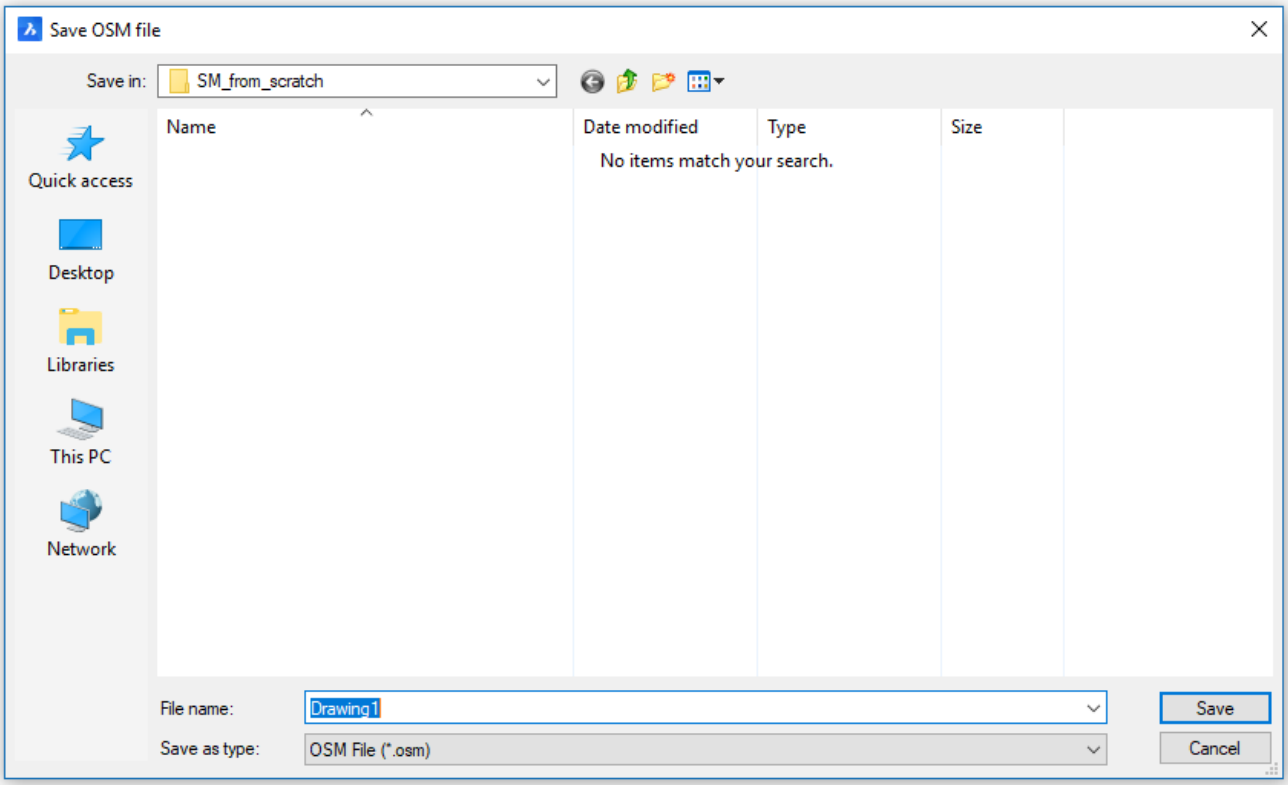
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Exports a sheet metal solid to the .osm (Open Sheet Metal) file format (native for CADMAN-B CAM system).The .osm part is saved as mm- or inch-based, depending on the value of the document MEASUREMENT (1.13.11) system variable. Document INSUNITS (1.13.11) are taken into account for proper scaling.

Accessing the Command



command bar: **smexportosm** menu bar: **Sheet Metal | Unfold | Export to OSM** toolbar: **Sheet Metal | Export to OSM** ribbon: **Sheet Metal | Flatten | Export to OSM : SmExportOSM Prompts you in the command bar:** Select a sheet metal solid, flange face or lofted bend face [add Decorations]: **(Select a sheet metal body or choose an option.)** Displays a file dialog: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/s

Figure 6.2265



Command Options

Option	Description
Settings	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee Set the accuracy of the approximation: The OSM file format requires splines to be replaced by linear segments and circular arcs. The accuracy of this approximation (allowable deviation of a segment/arc from the spline) can be specified for each run of the **SmExportOSM** command. Minimal Edge Length: Controls the minimal length of edges in the resulting .osm file.
add Decorations	Allows to export adjacent non-sheet metal parts in same .osm part

Related Commands

SmAssemblyExport (6.28.1) - traverses the Assembly Structure of the current drawing, searches for 3D solids in its component, and applies the SmConvert (6.28.4) and SmExport2d (6.28.7) commands. SmExport2d (6.28.7) - exports the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format. SmUnfold (6.28.29) - creates an unfolded 2D or 3D representation of a sheet metal part.

Figure 6.2266

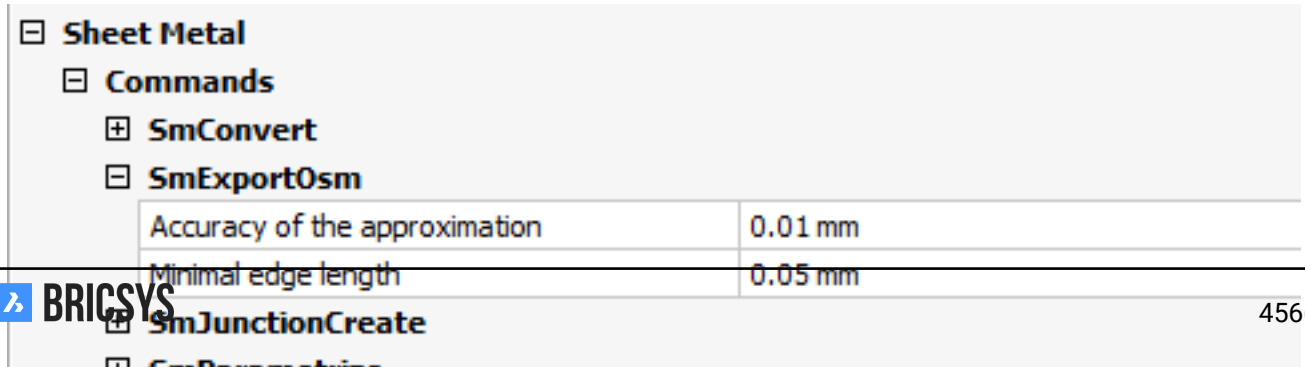


Figure 6.2267



6.28.9 SmExtrude

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline.

Accessing the Command


command bar: **smextrude** menu bar: **Sheet Metal | Bend | ExtrudePolyline** toolbar: **Sheet Metal |Extrude-Polyline** ribbon: **Sheet Metal | Create |ExtrudePolyline** : **SmExtrude Prompts you in the command bar:** Select a polyline: **(Select a polyline.)** The polyline is extruded dynamically. Specify height of extrusion: **(Pick a point or type a value.)** The thickening direction of the extruded surface is applied for each flange separately, in order to preserve the overall dimensions of polyline: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SM Selected Polyline (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SMExtrude1.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SMExtrude2.png}) The overall dimensions of the Sheet Metal part match the dimensions of the extruded polyline.

Command Options

No command options.

Related Commands

SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmFlangeBase (6.28.10) - creates a base (initial) flange of a sheet metal part from a closed planar profile. SmFlangeBend (6.28.11) - allows to bend an existing flange along a line, obeying k-factor for given bend radius. SmFlip (6.28.15) - changes reference side for selected flanges with optional shift of flange geometry by thickness. SmParametrize (6.28.21) - creates a consistent set of 3D constraints for a selected sheet metal part.

Figure 6.2268



Figure 6.2269

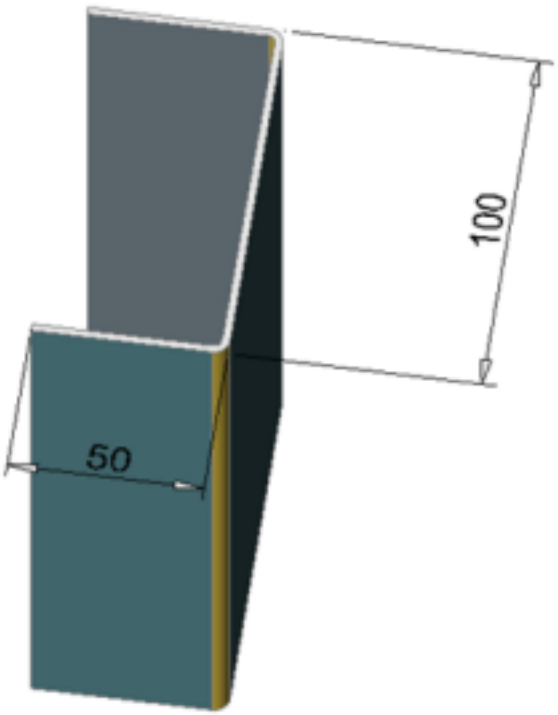


Figure 6.2270

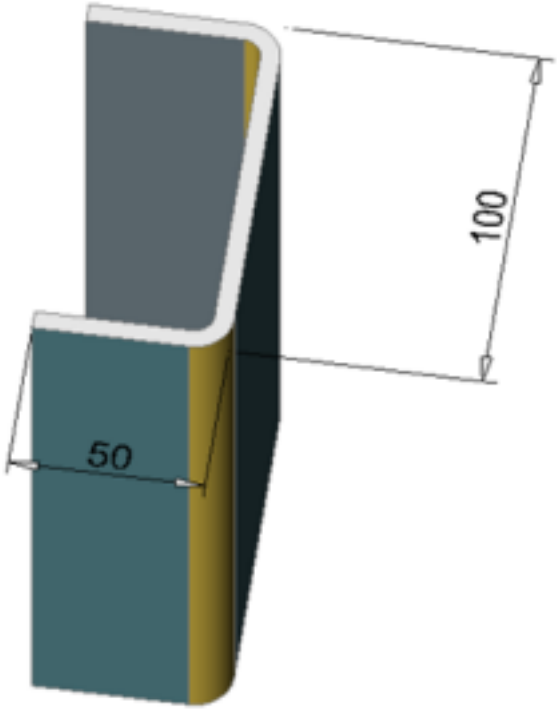


Figure 6.2271



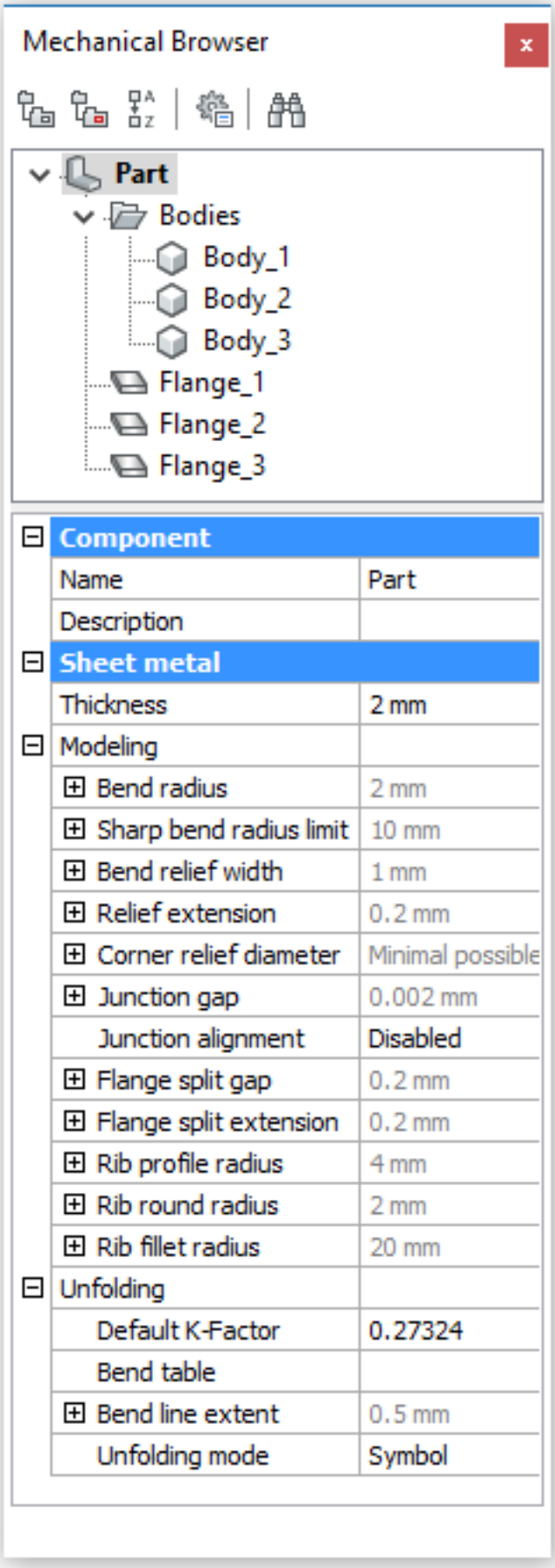
6.28.10 SmFlangeBase

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Creates a base (initial) flange of a sheet metal part from a closed planar profile.

Accessing the Command


command bar: **smflangebase** menu bar: **Sheet Metal | Create Base Flange** toolbar: **Sheet Metal | Create Base Flange** ribbon: **Sheet Metal | Create | Create Base Flange** Quad cursor menu: **Sheet Metal | Create Base Flange : SmFlangeBase Prompts you in the command bar:** Select closed polyline or a region: **(Select a closed polyline or a region)** Entities/subentities in set: 1 Select closed a polyline or a region: **(Select another closed polyline or a region or right click to confirm)** Make flange base Up/Middle/Down/<Accept model>: **(Select an option or press the Ctrl-key to cycle the various options; press Enter or right click to accept the selected option.)** A base flange is created for each selected closed polyline or region. In the Mechanical Browser (3.2.2) a body and a flange are added for each entity: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/sm

Figure 6.2272



Command Options

Option	Description
Up	The flange is extruded on top of the base entity.
Center	The flange is extruded on both sides of the base entity.
Down	The flange is extruded below the base entity.

Related Commands

SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmDissolve (6.28.6) - removes sheet metal data from the selected faces or entities. SmExtrude (6.28.9) - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange. SmFlip (6.28.15) - swaps sides of a selected flange so that reference faces are on the other geometric side of the flange. SmLoft (6.28.20) - creates sheet metal part with lofted bends and flanges from two non-coplanar curves. SmParametrize (6.28.21) - creates a consistent set of 3D constraints for a selected sheet metal part. SmRepair (6.28.24) - restores the 3D solid model of a sheet metal part by thickening one of its sides (all thickness faces become perpendicular to flange faces).

Figure 6.2273



6.28.11 SmFlangeBend

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Allows to bend an existing flange along a line, obeying k-factor for given bend radius.

Accessing the Command

 command bar: **smflangebend** menu bar: **Sheet Metal | Bend | Flange Bend** toolbar: **Sheet Metal | Bend | Flange Bend** ribbon: **Sheet Metal | Modify | Bend |Flange Bend** Quad: **Sheet Metal | Flange Bend : SmFlangeBend Prompts you in the command bar:** Select a flange face: **(Select a major face of flange.)** Select line or edge to bend the solid or draw a [New line] <New line>: **(Select a line or an edge.)** Pick a point on the side that will be moved: **(Pick a point on the side of the line that will be moved.)** Select flange position [Angle/Radius/Switch side/ACcept] <Accept>: **(Choose an option or press Enter.)**

Command Options

Option	Description
New line	Allows to draw a line to define the bend location; prompts you: **Start point of the line** - **(Pick a point.)** **End point of the line** - **(Pick a point.)**
Angle	Allows to set the bend angle; prompts you: **Enter bend angle [Back] <Back>: - (Do one of the following:)** • Type a value. • Press Enter to go back without changing the bend angle.
Radius	The default bend radius is defined by the **Modeling/Bend Radius** property in the Mechanical Browser for Sheet Metal (3.8.5). Allows to define the bend radius; prompts you: **Enter bend radius [Back] <Back>:** - **(Do one of the following:)** • Type a value. • Press Enter to go back without changing the bend radius.
Switch side	Allows to move the part of the flange on the other side of the line or edge.
Accept	Creates the bend at the current angle and radius.

Procedures

To bend multiple flanges simultaneously (**??**)

Related Commands

SmExtrude (6.28.9) - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline. SmFlangeBase (6.28.10) - creates a base (initial) flange of a sheet metal part from a closed planar profile. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange. SmFlangeRotate (6.28.14) - rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent

Figure 6.2274



6.28.12 SmFlangeConnect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Closes gaps between two arbitrarily oriented non-coplanar flanges.

Accessing the Command

 command bar: **smflangeconnect** menu bar: **Sheet Metal | Connect Flanges** toolbar: **Sheet Metal | Connect Flanges** ribbon: **Sheet Metal | Modify | Connect Flanges** Quad cursor menu: **Sheet Metal | Connect Flanges : SmFlangeConnect Prompts you in the command bar:** Select planar thickness faces of two flanges: **(Select the face on the first flange.)** Entities/subentities in set: 1 Select planar thickness faces of two flanges: **(Select the face on the second flange.)** The flanges are connected. If SELECTIONPREVIEW (1.13.11) is 2 or 3, thickness faces of flanges are highlighted. Press the TAB key to select obscured faces.

Quad cursor menu

Select the planar thickness faces of two flanges, then choose **Connect Flanges** () in the **Sheet Metal** command group. Select the thickness faces (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetM
A junction is created (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/smflang

Figure 6.2275

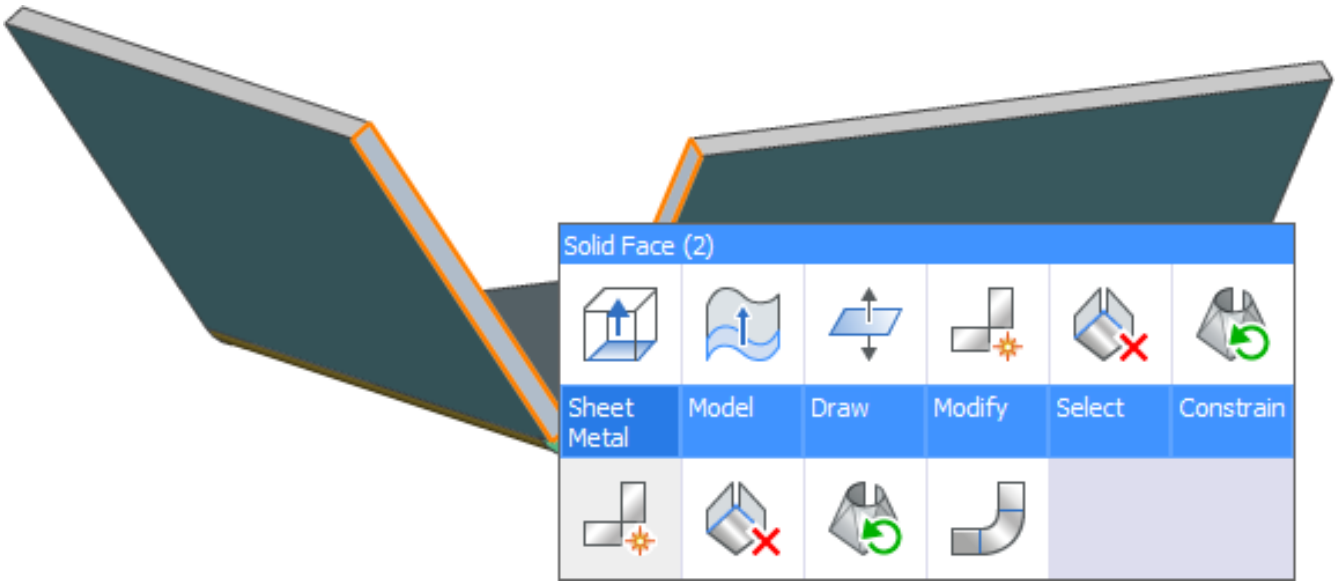
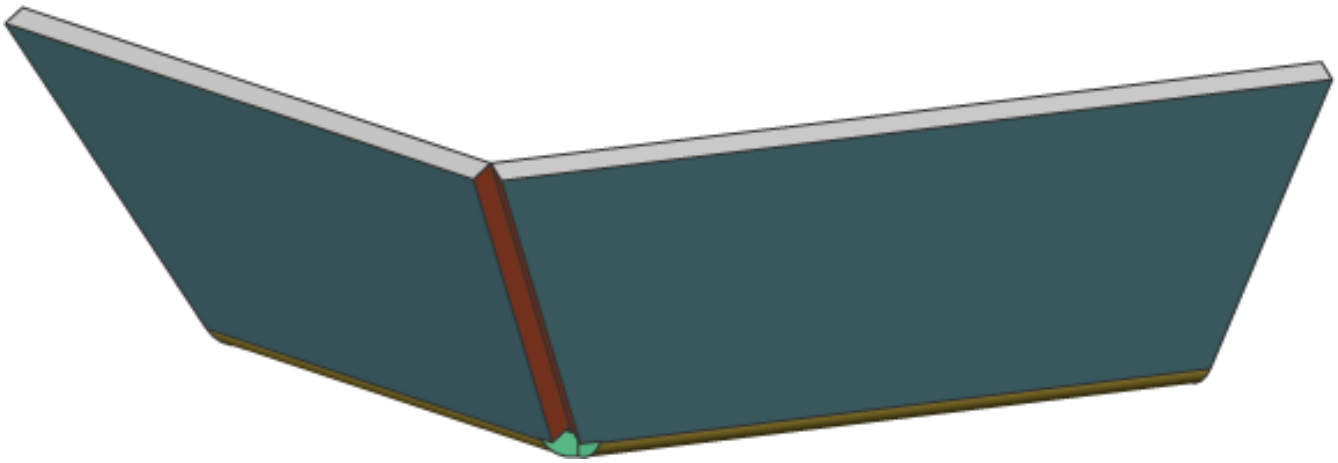


Figure 6.2276



Command Options

No command options.

Related Commands

SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange. SmFlangeRotate (6.28.14) - rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent. SmSplit (6.28.28) - splits a flange along a line drawn on its face. SmJunctionCreate (6.28.18) - converts hard edges (sharp edges between flange faces) and bends into junctions. SmParametrize (6.28.21) - creates a consistent set of 3D constraints for a selected sheet metal part. SmReliefCreate (6.28.22) - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.

Figure 6.2277



6.28.13 SmFlangeEdge

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.

Accessing the Command

 command bar: **smflangeedge** menu bar: **Sheet Metal | Create Edge Flange** toolbar: **Sheet Metal | Create Edge Flange** ribbon: **Sheet Metal | Modify | Create Edge Flange** Quad cursor menu: **Sheet Metal | Create Edge Flange : SmFlangeEdge Prompts you in the command bar:** Select edges on flanges: **(Select edges, then right click to proceed.)** Entities/subentities in set: 1 Position the end of the flange [Angle/Length/Radius/Taper angle/Width]: **(Specify a point in 3D space or type a value in the dynamic height and angle fields)**

Command Options

Option	Description
Dynamic dimensions	Define the height and angle of the flange by typing values in the dynamic height (1) and angle (2) entry fields: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S Enter a value in the height field, then press the TAB key to jump to the angle field. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S Type a value in the angle field, then do one of the following: • Press Enter twice to accept the current values. • Press the TAB key to enter a different value in the height field, then press Enter to accept the current values. • Keep pressing the TAB key to switch between the height and angle field to enter a different value, then press Enter to accept the current values.
Angle	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S Allows to first define the angle, then the height of the flange.
Length	Allows to first define the length, then the angle of the flange.
Radius	Allows to define the radius of the bend connected to the new flange.

Taper Angle(*)	Allows to create a flange with one or two tapered side faces. Prompts you: **Enter first taper angle [Back/Skip] <Skip>:-(Do one of the following:)** • Type a value in the taper angle entry field to define the taper angle for the first side face. • Select the **Skip** option to keep the first side face orthogonal. • Select the **Back** option to undo the current **Taper angle** procedure. **Enter second taper angle [Back/Skip] <Skip>: -(Do one of the following:)** • Type a value in the taper angle entry field to define the taper angle for the second side face. • Select the **Skip** option to keep the second side face orthogonal. • Select the **Back** option to undo the current **Taper angle** procedure. **Position the end of the flange [Angle/Length/Taper angle/Width]:** - **(Do one of the following:)** • Click a point or use the Dynamic Dimensions procedure to define the flange height and angle. • Choose the Width option to define a width different from the length of the selected edge. • Choose the **Taper angle** option again to define different taper angles.
Width(*)	By default the flange width equals the length of the selected edge. The Width option allows to define a different width. Prompts you: **Enter first offset [Back/Skip] <Skip>:- (Do one of the following:)** • Type a value in the dynamic field to define the offset from the start point of the edge. • Select the **Skip** option to keep the first side edge of the flange at the start point of the edge. • Select the **Back** option to undo the current **Width** procedure. **Enter second offset [Back/Skip] <Skip>: -(Do one of the following:)** • Type a value in the dynamic field to define the offset from the end point of the edge. • Select the **Skip** option to keep the second side edge of the flange at the end point of the edge. • Select the **Back** option to undo the current **Width** procedure. **Position the end of the flange [Angle/Length/Taper angle/Width]:- (Do one of the following:)** • Click a point or use the Dynamic Dimensions procedure to define the flange height and angle. • Choose the Taper angle option to define tapered side faces • Choose the **Width** option again to define different offsets.

Toggle connection(**)	Toggle connection helps to avoid material conflict when creating several flanges simultaneously: No connection (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee Pull outside with connection (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee Pull insidewith connection (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee Use Ctrl-key to cycle the options, then press Enter to accept the current flanges connection. If TIPS (1.12.2) = ON, the currently selected option is indicated in the Tips widget at the bottom of the screen.

(*) The **Taper Angle** and **Width** options are not available if multiple edges are selected. (**) The **Toggle connection** available when selected at least two neighbor edges.

Related Commands

SmBendCreate (6.28.2) - converts hard edges (sharp edges between flange faces) and junctions into bends. SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmDissolve (6.28.6) - removes sheet metal data from the selected faces. SmFlangeBase (6.28.10) - creates a base (initial) flange of a sheet metal part from a closed planar profile. SmFlangeBend (6.28.11) - allows to bend an existing flange along a line, obeying k-factor for given bend radius. SmFlangeConnect (6.28.12) - closes gaps between two arbitrarily oriented flanges. SmFlangeRotate (6.28.14) - rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent. SmSplit (6.28.28) - splits a flange along a polyline drawn on its face. SmFlip (6.28.15) - swaps sides of a selected flange so that reference faces are on the other geometric side of the flange. SmJunctionCreate (6.28.18) - converts hard edges (sharp edges between flange faces) and bends into junctions. SmParametrize (6.28.21) - creates a consistent set of 3D constraints for a selected sheet metal part. SmReliefCreate (6.28.22) - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge. SmRepair (6.28.24) - restores the 3D solid model of a sheet metal part by thickening one of its sides (all thickness faces become perpendicular to flange faces).

Figure 6.2278

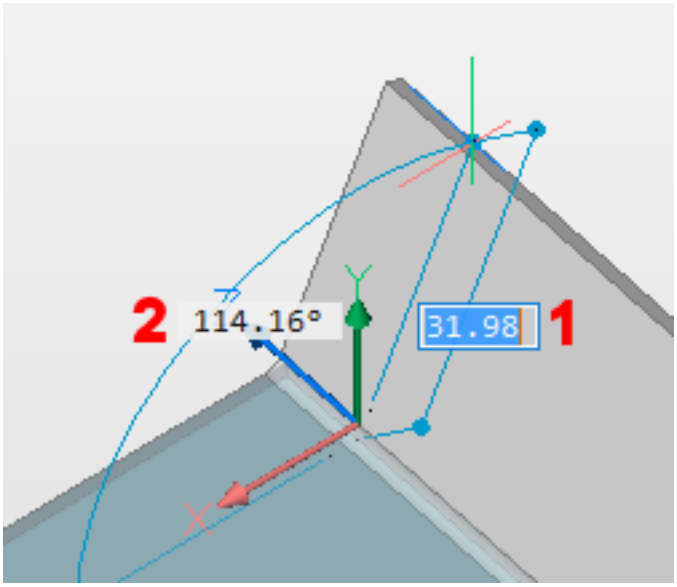


Figure 6.2279

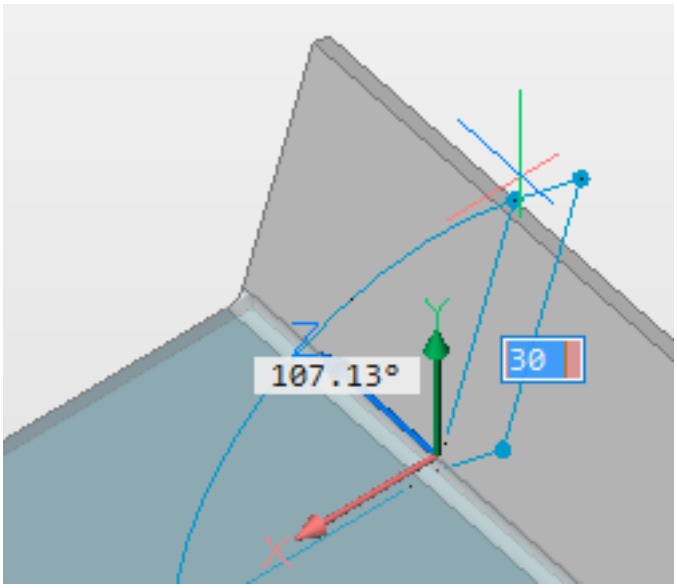


Figure 6.2280

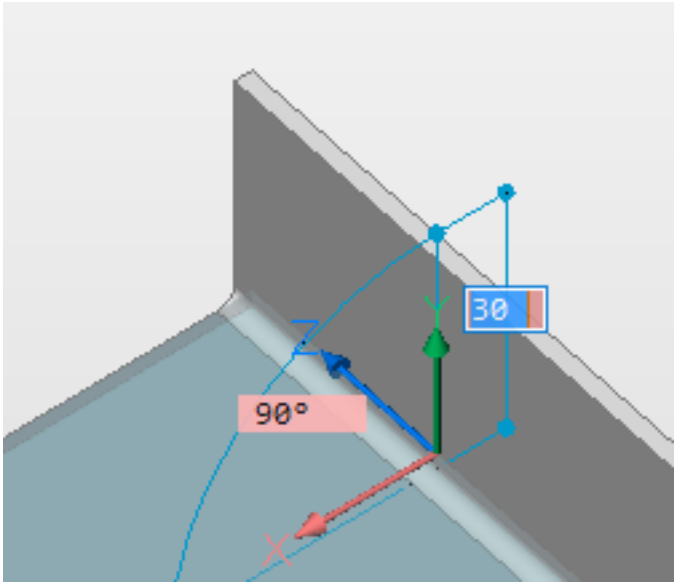


Figure 6.2281

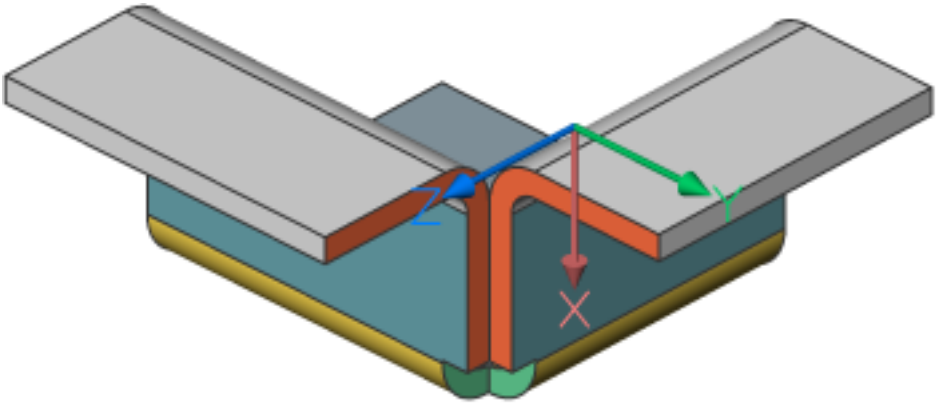


Figure 6.2282

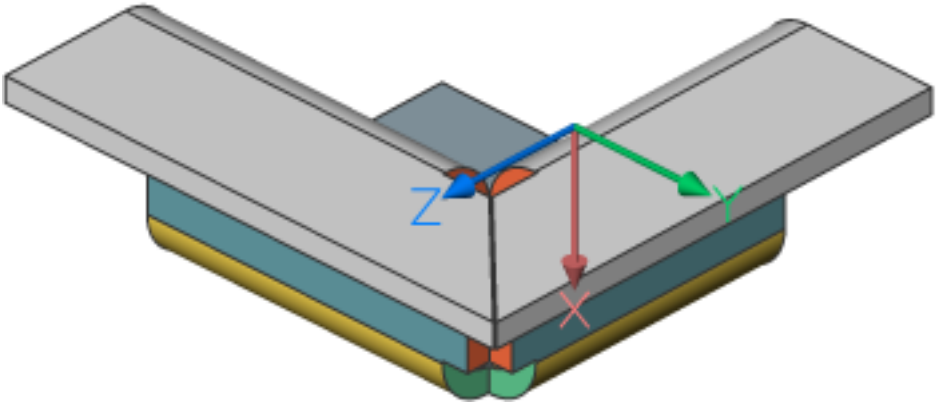


Figure 6.2283

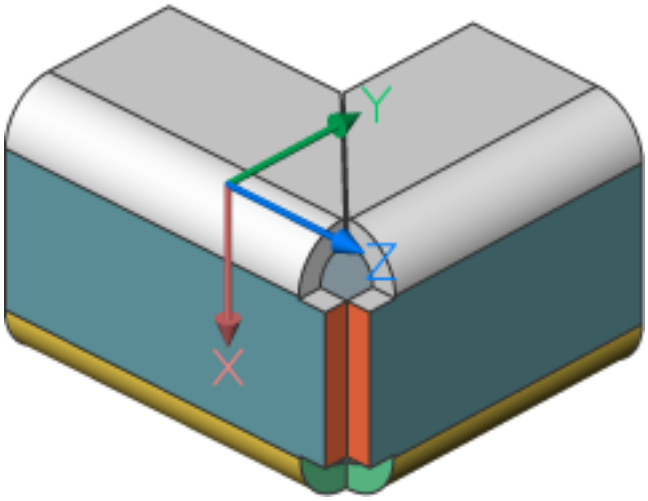


Figure 6.2284



6.28.14 SmFlangeRotate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent.

Accessing the Command


command bar: **smflangerotate** menu bar: **Sheet Metal | Rotate Flange** toolbar: **Sheet Metal | Rotate Flange** ribbon: **Sheet Metal | Edit | Rotate Flange** Quad cursor menu: **Sheet Metal | Rotate Flange : SmFlangeRotate Prompts you in the command bar:** Select a flange face to rotate: **(Select the face of a flange.)** A dynamic angle field displays the angle between the selected flange and the base flange. Press the TAB key to display the absolute rotation angle. **(Type a value in the dynamic angle field and press Enter to confirm.)**

Command Options

No options.

Related Commands

SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmExtrude (6.28.9) - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline. SmFlangeConnect (6.28.12) - closes gaps between two arbitrarily oriented flanges. SmFlangeBend (6.28.11) - allows to bend an existing flange along a line, obeying k-factor for given bend radius. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange. SmFlip (6.28.15) - swaps sides of a selected flange so that reference faces are on the other geometric side of the flange.

Figure 6.2285



6.28.15 SmFlip

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Swaps sides of a selected flange so that reference faces are on the other geometric side of the flange. Optionally shifts the flange over the thickness of the sheet metal part.

Accessing the Command

 command bar: **smflip** menu bar:Sheet Metal | Flange | **Flip flange** toolbar: **Flange | Flip FlangeFeature** ribbon: **Sheet Metal | Modify | Flip Flange** Quad cursor menu: **Sheet Metal |Flip FlangeFeature : SmFlip Prompts you in the command bar:** Select solids or flange faces to flip flange(s) [Flip reference side only/Entire model] <Entire model>:**(Select a solid body or a flange faces or press Enter to flip sides for all flanges.)** Entities in set: 1 Select solids or flange faces to flip flange(s) [Flip reference side only/Entire model]: **(Select more faces or press Enter.)**

Command Options

Option	Description
Flip reference side only	Turns OFF the shift of flanges by thickness, in this mode reference sides for selected flanges are just swapped. The command prompt now reads: **Select solids or flange faces to flip reference side of flange(s) [Flip flange/Entire model] <Entire model>:**
Entire Model	Applies flip operation to all flanges in the drawing.

Changing reference faces with shift by thickness might be necessary if SmExtrude (6.28.9) creates flanges which collide with another part:

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/SmFlipCollision.png}) Collisions between a solid and sheet metal flanges	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SmFlipCollisionSolved.png}) Collisions solved after applying SmFlip.

Related Commands

SmConvert (6.28.4) - automatically recognizes flanges and bends in a 3D solid. SmFlangeBase (6.28.10) - creates a base (initial) flange of a sheet metal part from a closed planar profile. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange. SmParametrize (6.28.21) - creates a consistent set of 3D constraints for a selected sheet metal part.

Figure 6.2286

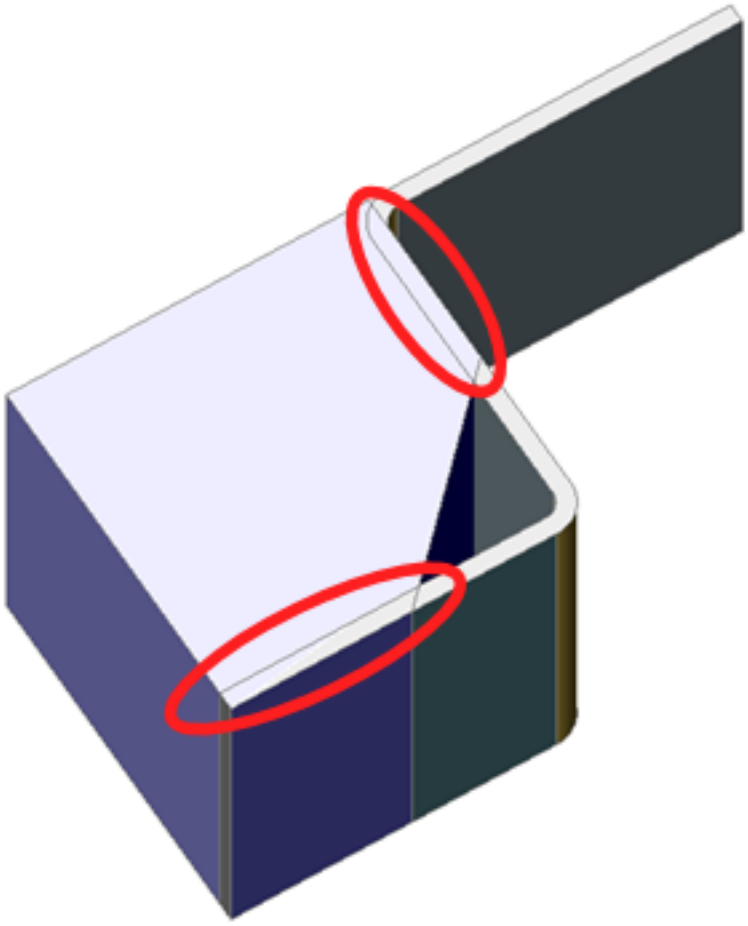


Figure 6.2287

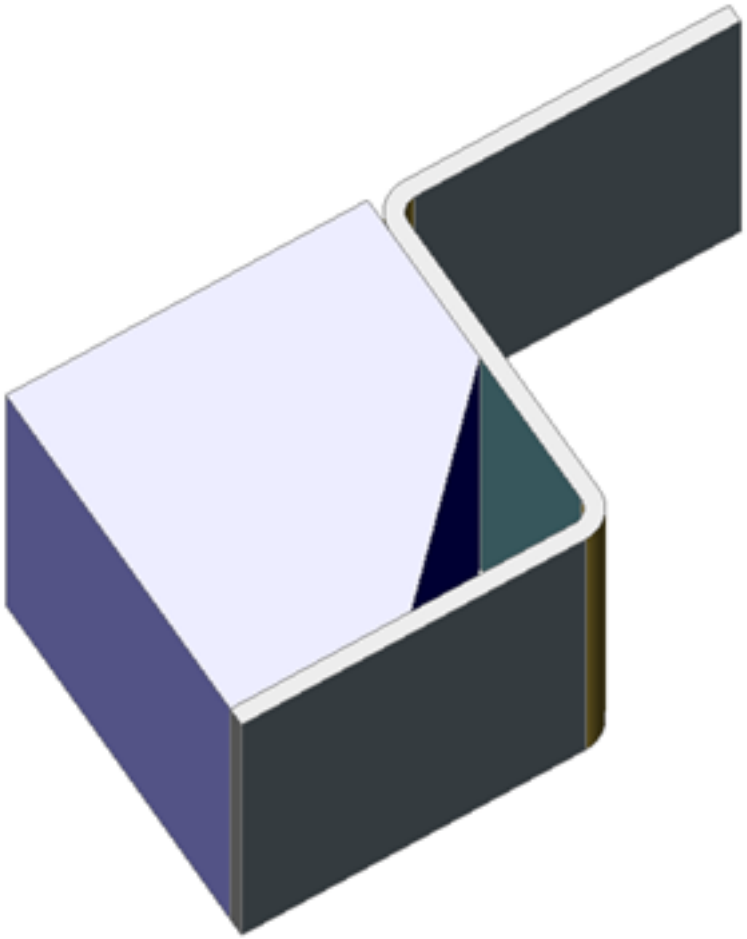


Figure 6.2288



6.28.16 SmForm

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Converts a selected set of faces to a form feature or explodes form feature into flanges and bends. In **Create** mode the command allows to define a form feature by selecting the faces manually, which can be necessary if automatic recognition by SmConvert (6.28.4) does not produce the expected result. E.g.: if you insert and dissolve the **Cardguide** form feature from the library, SmConvert (6.28.4) recognizes two forms features, because their geometry is separated. Manual selection of the form feature faces when executing SmForm solves the problem. Form features created by SmForm have the same set of basic operations as if they were created by BmInsert (6.8.12) or recognized by SmConvert (6.28.4). In **Explode** mode the command dissolves the form feature and tries to recognize its geometry as flanges and bends. This can be helpful if SmConvert (6.28.4)incorrectly recognizes the design as a form feature.

Accessing the Command


command bar:**smform** menu bar:Sheet Metal | Form | Define form Faces toolbar: Sheet Metal | **Define form Faces>** ribbon: **Sheet Metal | Modify | Define Form : SmForm Prompts you in the command bar:** Select form feature faces [Create new form feature/Explode form feature] <Create new form feature>: **(Select faces or choose an option to switch the command mode to Create or Explode.)**

Command Options

Option	Description
Create new form feature	Sets the command in **Create** mode, which creates new form features. Prompts you to select faces which do not belong to any existing feature.
Explode form feature	Sets the command in **Explode** mode, which explodes existing form features into flanges and bends. Prompts you to select faces of existing form features. At least one face of each form feature to be exploded must be selected.

Procedures

Working with form features (3.8.18)

Related Commands

BmInsert (6.8.12) - inserts an existing mechanical component or sheet metal form feature into the current drawing. SmConvert (6.28.4) - automatically recognizes flanges and bends in a 3D solid. SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmDissolve (6.28.6) - removes sheet metal data from the selected features.

Figure 6.2289



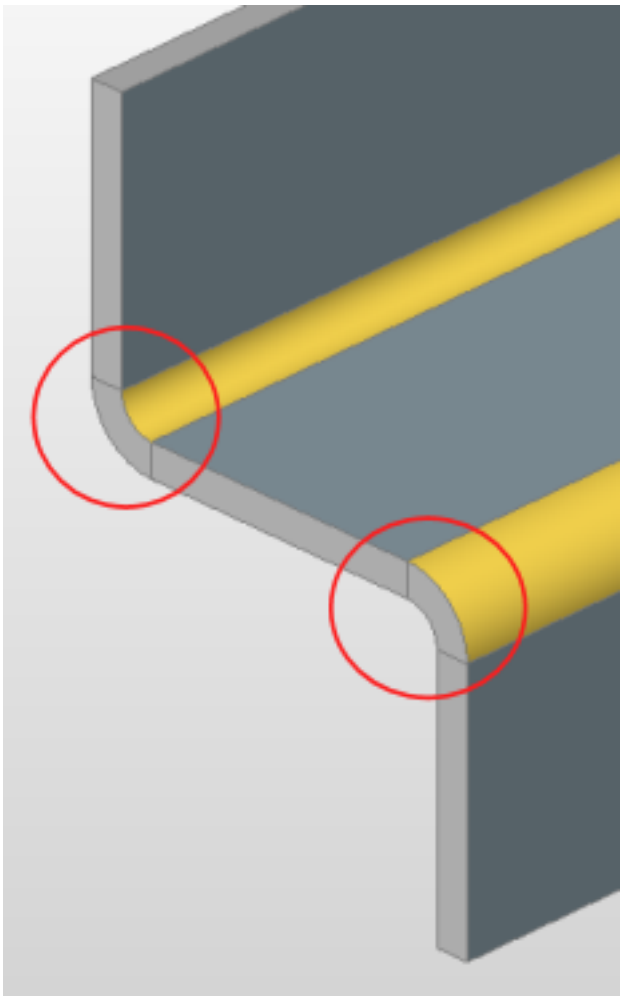
6.28.17 SmImprint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Splits thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends.

Accessing the Command

 command bar: **smimprint** menu bar: **Sheet Metal | Split thickness face** toolbar: **Sheet Metal | Split thickness face** ribbon: **Sheet Metal | Modify | Split thickness face** Quad cursor menu: **Sheet Metal | Split thickness face : SmImprint Prompts you in the command bar:** Select thickness face(s): **(Select a thickness face)** Entities in set: 1 4 edge(s) were imprinted. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SmImprint

Figure 6.2290



Command Options

No options.

Related Commands

SmJunctionCreate (6.28.18) - converts hard edges (sharp edges between flange faces) and bends into junctions.
SmJunctionSwitch (6.28.19) - allows changing a symmetrical junction feature to one with overlapping faces.

Figure 6.2291



6.28.18 SmJunctionCreate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Converts hard edges (sharp edges between flange faces) and bends into junctions.

Accessing the Command

 command bar: **smjunctioncreate** menu bar: **Sheet Metal | Create Junction** toolbar: **Sheet Metal | Create Junction** ribbon: **Sheet Metal | Modify | Create Junction** Quad cursor menu:**Sheet Metal | Create Junction : SmJunctionCreate Prompts you in the command bar:** Select hard edges or bends or flanges or 3D solids [Entire model] <Entire model>: **(Select flanges, hard edges, bends or solids. You can also select "curved" edges between flanges and bends; between flanges and lofted bends.)** Entities in set: 1 Select hard edges or bends or flanges or 3D solids [Entire model] <Entire model>: **(Select more items or right click.)**

Command Options

Option	Description
Entire model	Analyzes the entire model to detect hard edges.

Command Settings

Settings	Heal coincident junction faces • Switches the mode for recognition of junction-like coincident faces and conversion to regular junctions. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee

Related Commands

SmBendCreate (6.28.2) - converts hard edges (sharp edges between flange faces) and junctions into bends. SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmDissolve (6.28.6) - removes sheet metal data from the selected faces or entities. SmFlangeConnect (6.28.12) - closes gaps between two arbitrarily oriented flanges. SmImprint (6.28.17) - splits thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends. SmJunctionSwitch (6.28.19) - allows changing a symmetrical junction feature to one with overlapping faces. SmReliefCreate (6.28.22) - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.

Figure 6.2292

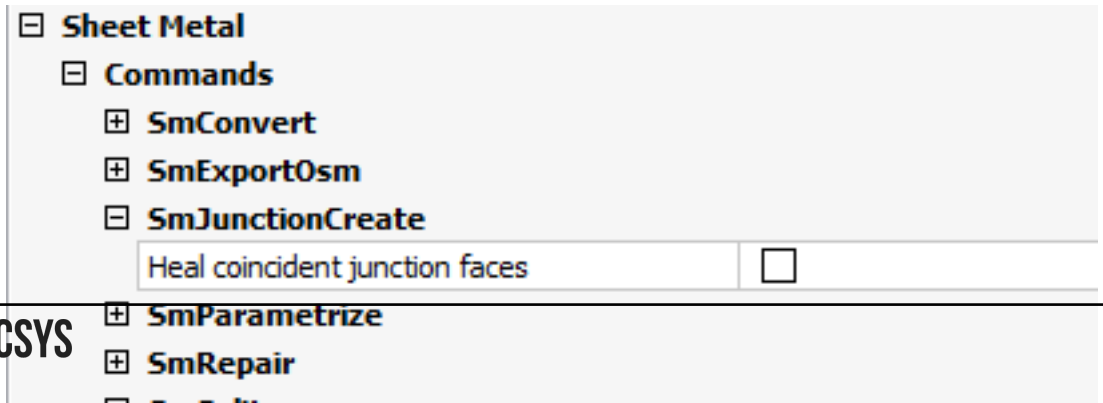


Figure 6.2293



6.28.19 SmJunctionSwitch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Allows changing a symmetrical junction feature to one with overlapping faces.

Accessing the Command


command bar: **smjunctionswitch** menu bar: **Sheet Metal | Switch Junction type** toolbar: **Sheet Metal | Switch Junction type** ribbon:**Sheet Metal | Modify | Switch Junction type** Quad cursor menu:**Sheet Metal | Switch Junction type : SmJunctionSwitch Prompts you in the command bar:** Select junction faces: **(Select the junction face of left flange, then right click.)** Entities in set: 1 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/C
The junction face of the left flange overlaps the junction face of right flange. (\vref {fig:C:/Dev/Task-2448/images/staticbricsys

Figure 6.2294

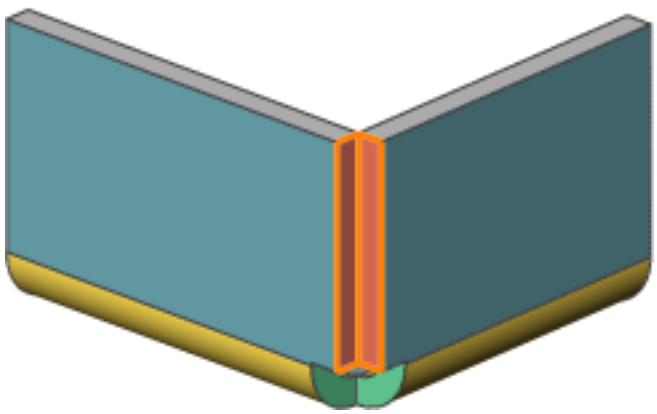
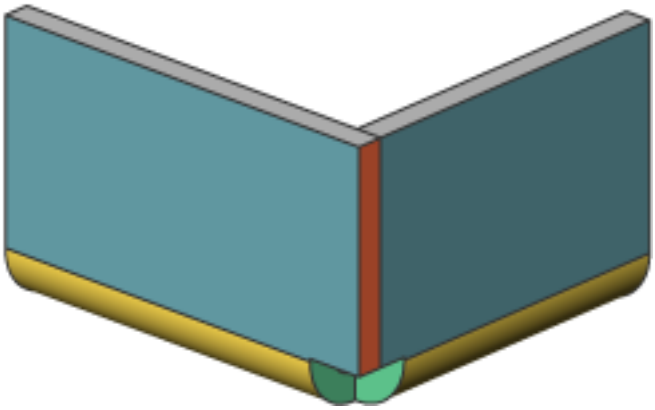


Figure 6.2295



Command Options

No command options.

Related Commands

SmImprint (6.28.17) - splits thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends. SmJunctionCreate (6.28.18) - converts hard edges (sharp edges between flange faces) and bends into junctions.

Figure 6.2296



6.28.20 SmLoft

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Creates sheet metal part with lofted bends and flanges from two non-coplanar curves.

Accessing the Command


command bar: **smloft** menu bar: **Sheet Metal | Create Sheet Metal Loft** toolbar: **Sheet Metal | Create Sheet Metal Loft** ribbon: **Sheet Metal | Create | Create Sheet Metal Loft : SmLoft Prompts you in the command bar:** Select two cross sections: **(Select two, non coplanar open or closed 2D entities.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SmLoftProfiles.png}) Set [fillets Radius/Thickness/Single bend/Flanges with bends] <Flanges with bends>: **(Press Enter or choose an option.)** Set [fillets Radius/Thickness/Single bend/Flanges with bends/thicken Inside/Both sides/thicken Outside] <thicken Inside>: **(Press Enter or choose an option.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SmLoftAlignmentOptions.p
Hit the CTRL key to adjust the alignment: Outside, Both Sides or Inside (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h

Figure 6.2297

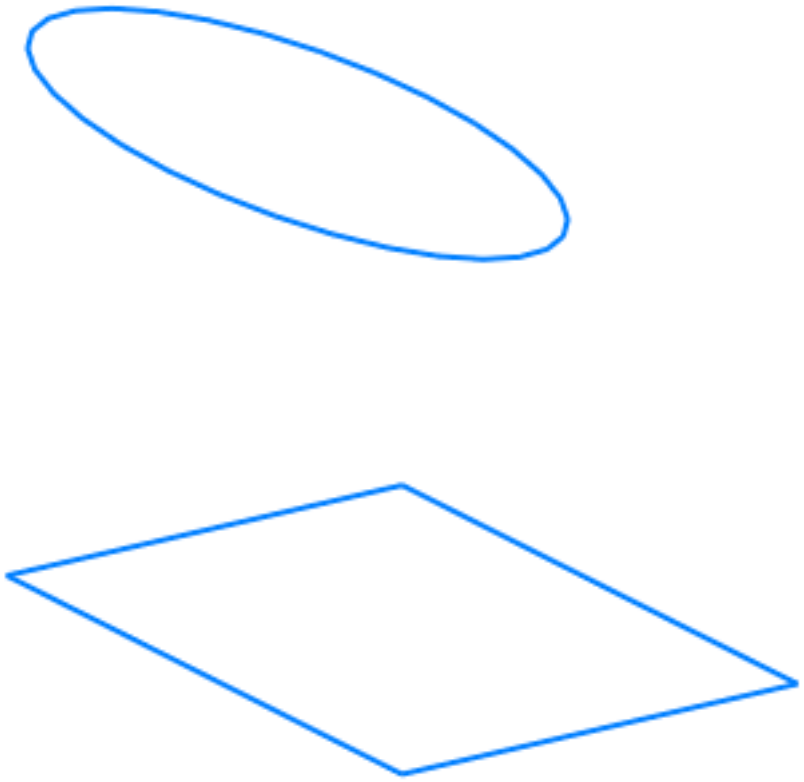


Figure 6.2298

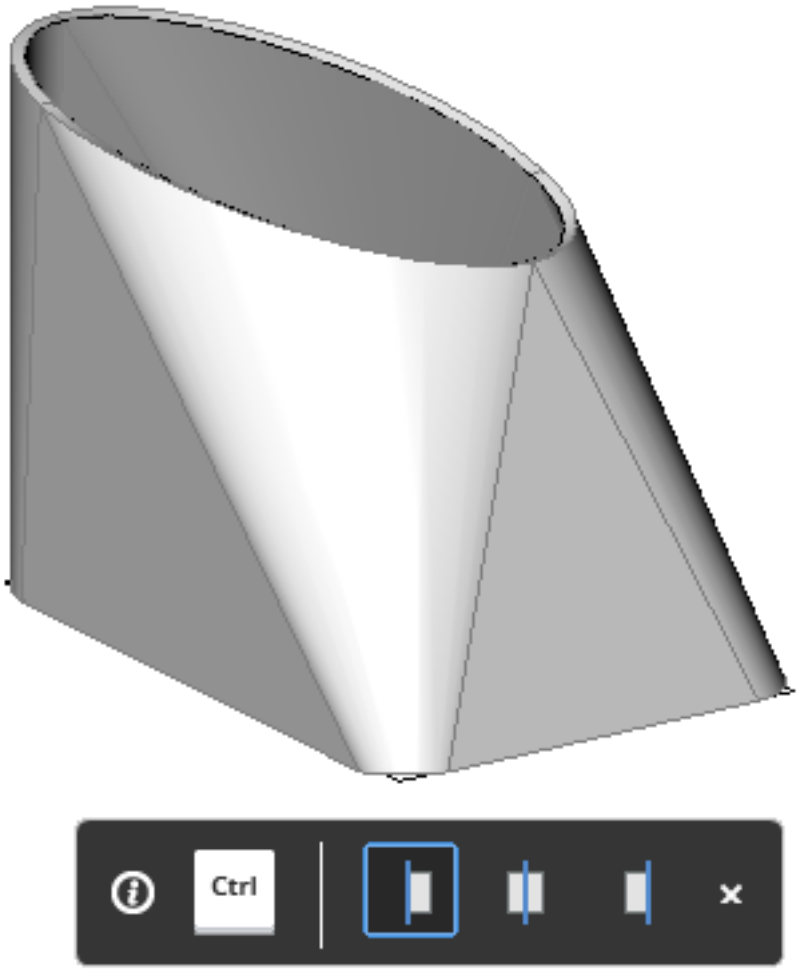
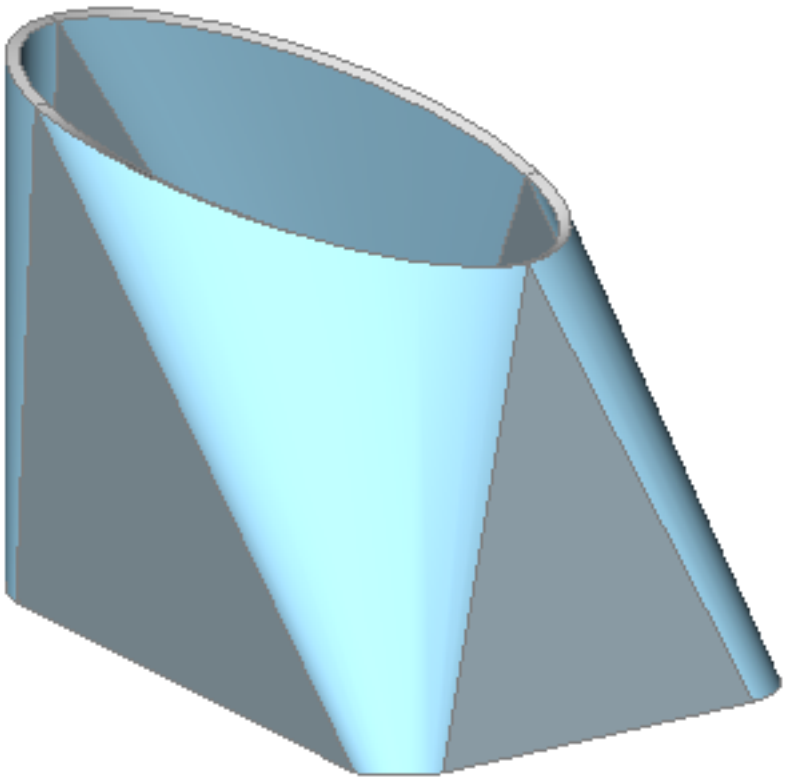


Figure 6.2299



Command Options

Option	Description
Fillets Radius	Sets radius of fillets which will be created on straight lines connection. Prompts you: **Select fillets radius <auto>:- (Do one of the following:)** • Type a value for the fillet radius. • Press Enter to automatically calculate the minimal value for the fillet radius, which allows to create the model with the current thickness. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S Automatically calculated radius (left); user defined radius (right)
Thickness	Sets model thickness. Prompts you: **Select thickness <2.000000>:-(Type a value or press enter to accept the default value.)**
Single Bend	Creates only lofted bend feature without any flanges. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S Single lofted bend and its unfolded view.
Flanges with Bends	Creates a flange feature for each straight segment of the selected 2D entities or a lofted bend feature for curved segments. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S Lofted bend with flanges and its unfolded view. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S From left to right: thickening outside, both sides an inside.
Inside	Thickening direction is selected that way that 3D part is inside the selected 2D entities.
Both Sides	Thickens at both sides of the selected 2D entities.
Outside	Thickening direction is selected that way that 3D part is outside the selected 2D entities.

Related Commands

SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmExtrude (6.28.9) - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline. SmFlangeBase (6.28.10) - creates a base (initial) flange of a sheet metal part from a closed 2D entity. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.

Figure 6.2300

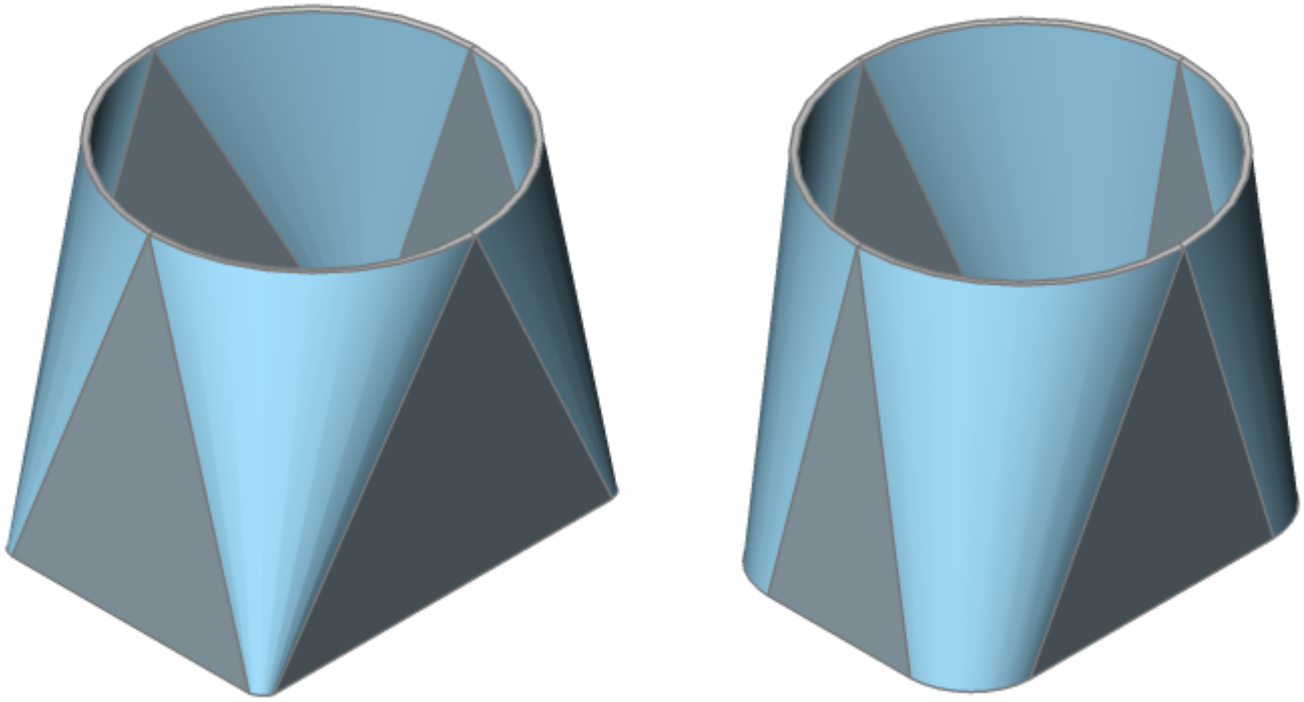


Figure 6.2301

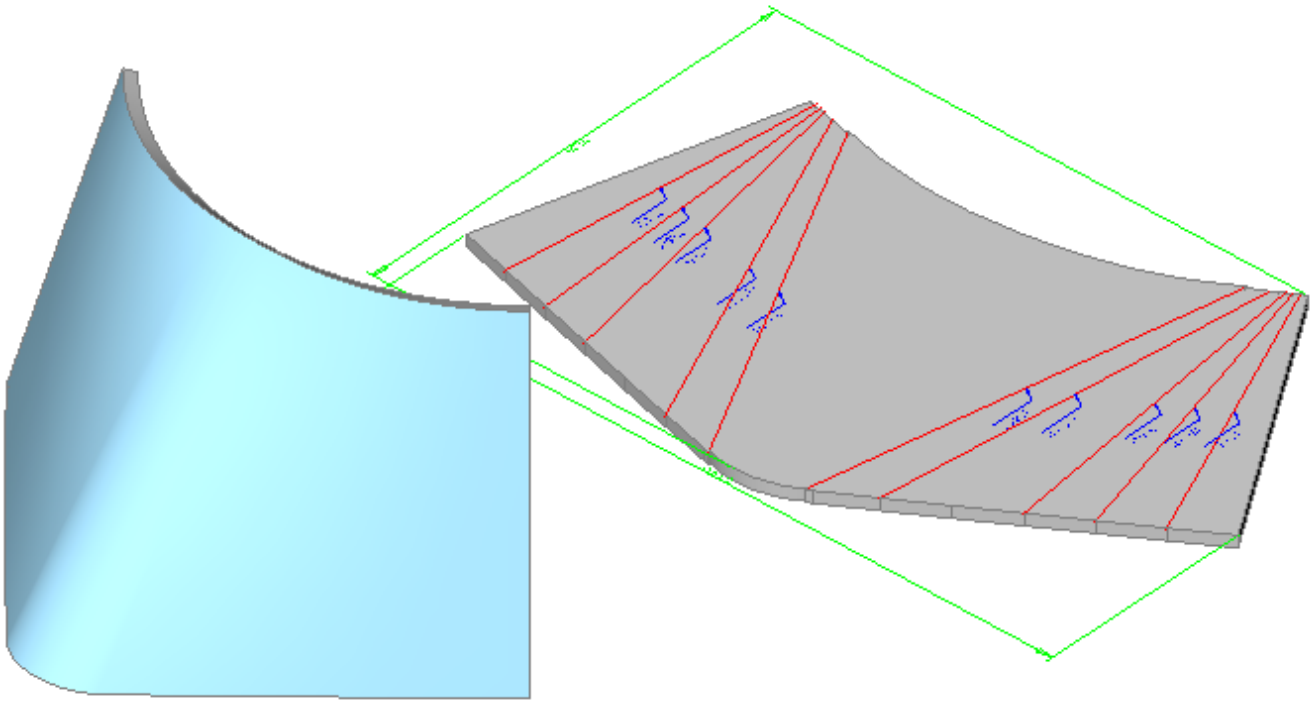


Figure 6.2302

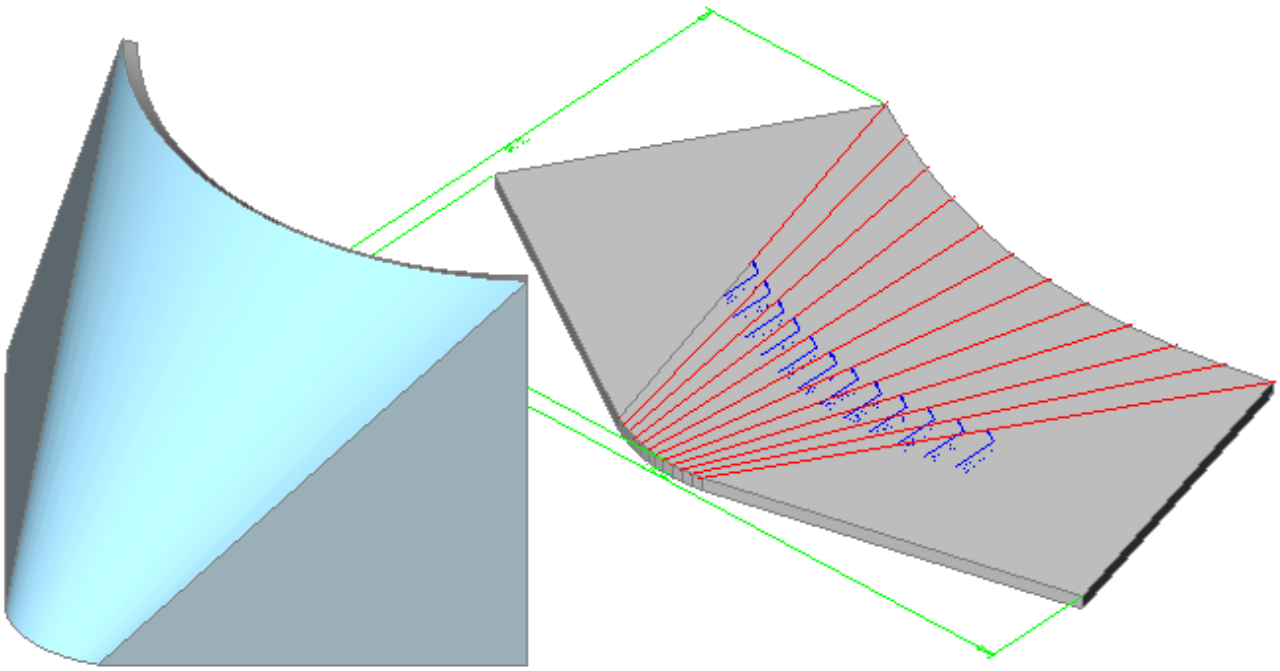


Figure 6.2303

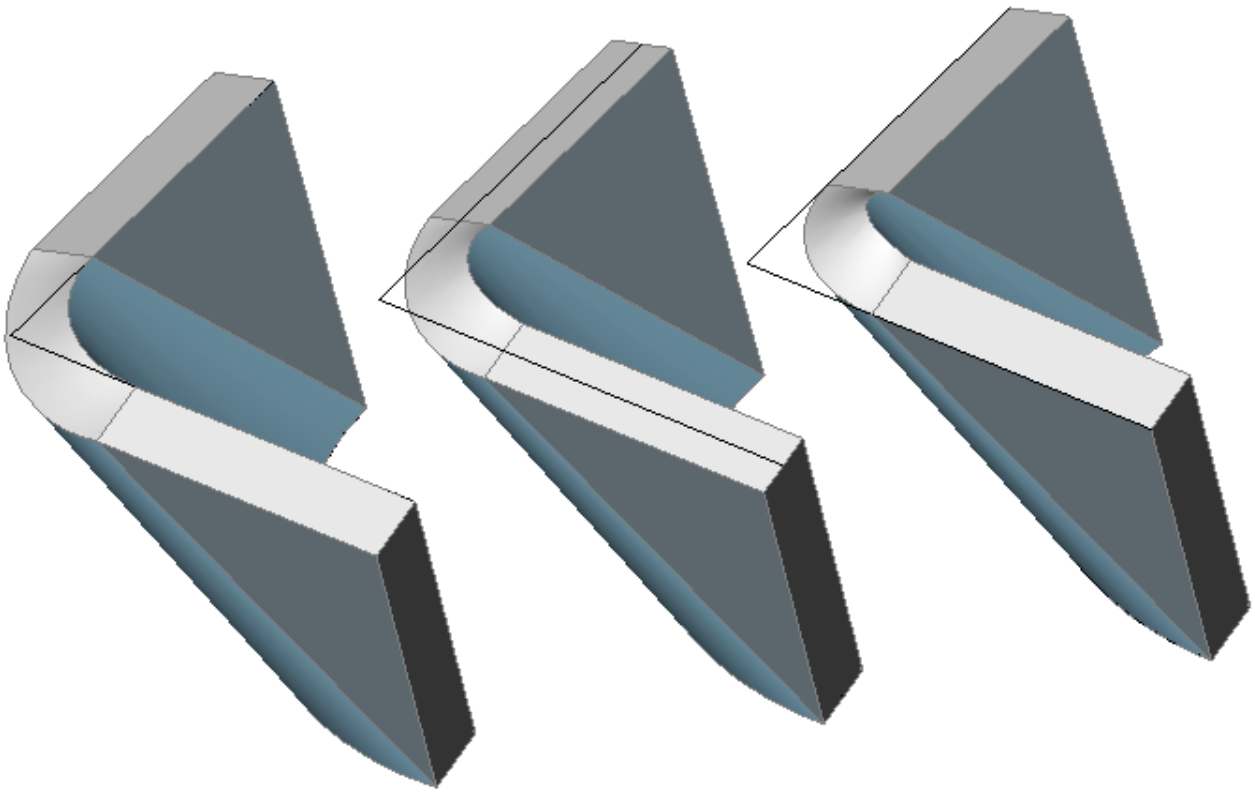
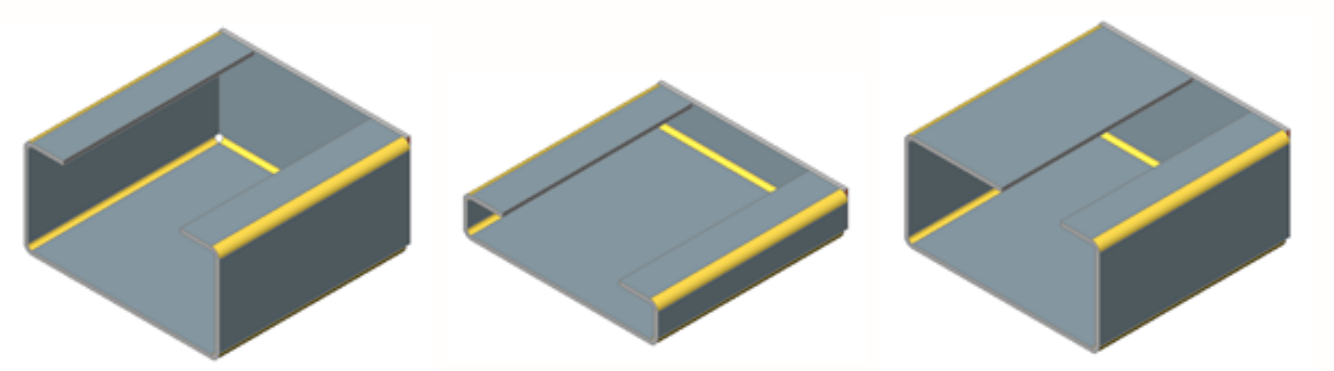


Figure 6.2304



Figure 6.2305



6.28.21 SmParametrize

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Creates a consistent set of 3D constraints for a selected sheet metal part. Recognizes rectangular array of holes on flanges. The command takes into account implicit constraints from the sheet metal features and ensures that the constraint system is not overdefined. Please note that for the best results, the sheet metal part should have a complete set of features, including flanges, bends, junctions and bend and corner reliefs. Editing the constraints allows to obtain a series of similar sheet metal parts with different dimensions. Example of a fully parametrized sheet metal part (left) and different sets of parameters applied (center and right): (\vref {fig:C:/Dev/Task-2448/images/staticbricsysc

Accessing the Command



command bar: **smparametrize** menu bar: **Sheet Metal | Parametrize** toolbar: **Sheet Metal | Parametrize** ribbon: **Sheet Metal | 3D Constraints | Parametrize** Quad cursor menu:**Sheet Metal | Parametrize : SmParametrize Prompts you in the command bar:** Select 3D solids to parametrize or [Entire model] <Entire model>: **(Select a sheet metal part or press Enter to parametrize all sheet metal parts in the drawing.)** Created 6 distance constraints Created 3 fix constraints Created 3 coincident constraints Created 0 tangent constraints Created 0 rigid set constraints Total: 12 constraints and 0 array(s) created The constraints can be edited in the Mechanical Browser (3.2.2) panel.

Command Options

Option	Description
Settings	Hole parametrization. • Parametrize holes : Toggles creation rigid sets constraints for internal holes on flanges. • Convert holes to array : Toggles check possibility to group holes into parametric rectangular array. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee

Related Commands

MechanicalBrowserOpen (6.8.29) - opens the Mechanical Browser panel. SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmExtrude (6.28.9) - creates sheet metal part from a planar polyline by extruding it orthogonally to the plane of the polyline. SmFlangeBase (6.28.10) - creates a base (initial) flange of a sheet metal part from a closed 2D entity. SmFlangeConnect (6.28.12) - closes gaps between two arbitrarily oriented flanges. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange. SmFlip (6.28.15) - Swaps sides of a selected flange so that reference faces are on the other geometric side of the flange. SmReliefCreate (6.28.22) - makes proper corner and bend reliefs.

Figure 6.2306

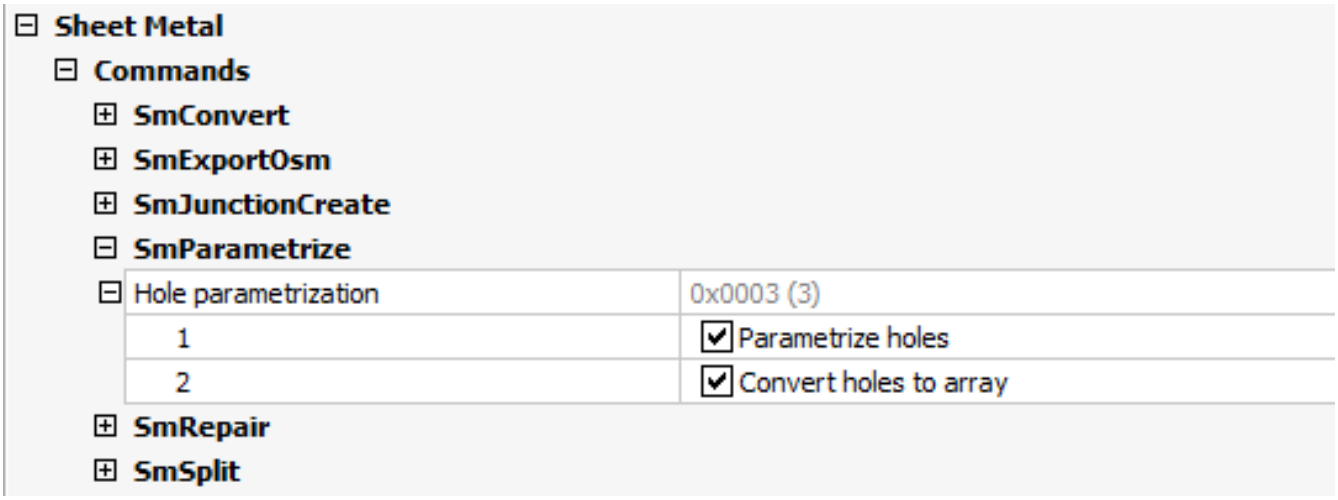


Figure 6.2307



6.28.22 SmReliefCreate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Makes proper corner and bend reliefs.Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built at the start and end of a flange edge. Command supports the creation of reliefs on wrong bends. The command automatically decides whether a relief has to be created for given bend. The size is either automatically chosen by default from theMechanical Browser for Sheet Metal (3.8.5) in the drawing or can be entered by user. In order to force the creation of a bend relief for the given bend, the user has to select the face of desired bend. In this case a bend relief will be created, even if the command decides a bend relief is not needed (i.e. it detects there are no clashes or stretches of the material).

Accessing the Command

 command bar: **smreliefcreate** menu bar: **Sheet Metal | Create Relief** toolbar: **Sheet Metal | Create Relief** ribbon: **Sheet Metal | Modify | Create Relief** Quad cursor menu: **Sheet Metal | Create Relief : SmReliefCreate Prompts you in the command bar:** Select a hard edge, bend face, flange face or 3D solid [Entire model]: <Entire model>: **(Select an item.)** Enter relief size by bend radius ratio or [force Bend reliefs/Auto] <Auto>: **(Type a value or choose an option.)**

Command Options

Option	Description
Entire model	Creates reliefs on all hard edges, bends, and corners in the entire model.
Force bend reliefs	**Force bend reliefs [ON/OFF/Back] <Back>:** • **ON**: Forces bend relief creation on bends. Bend reliefs will be created on any bend that does not have orthogonal side faces. • **OFF**: Bend reliefs on bends will be created only when a torn edge is found. • B**ack**: Returns to the default command prompt
Auto	The command automatically detects bend radii and adjusts the relief size accordingly.

Related Commands

SmBendCreate (6.28.2) - converts hard edges (sharp edges between flange faces) and junctions into bends. SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmDissolve (6.28.6) - removes sheet metal data from the selected faces or entities. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange. SmJunctionCreate (6.28.18) - converts hard edges (sharp edges between flange faces) and bends into junctions. SmReliefSwitch (6.28.23) - converts corner reliefs to circular, rectangular or V-type reliefs. Converts bend reliefs to smooth, round, rip or rectangular bend reliefs. Allows to change the parameters of existing corner reliefs. SmSplit (6.28.28) - splits a flange along a line drawn on its face.

Figure 6.2308



6.28.23 SmReliefSwitch

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Converts corner reliefs to circular, rectangular or V-type reliefs. Converts bend reliefs to smooth, round, rip or rectangular bend reliefs. Allows to change the parameters of existing corner reliefs.

Accessing the Command

 ===ERROR WITH DOWNLOADED IMAGE===  command bar: **smreliefswitch** menu bar: **Sheet Metal | Relief | Switch to ...** toolbar: **Sheet Metal | Relief | Switch to ...** ribbon: **Sheet Metal | Modify | Switch to ...** Quad cursor menu:**Sheet Metal | Switch to ... : SmReliefSwitch Prompts you in the command bar:** Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/Entire model] <Entire model>: **(Select a relief face, bend face, flange face or a 3D Solid or press Enter to apply the V-type relief to the entire model; or choose an option.)** Entities in set: 1 Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/Entire model] <Entire model>: **(Select a relief face, bend face, flange face or a 3D Solid or choose an option or press Enter.)**

Command Options

Option	Description
Rectangular  (Corner)	Switches the selected corner reliefs to rectangular. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S Prompts you: **Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/Entire model] <Entire model>:- (Do the following:)** • (option) Choose relief extension: You are prompted: **Enter the extension [Auto] <Auto>:- (Do one of the following:)** • Type a value • Select Auto to keep the current extension. • Select the reliefs to be switched or press Enter to select the entire model.
Rectangular  (Bend)	Switches the selected bend reliefs to rectangular: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S

Circular 

Switches the selected corner reliefs to circular. Only corner reliefs which are related to exactly two bends can be switched to circular reliefs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
Prompts you: **Select faces, 3d solids to switch reliefs to circular [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/relief Diameter/REference point/Entire model] <Entire model>:** - **(Do the following:)**

- (option) Choose relief Extension :sets the relief extension. Prompts you:**Enter the extension [Auto] <Auto>:** - **(Type a value or select Auto to keep the current extension of the selected reliefs.)**
- (option) Choose relief Diameter :sets the relief diameter. Prompts you:**Enter the desired diameter [Auto] <Auto>: - (Type a value or select Auto to use the minimal diameter.)**
- (option) Choose Reference point :sets the reference for the relief. Prompts you: **Select the reference of the relief [Middle/Corner/from Global settings] <from Global settings>:**
  - Middle : the intersection of the bend lines.
  - Corner : the circular arc through the base points of the corresponding rectangular reliefs.
  - From global settings : Take value from global sheet metal settings of document.
- Select the reliefs to be switched or press Enter to select the entire model.

V-type 

Switches the selected corner reliefs to V-type. Only corner reliefs which are related to exactly two bends can be switched to v-type. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/Entire model] <Entire model>:- **(Do the following:)**

- (option) Choose relief extension: You are prompted: **Enter the extension [Auto] <Auto>:** - **(Do one of the following:)**
- Type a value
- Select Auto to keep the current extension.
- Select the reliefs to be switched or press Enter to select the entire model.

Smooth 	Switches the selected bend reliefs to smooth. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S Prompts you: **Select faces, 3d solids to switch reliefs to smooth [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/Entire model] <Entire model>:- (Select the bend reliefs to be switched or press Enter to select the entire model.) Smooth** bend relief can be applied only if the current relief type is Rectangular. If not, BricsCAD responds: Unable to build relief geometry for this configuration.
Rip 	Switches the selected bend reliefs to rip. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S **Select faces, 3d solids to switch reliefs to rip [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/Entire model] <Entire model>:- (Select the bend reliefs to be switched or press Enter to select the entire model.) Rip** bend relief can be applied only if the current relief type is Rectangular. If not, BricsCAD responds: Unable to build relief geometry for this configuration.
Round 	Switches the selected bend reliefs to round. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S **Select face(s), 3d solid(s) to switch relief(s) to round [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/Entire model] <Entire model>:- (Select the bend reliefs to be switched or press Enter to select the entire model.) Round** bend relief can be applied only if the current relief type is Rectangular. If not, BricsCAD responds: Unable to build relief geometry for this configuration.
Entire Model	All corner or bend reliefs in all sheet metal parts in the model will be switched to the specified type using the specified parameters. If this option is chosen as the response to the initial prompt, all corner reliefs switch to V-type.

Relief extension illustration:

rectangular and v-type reliefs: extension (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou
• circular reliefs: minimal extension, diameter of the relief in the unfolded representation, reference to determine the center of the relief in the unfolded representation (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cmd
• V-type reliefs: extension (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SmReliefS typeParameters.png})

Figure 6.2309

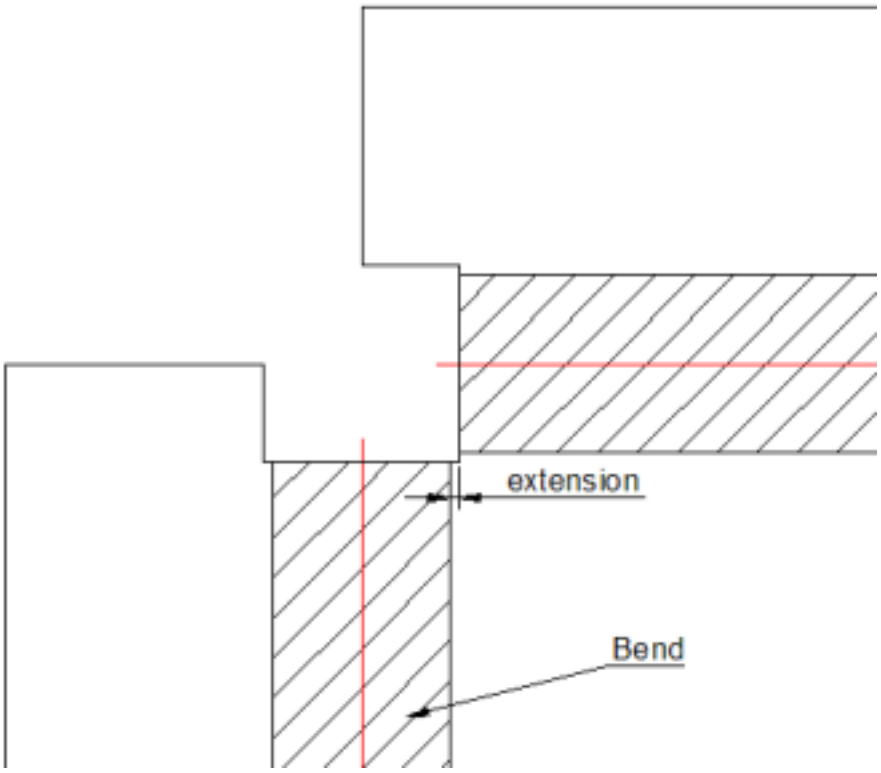


Figure 6.2310

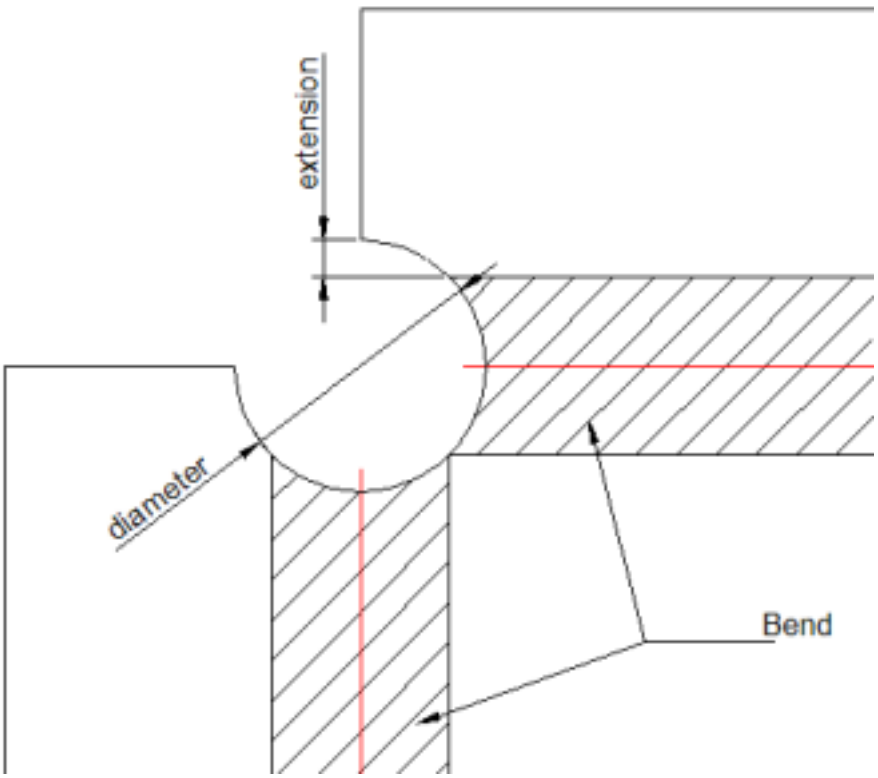
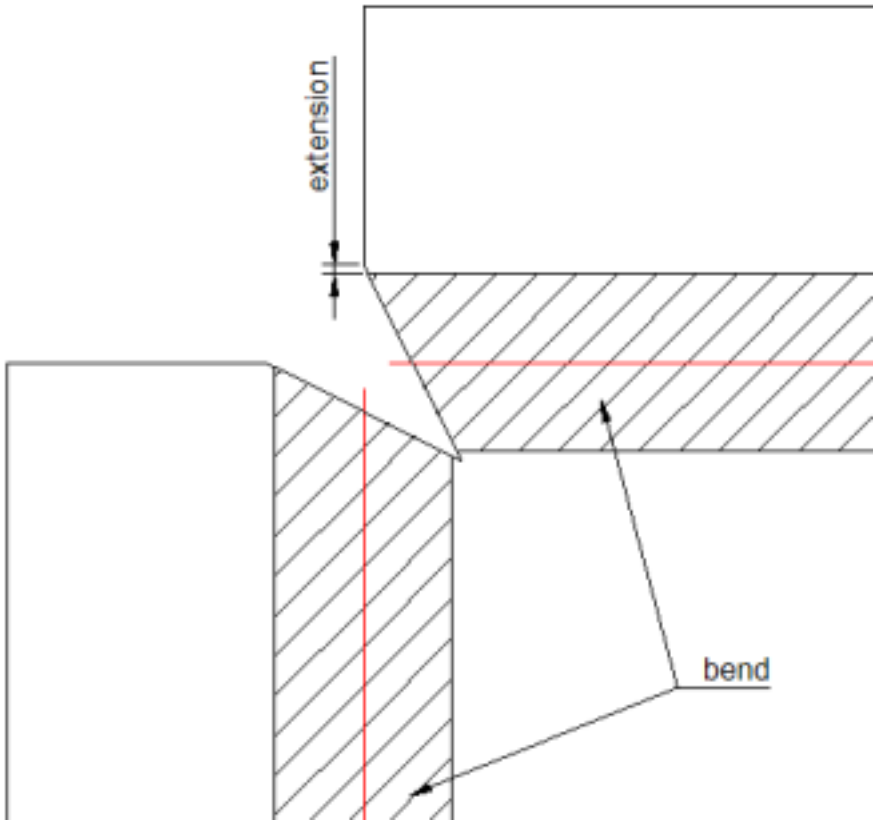


Figure 6.2311



Procedures

Switching relief type (3.8.13)

Related Commands

SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange. SmJunctionSwitch (6.28.19) - allows changing a symmetrical junction feature to one with overlapping faces. SmReliefCreate (6.28.22) - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built at the start and end of a flange edge.

Figure 6.2312

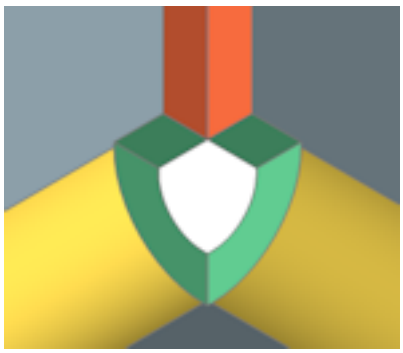


Figure 6.2313



Figure 6.2314

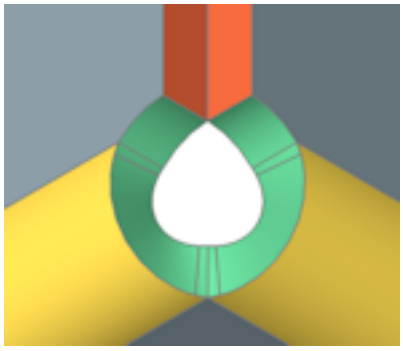


Figure 6.2315

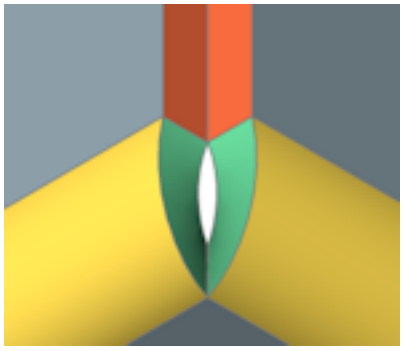


Figure 6.2316



Figure 6.2317

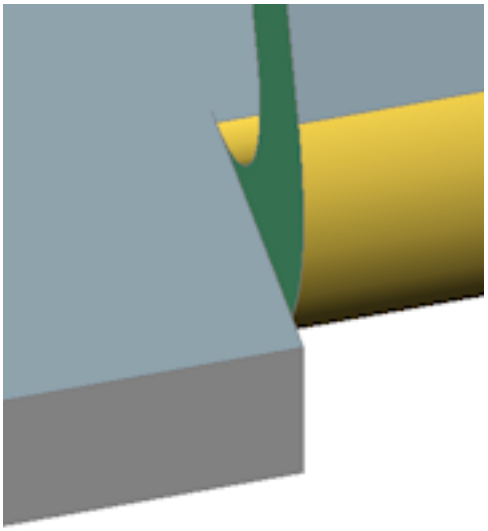


Figure 6.2318

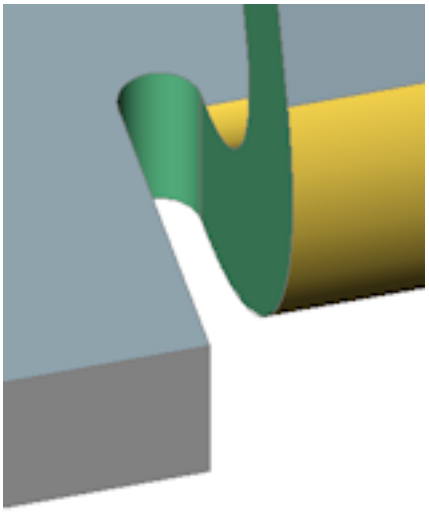


Figure 6.2319



6.28.24 SmRepair

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Restores the 3D solid model of a sheet metal part by thickening one of its sides: all thickness faces become perpendicular to flange faces. Fixes problems specific to parts with lofted bend features: merges adjacent bends and provides tangential connection with flanges. The command is able to convert wrong bends to regular bends. If a single wrong bend, it will be repaired. If a 3D solid is selected, then all the wrong bends will be converted to regular bends.

Accessing the Command


command bar: **smrepair** menu bar: **Sheet Metal | Repair Sheet Metal** toolbar: **Sheet Metal | Repair Sheet Metal** ribbon: **Sheet Metal | Heal | Repair Sheet Metal** Quad cursormenu:**Sheet Metal | Repair Sheet Metal : SmRepair Prompts you in the command bar:** Select flange face or lofted bend face to repair or [switch on Local repairing/Scan for lost details]: Switch on Local repairing: Select one wrong bend face, thickness faces, flange faces or side edges to repair or [switch on Global repairing]: Scan for lost details: Enter scan tolerance <0.01>: **(Press Enter or specify tolerance value.)** A body with thickness faces perpendicular to the selected face and without adjacent lofted bends will be created. Notice that the size and position of the opening on the selected face is preserved; the size and position of the opening in the opposite face is adjusted. **Global repair** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/smrepair1.png}) (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/smrepair2.png}) before after **Local repair (flange)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/smrepair3.png} (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/smrepair4.png}) before (flange face selected) after **Local repair (wrong bend)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/smrepair6.png}) before (wrong bend face selected) after

Command Options

Option	Description
Scan for lost details	Searches and retains as separated bodies model parts which remained after repairing. Prompts you: **Enter scan tolerance <0.01>:** - **(Type a value or press enter to accept the default value.)** Scan tolerance: The fraction of the initial model volume, which will be ignored, only larger bodies will be retained. E.g. if the scan tolerance = 0.25, only parts with a volume that is larger than 25% of the initial body will be retained. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S From left to right: • Initial model with unrecognized form feature • Model after SmRepair without scanning for lost details: the circular feature is removed. • Model after SmRepair with scanning. Two bodies are created: a horizontal flange with a hole and a separate body (highlighted in green color). Bodies can be unified using the **Union** command if required.

Settings	Merge lofted bends: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee If turned on, adjacent lofted bends with tangencial connection will be merged to single lofted bend. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S From left to right: • Initial model with two lofted bend features • Model after SmRepair without lofted bend repair: the hole became perpendicular to the features faces. • Model after SmRepair with lofted bend repair: two lofted bend features are united to a single one.

Procedures

To repair a sheet metal body (3.8.14)

Related Commands

SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmDissolve (6.28.6) - removes sheet metal data from the selected faces or entities.

Figure 6.2320

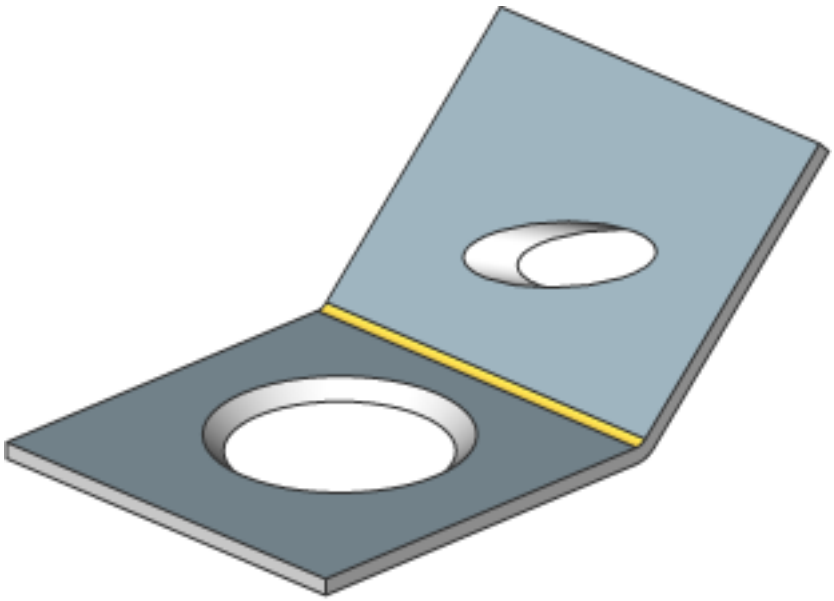


Figure 6.2321

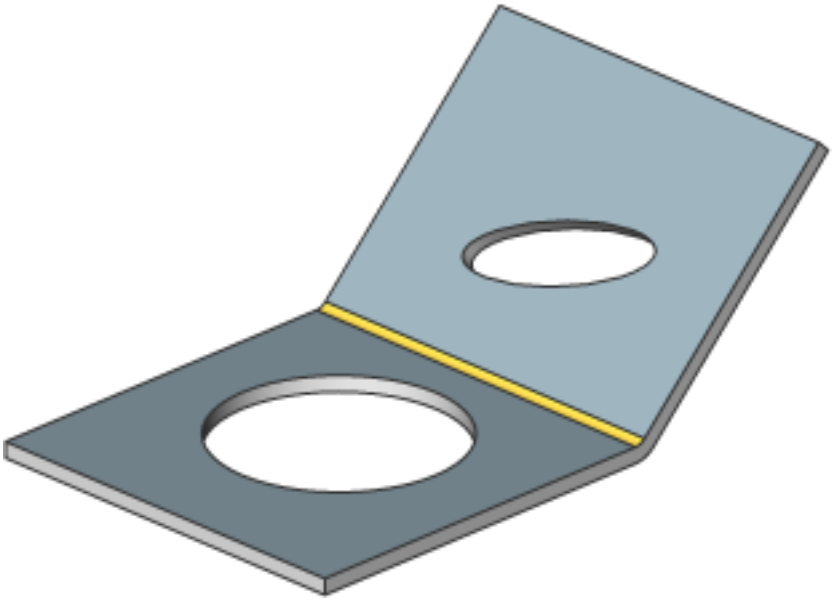


Figure 6.2322

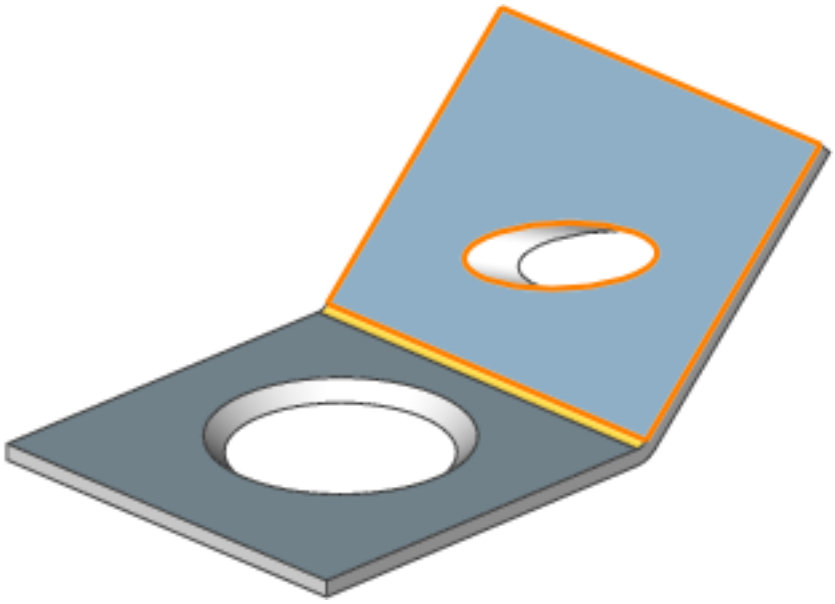


Figure 6.2323

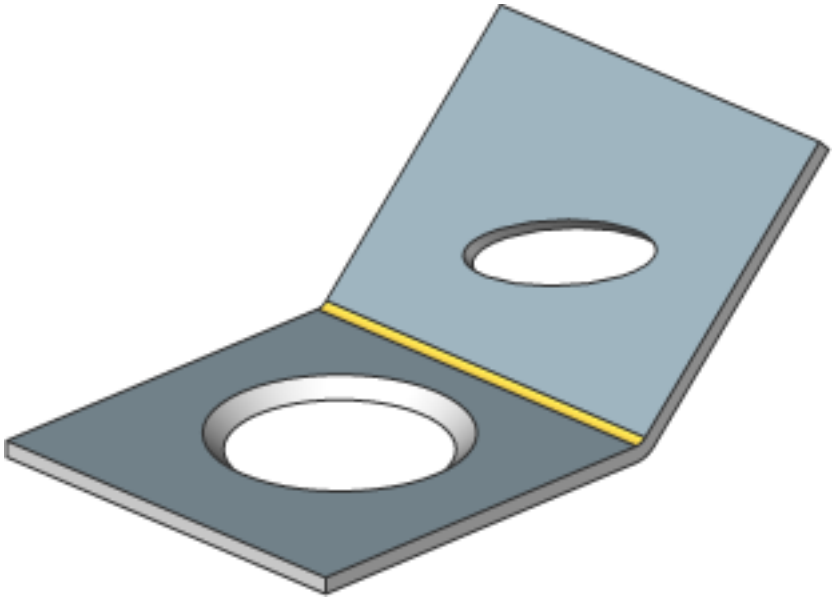


Figure 6.2324

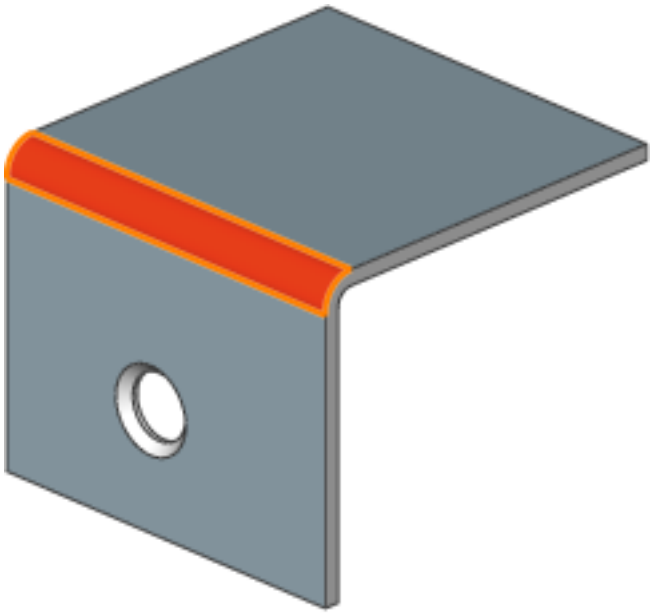


Figure 6.2325

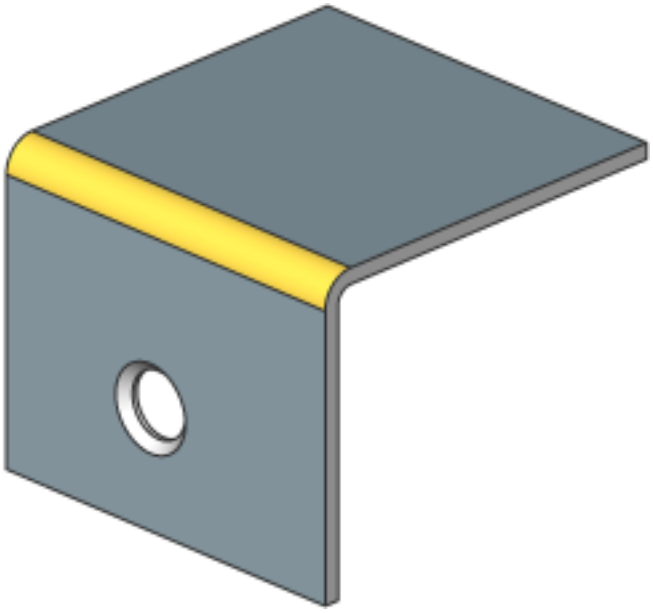


Figure 6.2326

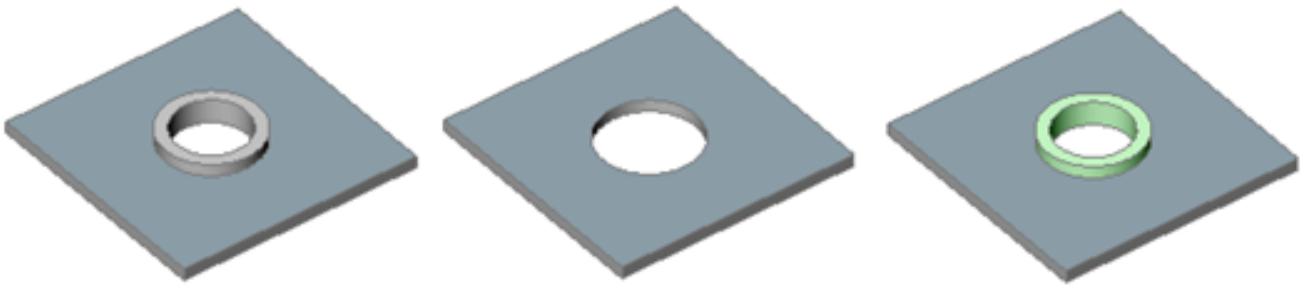


Figure 6.2327

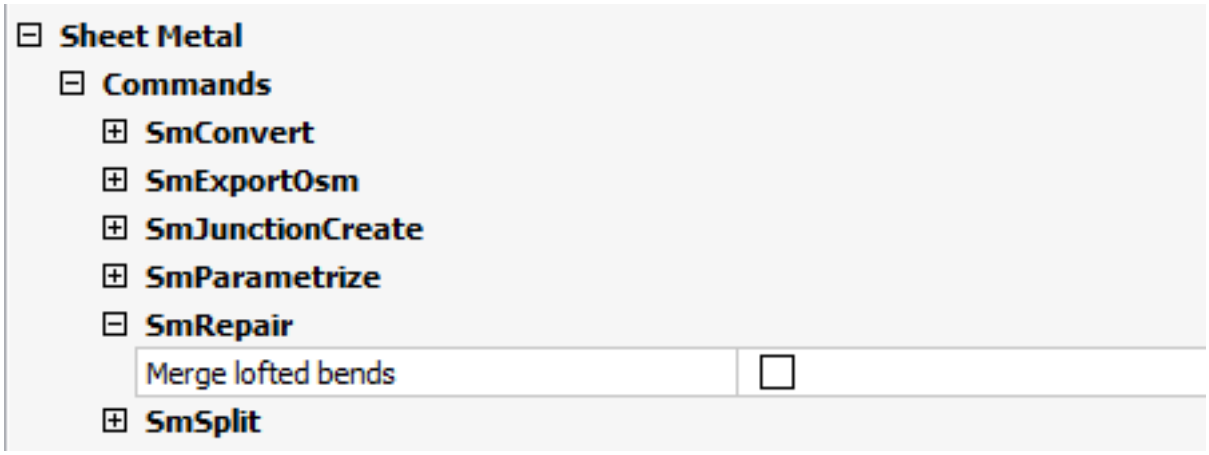


Figure 6.2328

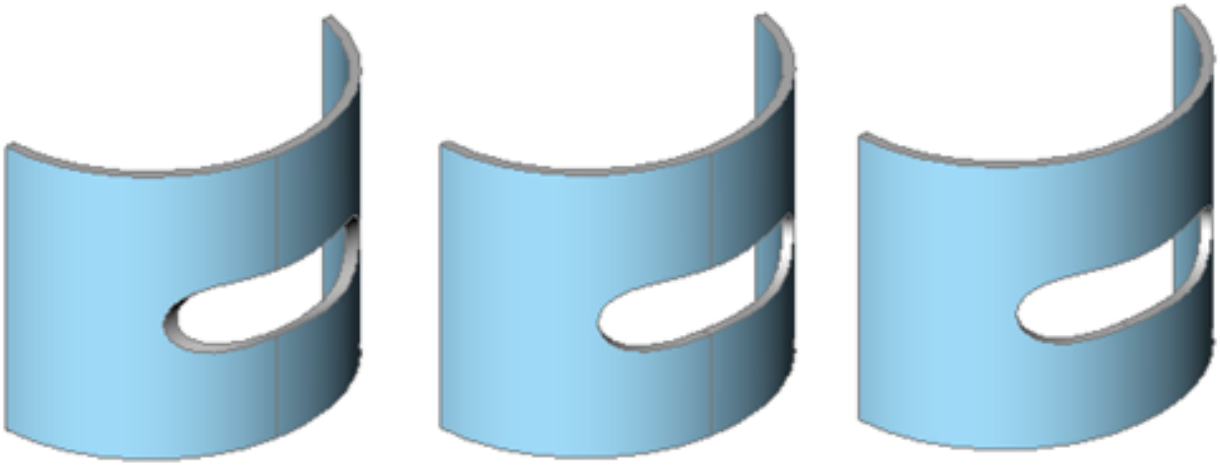
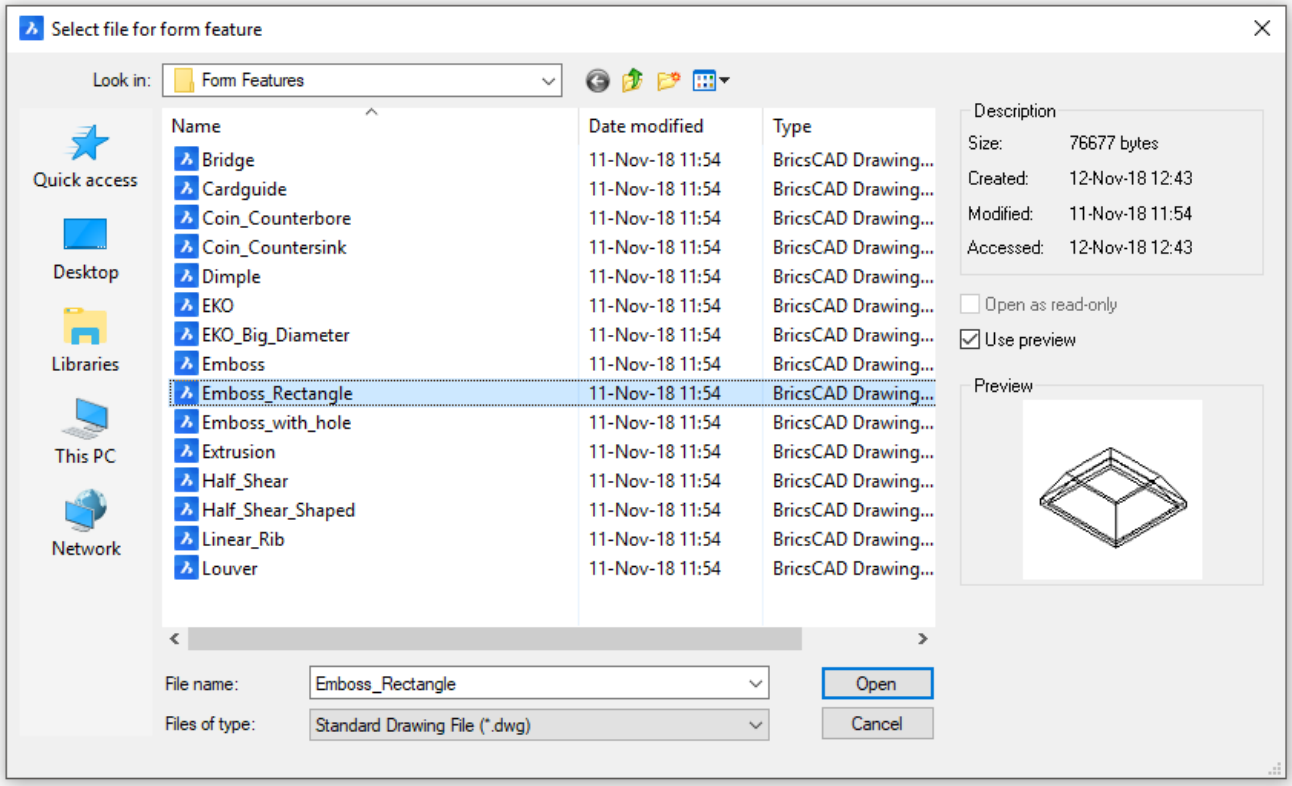


Figure 6.2329



Figure 6.2330



6.28.25 SmReplace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Replaces form features (including recognized ones) in sheet metal parts with form features from the built-in or user library. Accessing the Command  command bar: **smreplace** menu bar: **SheetMetal | Form | Replace Form Features** toolbar: **Sheet Metal | Replace Form Features** ribbon: **Sheet Metal | Modify | Replace Form Features** Quad cursor menu: **Sheet Metal | Replace Form Features : SmReplace Prompts you in the command bar:** Select several entities/subentities: **(Select a form feature.)** Entities in set: 1 Select several entities/subentities: **(Select more form features or press Enter to stop selecting.)** A dialog box displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/smreplacedialog.png}) **(Select a form feature, then press the Open button.) Prompts you:** Replace all similar form features [Yes/No/parametersMode] <Yes>: **(Press Enter to replace all similar form features in the model or select an option.)**

Command Options

Option	Description
Yes	Replaces all form features of the same type as selected feature(s).
No	Repaces the selected form feature(s) only.

Parameters Mode	Allows to control whether parameter values are inherited from the form features being replaced or not. Prompts you: **Select mode of replacement [component Defaults/Inherit from replaced feature/Edit parameters] <Inherit from replaced feature>:** • Component Defaults: Parameter values are copied anyway. • Inherit from Replaced Feature: If possible parameter values are copied from the form feature(s) being replaced. • Edit parameters: Allows to edit all parameters. Prompts you: **Enter parameter name or press Enter to finish** - **(Type a parameter name.)** **Enter Expression** - **(Enter an expression.)**

Related Commands

BmInsert (6.8.12) - inserts an existing mechanical component or sheet metal form feature into the current drawing.

Figure 6.2331



6.28.26 SmRibCreate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Creates associative Rib form features on sheet metal parts from 2D profiles. The feature is updated automatically when its defining profile is changed. The difference with linear rib features inserted from the library byBmInsert (6.8.12) is that rib features created by **SmRibCreate** can have an arbitrary trajectory.

Accessing the Command

 command bar: **smribcreate** menu bar: **Sheet Metal | Form | Create Rib Feature** toolbar: **Sheet Metal | Create Rib Feature** ribbon: **Sheet Metal | Modify | Create Rib Feature : SmRibCreate Prompts you in the command bar:** Select flange face: **(Select a flange face.)** Select control curve [Profile radius/Round radius]: (Select a curve entity or choose an option.) Entities in set: 1 Select curve or [Profile radius/Round radius]: **(Select more curve entities or press Enter.)**

Command Options

Option	Description
Select a curve	Select one or more 2D entities: lines, polylines, circles, arcs, ellipses, elliptical arcs, and splines are accepted. The 2D curves can be open or closed but cannot intersect itself or other form features.
Profile Radius	Sets the radius of the rib profile (blue face in the image below). The initial value of the Profile Radius is taken from the Mechanical Browser for Sheet Metal (3.8.5). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee
Round Radius	Sets the radius of the smoothing fillets (green faces in the image below). The initial value of the Round Radius is taken from the Mechanical Browser for Sheet Metal (3.8.5). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Shee

Related Commands

BmInsert (6.8.12) - inserts an existing mechanical component or sheet metal form feature into the current drawing.
SmConvert (6.28.4) - automatically recognizes flanges and bends in a 3D solid.

Figure 6.2332

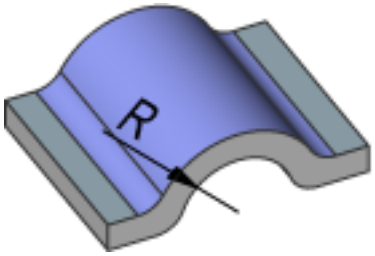


Figure 6.2333

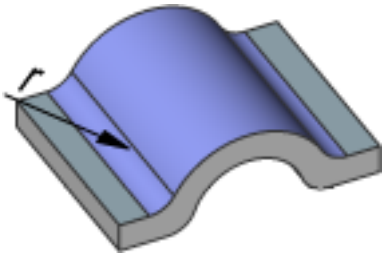


Figure 6.2334



6.28.27 SmSelect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) This command is oriented to work on solids with sheet metal features. Selects hard edges, same form features, non-orthogonal thickness edges and side of the sheet metal part.

Accessing the Command

 **Same Form Features**  **Hard Edges** command bar: **smselect** menu bar: **Sheet Metal | Select Same Form Features**, **SelectHard Edges and SelectNon-orthogonal thickness edges** toolbar: **Sheet Metal | Select Same Form Features**, **SelectHard Edges and SelectNon-orthogonal thickness edges** ribbon: **Sheet Metal | Select Same Form Features**, **SelectHard Edges and SelectNon-orthogonal thickness edges** Quad cursor menu:**Select | Select Same Form Features**, **SelectHard Edges and SelectNon-orthogonal thickness edges : SmSelect Prompts you:** Select the option [Hard edges/Same form features/SImilar form features/Non-orthogonal thickness faces/SIDe of sheet metal part]:

Command Options

Option	Description
 Hard edges	Selects all hard edges in the model.
 Same form features	Selects all form features which are identical to the selected feature(s). Prompts you: **Select form feature faces** - **(select a form feature face.)** Entities in set: 1 **Select form feature faces** - **(select more form feature faces or press Enter.)**
Similar form features	Selects all form features which are similar (= same type, but different parameter values) to the selected feature(s). Prompts you: **Select form feature faces** - **(select a form feature face.)** Entities in set: 1 **Select form feature faces** - **(select more form feature faces or press Enter.)**
Non-orthogonal thickness faces	Selects all edges of specified flange common with non-orthogonal thickness faces.
Side of sheet metal part	Requires flange, regular or lofted bend face to start. Opion will select faces whih are on same side of sheet metal part within selected face.

Related Commands

DmSelect (6.12.26) - selects edges and faces of 3D solids or surfaces based on their geometric properties. DmSelectEdges (6.12.25) - selects the edges of faces and solids. SmBendCreate (6.28.2) - converts hard edges (sharp edges between flange faces) and junctions into bends. SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmJunctionCreate (6.28.18) - converts hard edges (sharp edges between flange faces) and bends into junctions.

Figure 6.2335



6.28.28 SmSplit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Splits a flange along a polyline drawn on its face; splits lofted bend along the line on the ruled surface.

Accessing the Command

 **Split**  **Smart Split** command bar: **smsplit** menu bar: **Sheet Metal | Split | Split** and **Smart Split** toolbar: **Sheet Metal | Split** and**Smart Split** ribbon: **Sheet Metal | Modify | Split** and **Smart Split** Quad cursor menu: **Sheet Metal | Smart Split : SmSplit Prompts you in the command bar:** Select flange or lofted bend face:**(Select the face of flange or lofted bend.) Flange:** Select lines or edges to split the flange or [use SMart split/draw a New line] <draw a New line>: **(Select a polyline or edge on the flange or choose an option.) Lofted bend:**Select point on lofted bend face: **(Select a point on a ruled surface)**

Command Options

Option Input Description Flange face Allows to split flange by drawing a new line or using existent line/polyline. prompts you:**Select flange or lofted bend face:** - **(Select a flange) Select lines or edges to split the flange or [use SMart split/draw a New line] <draw a New line>**: - **(choose an option)** prompts you: **Make split [Center/Left/Right/Accept] <Accept>:** -**(Repeatedly hit the Ctrl-key to cycle the options, then press Enter to accept the current split.)** Lofted bend face Allows to split the lofted bend by selecting a single point on a lofted bend face. Prompts you: **Select flange or lofted bend face:** - **(Select a lofted bend face) Select point on lofted bend face:**

- Select point for split: Specify a point on selected lofted bend.

Use Smart Split
Flange face Allows to split flange specifying a point on a flange. prompts you:**Select flange or lofted bend face: -(Select a flange) Select lines or edges to split the flange or [use SMart split/draw a New line] <draw a New line>:** - **(choose an option:)** Use smart split Allows to split the flange by selecting a single point at a flange corner. Variants for placing a second point will be automatically suggested; prompts you: **Select point for split:** - **(Snap to a vertex of the selected flange.)** prompts you: **Make split [Center/Left/Right/Accept] <Accept>:** - **(Repeatedly hit the Ctrl-key to cycle the options, then press Enter to accept the current split.)** If TIPS (1.12.2) = ON, the currently selected option is indicated in the Tips widget at the bottom of the screen. New Line Allows to draw a line on the selected flange. Prompts you: **Start point of the line:** - **(Specify the start point of the line.) End point of the line:** - (**Specify the end point of the line.)** Settings (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/smsplitdialog.png})

- Convert bend to junction: Allows to recognize an L-corner on a flange and automatically convert part of the split bend into a junction.
- Make orthogonal bend split: Determines whether a bend will be split orthogonally to the bend axis if the bend line crosses the bend at an angle.
- Heal coincident miter faces: Switches the mode for the recognition of miter-like design made of coincident faces; OFF = keep; ON = convert to regular miters.

Related Commands

SmConvert (6.28.4) - automatically recognizes flanges, bends and lofted bends in a 3D solid. SmDelete (6.28.5) - removes a bend or a junction by restoring a hard edge between two flanges; removes a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Removes miter by restoring geometry being cut by the feature. SmFlangeConnect (6.28.12) - closes gaps between two arbitrarily oriented flanges. SmFlangeEdge (6.28.13) - creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange. SmReliefCreate (6.28.22) - makes proper corner and bend reliefs. Corner reliefs are built on corners which have three or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.

Figure 6.2336

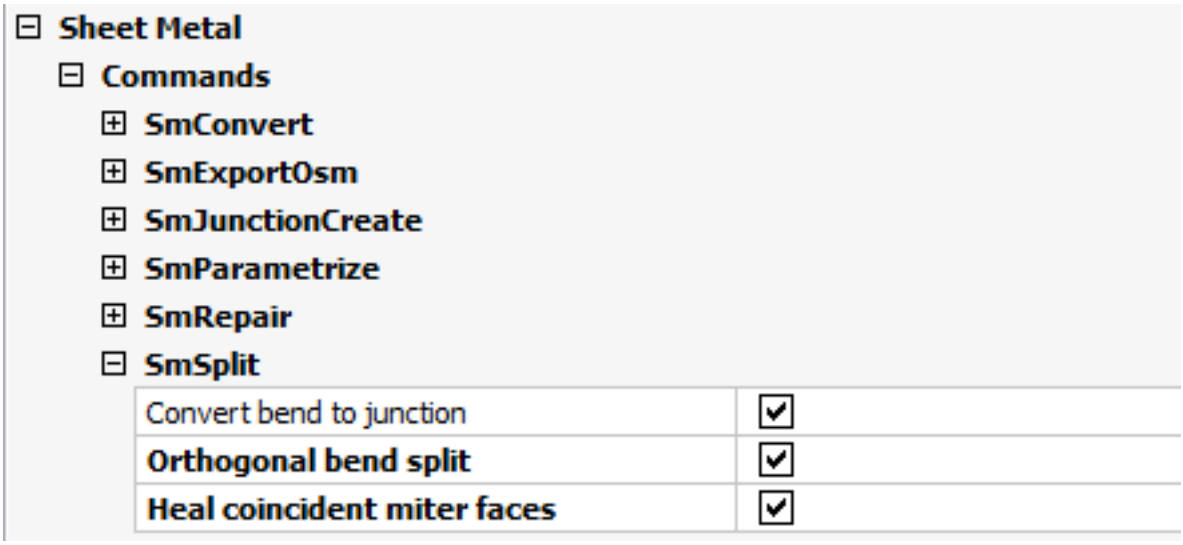


Figure 6.2337



6.28.29 SmUnfold

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-mechanical.png}) Creates an unfolded 2D or 3D representation of a sheet metal part.

Accessing the Command

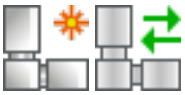
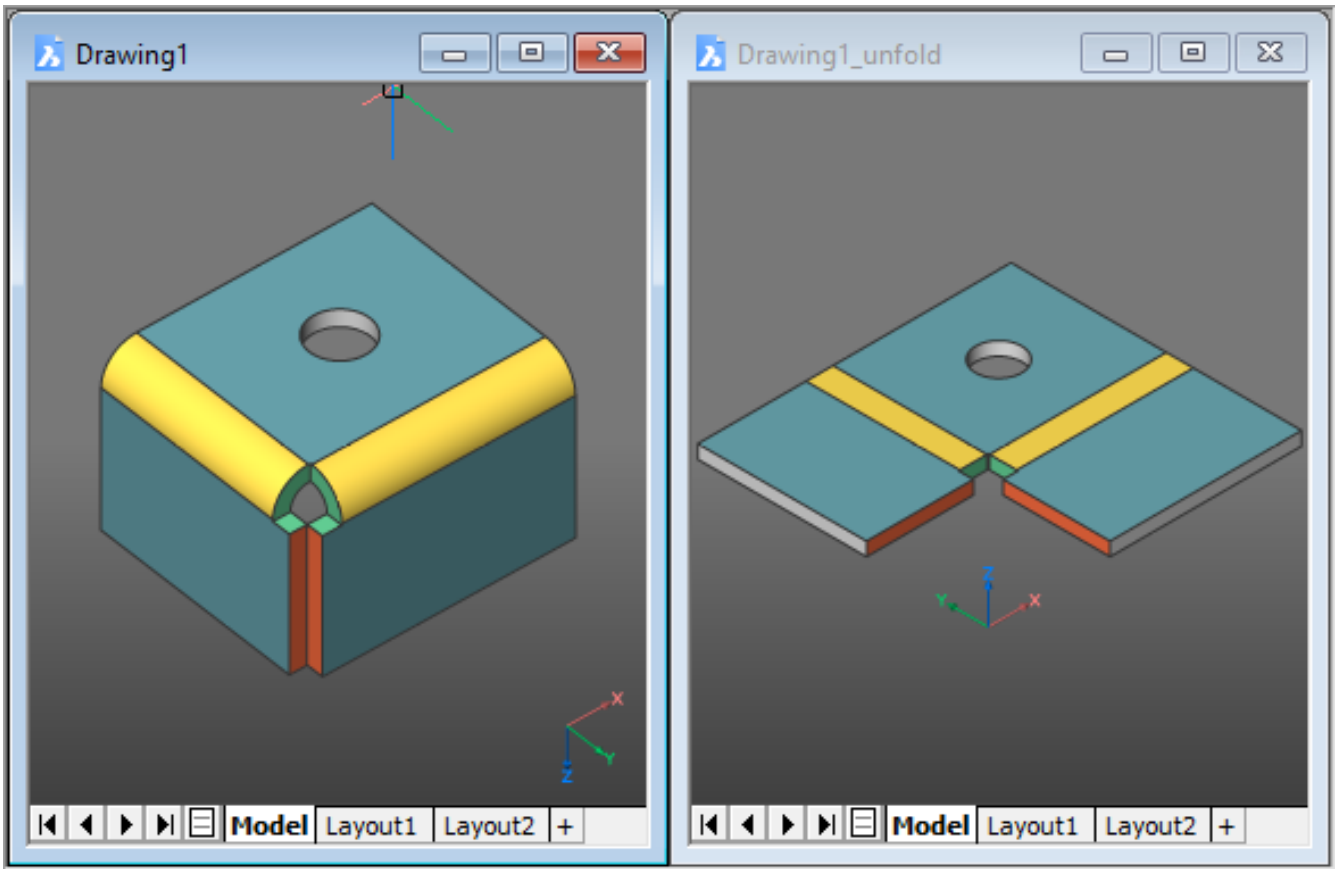

command bar: **smunfold** menu bar: **Sheet Metal | Unfold | Unfold Body** and **Associatively Unfold Body** toolbar: **Sheet Metal | Unfold Body** and **Associatively Unfold Body** ribbon: **Sheet Metal | Flatten | Unfold Body**and **Associatively Unfold Body** Quad cursor menu: **Sheet Metal | Unfold Body : SmUnfold Prompts you in the command bar:** Select a flange or lofted bend face to start unfolding [Associative]: **(Select the base flange of the sheet metal part.)** Select position of the unfolded body: **(Specify a point in the drawing.)** Validate the unfolded body and select an option [save 2D geometry/save 3D geometry/Optimize bend annotations/Keep] <Keep>: **(Press Enter to keep the unfolded body in the drawing or choose an option.) Associative mode:** opens an additional window with model's unfolded representation. Both folded (left) and unfolded(right) representations have bidirectional associativity which allows transferring modifications to neighbor model automatically when changing window focus. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/SheetMetal/smunfold1.png})

Figure 6.2338



Command Options

Option	Description
Keep	Places the unfolded 3D solid in the current drawing.
Save 2D geometry	Saves the 2D geometry of the unfolded metal part as a separate drawing file. Displays a Save File dialog window. • Select a folder. • Type a name in the **File Name** field. • Select either **Drawing Exchange Format (*.dxf)** or **Standard Drawing File (*.dwg)** from the **Save as type** list button. • Click the **Save** button to create the file.
Save 3D geometry	Saves the 3D solid of the unfolded metal part as a separate drawing file. Displays a Save File dialog window. • Select a folder. • Type a name in the **File Name** field. • Select either **Drawing Exchange Format (*.dxf)** or **Standard Drawing File (*.dwg)** from the **Save as type** list button. • Click the **Save** button to create the file.
Optimize Bend Annotations	Optimizes the bend annotations in the exported geometry.

Procedures

To unfold the sheet metal body (3.5.9) To unfold the sheet metal body in associative mode (3.8.17)

Related Commands

6.29 T

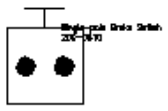
6.29.1 Table

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a table entity in the drawing through a dialog box, either from scratch or from data in an external file or a data link. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/eg-withblocks-1.png}) **Note:** To place blocks in cells, use the TableMod (6.29.5)command.

Figure 6.2339



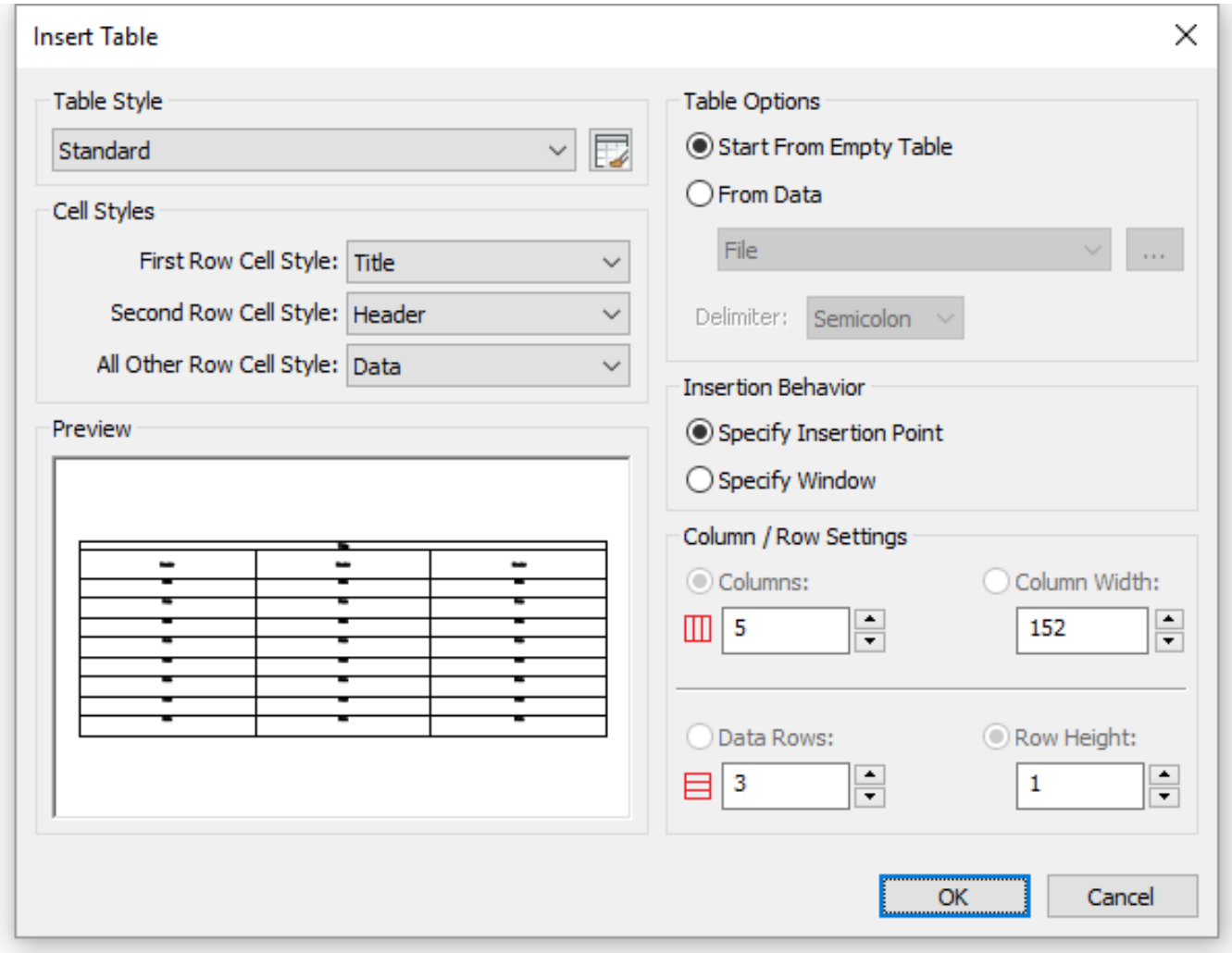
Figure 6.2340

Automotive Electrical System - Bill of Materials					
Component	Description	Stock No	Total	Price	Part
BSWITCH	Single-pole Brak	200-0510	1	6.78	
DOMESW	Single-pole Dome	200-0020	2	2.54	
FUSE40	40 Amp Fuse	300-0040	1	1.23	

Accessing the Command

 command bar: **table** menu bar: **Draw | Table** ribbon: **Draw | Tables | Table toolbar: Draw |**  **: Table** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/InsertTableDialog Choose options, and then clickOK .

Figure 6.2341



Dialog Box Options

Option	Description
Table Style	Chooses an existing style or else creates a new one: • Click the droplist, then select an existing table style. • Click **Table Styles Explorer** to access the Table section of the Drawing Explorer and create or edit table styles; see the TableStyle (6.29.6) command.
Cell Styles	A table style can have different sub-styles for the first two rows; these sub-styles are named Title, Header, and Data: • **First Row Cell Style** - choose a sub-style from the droplist, which by default is Title; a title row is the topmost row of the table • **Second Row Cell Style** -choose a sub-style from the droplist, which by default is Header; a header row is the first row of a column • **All Other Row Cell Styles** -choose a sub-style from the droplist, which by default is Cell; a cell is all other rows in the table The properties of the Header, Border, and Cell sub-styles are defined by the table style.
Table Options	Determines the source of the new table: **Start From Empty Table** - creates a new, empty table with the parameters listed below **From Data** - reads data from a file or a datalink, and then constructs a table to fit the data; click the droplist to chose the source of the data: • - click Browse and then from the file dialog box choose a data files in CSV (comma separate value) or a spreadsheet file in XLS, XLSX, and XLM format • **DataLink** - displays the DataLink Manager dialog box to choose a named data link; see the DataLink (6.10.2) command (Available only with the File option) **Delimiter** - specifies the delimiter character used to separate fields in CSV data files; click the droplist and then choose a delimiter: • **Semicolon**- ; • **Comma**- , • **Space**- • **Tab**-

Insertion Behavior	**Specify Insertion Point** - specifies where the upper-left corner of the table will be inserted in the drawing; you will be prompted for the location after you click **OK**to close the dialog box. **Specify Window** - specifies the area that the table should consume in the drawing; this is useful when you need the table to fit a specific area, such as in a revision table or title block. After you clickOK, the program will prompt you for the two corners of the window: 1. **Specify first corner**- **(enter a point for top left corner of the table.)** 2. **Specify second corner**- **(enter a point for the bottom right corner of the table.)** As you move the mouse, a ghosted table dynamically resizes itself.
Columns / Rows Settings	(Available only with the Specify Insertion Point option) Specifies the number of columns or rows and the cell width or height. Choose one of • **Columns** - specifies the number of columns in the table • **Column Width** - specifies the initial width of all columns And choose one of • **Data Rows**- specifies the number of data rows in the table; the title and header rows are added automatically by the program • **Row Height**- specifies the initial height of all rows The number of columns and rows, and size of cells can be adjusted later, after the table is inserted.
OK	Closes the dialog box, and then prompts you to inserts the table in the drawing. The TablEdit (6.29.3) command immediately launches so that you can begin filling in the cells.

Procedures

- To create an empty table (1.21.25)
- To create a table from data (1.21.25)
- To create a table from an Excel spreadsheet (1.21.25)
- General procedure to create fields (1.21.23)

Related Commands

DataLink (6.10.2) -Links data from an Excel spreadsheet directly into drawing tables for bi-directional updating. -Table (6.29.2) - creates a table entity in the drawing through the command bar. Tablemod (6.29.5) - modifies the properties of a selection of table cells. Tabledit (6.29.3) - edits the cell content. TableExport (6.29.4) - exports the content of a table entity in CSV file format. TableStyle (6.29.6) - creates and edits table styles through the Drawing Explorer - Table Styles dialog box. Field (6.14.3) - inserts a field in a multiline text, table cell or attribute.

Figure 6.2342



Figure 6.2343

Table Title Style		
Header Style	**Header Style**	**Header Style**
.15	Row Style	1.0
.26	ACD	2.0
.37	BEF	3.4
.48	CGH	4.5
.59	DIJ	5.6

6.29.2 -Table

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a table entity in the drawing through the command bar. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/S/table-eg.png})

Accessing the Command

command bar: **-table : -Table Prompts you in the command bar:** Enter number of columns or [Auto] <4>:**(Specify the number of columns, press Enter to accept the default columns, or else press A)** Enter number of rows or [Auto] <4>: **(Specify the number of rows, press Enter to accept the default columns, or else press A)** Specify insertion point or [Style/Width/Height]:**(Specify the insertion point, or else choose an option)** Specify first corner:(**Pick a point for the top left corner of the table, or enter x,y coordinates)** Specify second corner: **Pick a point for the bottom right corner of the table, or enter x,y coordinates)**

Command Options

Option	Description
Enter number of columns	Specifies the number of columns: • Enter a number • Press **Enter** to accept the default number of columns • Press **A** for the program to insert one column
Enter number of rows	Specifies the number of rows: • Enter a number • Press **Enter** to accept the default number of rows • Press **A** for the program to automatically determine the number of rows when you insert the table in the drawing

Auto	Defines the number of columns and/or rows dynamically when inserting the table; prompts you in the command bar: 1. **Specify first corner**- **(enter the position of the top left corner of the table.)** 2. **Specify second corner** - **(enter the position of the bottom right corner of the table)** As you move the cursor, the program adds more or less rows.
Style	Specifies the table style; prompts you: **Enter table style name <Standard>** - **(type the name of the table style, or press Enter to accept the default style.)**
Width	Specifies the column width; prompts you: **Enter cell width <6.0000>** - **(type the column width or press Enter to accept the default width)**
Height	Specifies the minimum row height; prompts you: **Enter minimum cell height <4>** - **(type the height or press Enter to accept the default.)**
Specify insertion point	Specify the position of the top left corner of the table.The TablEdit (6.29.3) command immediately launches so that you can begin filling in the cells.

Procedures

- To create an empty table (1.21.25)
- To create a table from data (1.21.25)
- To create a table from an Excel spreadsheet (1.21.25)
- General procedure to create fields (1.21.23)

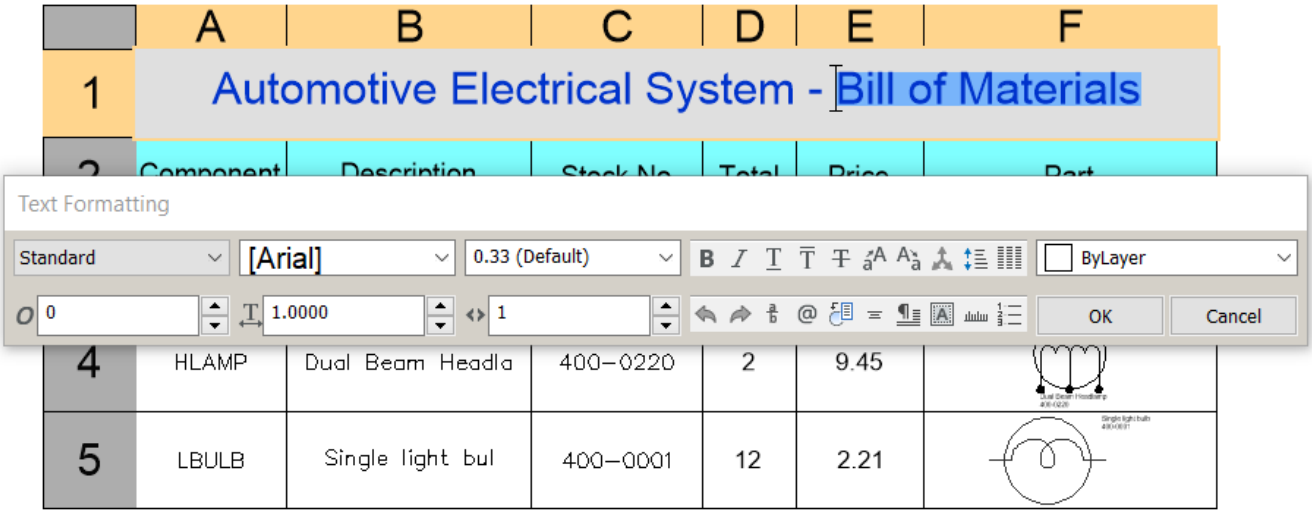
Related Commands

Table (6.29.1) - creates a table entity in the drawing through a dialog box. Tablemod (6.29.5) - modifies the properties of a selection of table cells. Tabledit (6.29.3) - edits the cell content. TableExport (6.29.4) - exports the content of a table entity in CSV file format. TableStyle (6.29.6) - creates and edits table styles through the Drawing Explorer - Table Styles dialog box.

Figure 6.2344



Figure 6.2345



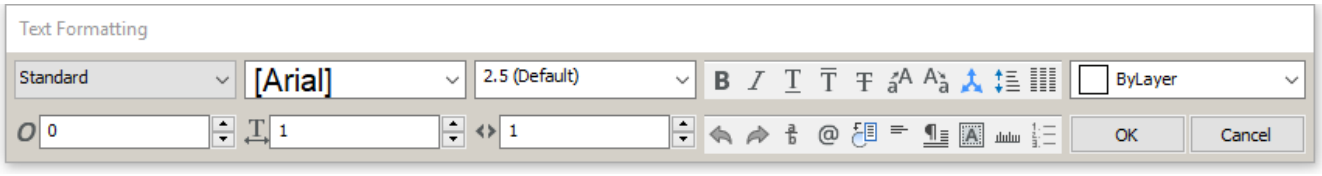
6.29.3 Tabledit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Edits text in a table cell. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 withtoolbar-eg.png}) **Note:** To edit the format of cells, use the TableMod (6.29.5) command.

Accessing the Command

command bar: **tabledit** mouse: double click a cell to open the Text Formatting bar. **: Tabledit Prompts you in the command bar:** Pick table cell to edit:**(Click a table cell)** Highlights the cell, and then displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/MtextEditor.png}) The Text Formatting dialog box operates exactly like the one from the MText command.

Figure 6.2346



Dialog Box Options

See theMtext (6.21.35) command.

Procedures

- To edit the table content (1.21.25)
- Editing a table in the Properties bar (1.21.25)
- Editing a cell in the Properties bar (1.21.25)

Related Commands

Table (6.29.1) - creates a table entity in the drawing through a dialog box. -Table (6.29.2) - creates a table entity in the drawing through the command bar. Tablemod (6.29.5) - modifies the properties of a selection of table cells. TableExport (6.29.4) - exports the content of a table entity in CSV file format. TableStyle (6.29.6) - creates and edits table styles through the Drawing Explorer - Table Styles dialog box. Mtext (6.21.35) - places multi-line text in drawings through a dialog box. Ddedit (6.10.14) - edits single-line text, multi-line text, attribute definitions, and attribute text.

Figure 6.2347



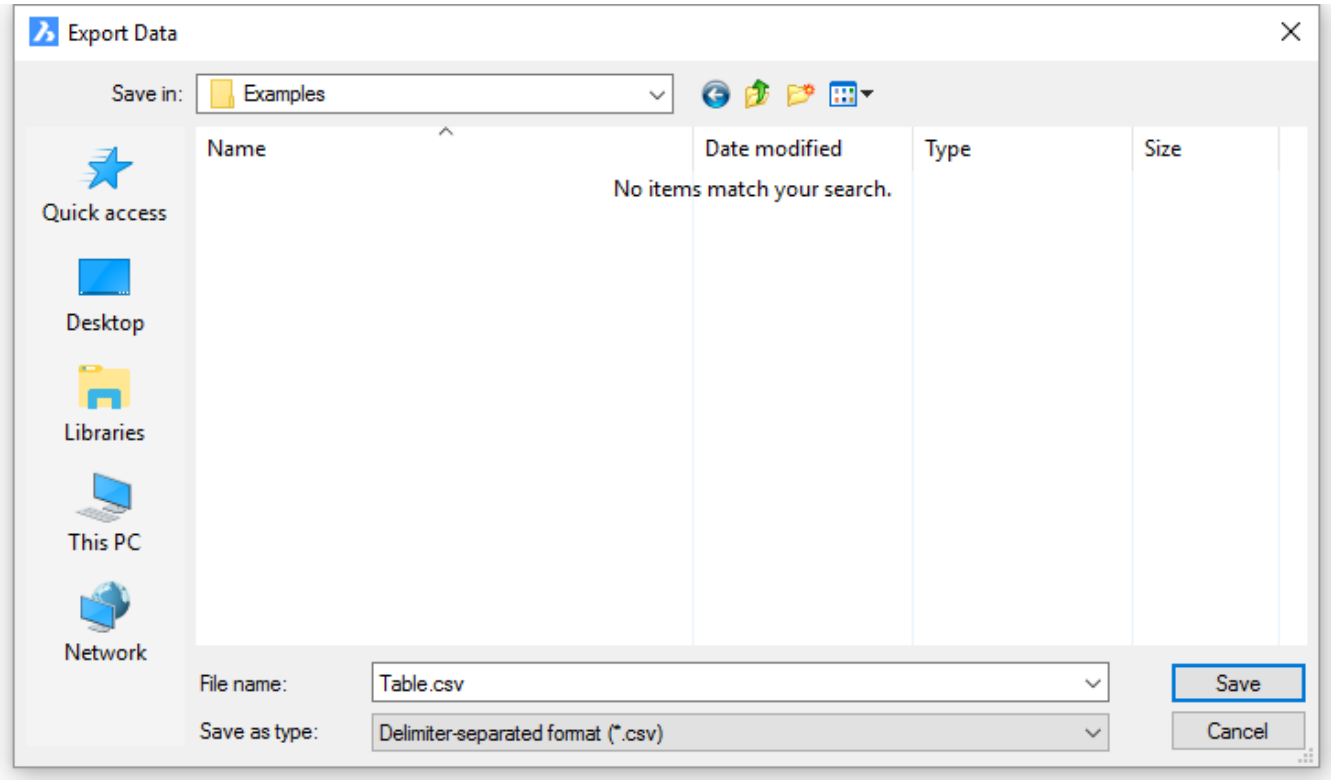
6.29.4 TableExport

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Exports the contents of a table to a CSV or XML file. **Note:** The CSV file can be imported into spreadsheet and database programs to format and further process the data.

Accessing the Command

command bar: **tableexport** context menu: **Export Table : TableExport Prompts you in the command bar:** Select Table to export:**(Select a table)** Displays the Export Data dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/

Figure 6.2348



Command Options

Option	Description
Save in	Select a folder in which to save the file
File name:	Type a name for the file; the default is **Table.csv**
Save as type	Choose the format of the file: • Delimiter-separated format (*.csv) • XML Spreadsheet format (*.xml)
Save	Exports the cell data and creates the export file.
Cancel	Cancels the TableExport command and so does not export the data.

Procedures

- To export a table (1.21.25)
- Defining the list separator character (1.21.25)

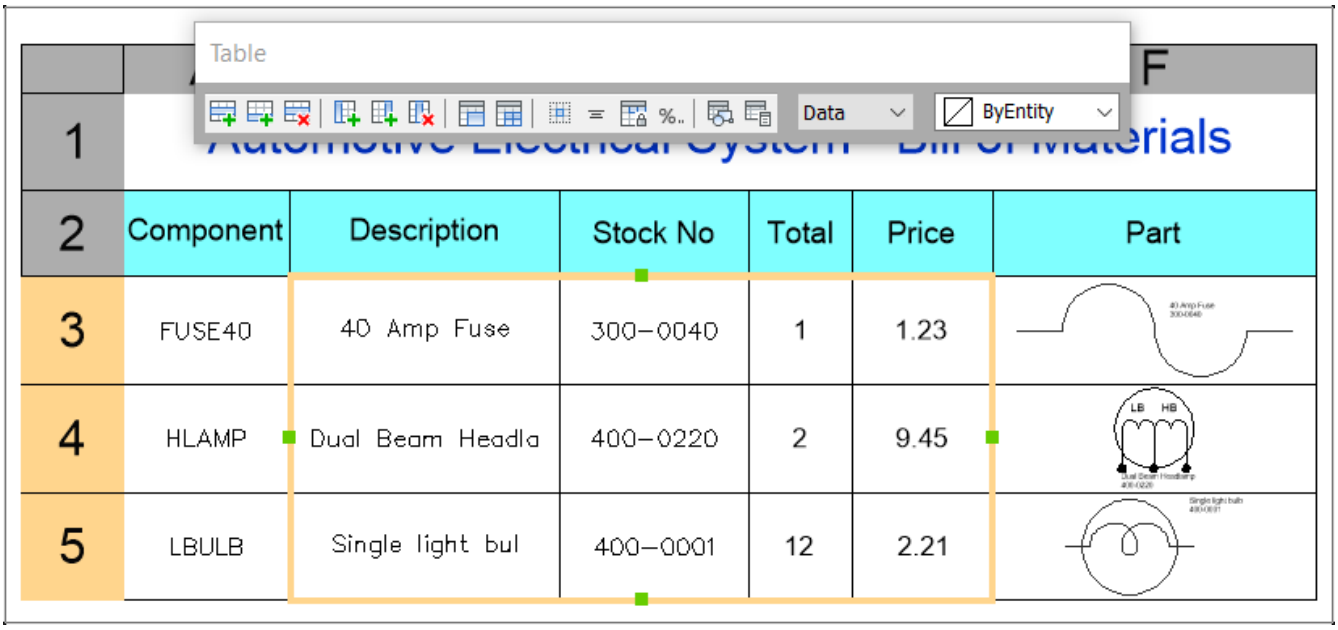
Related Commands

Table (6.29.1) - creates a table entity in the drawing through a dialog box. -Table (6.29.2) - creates a table entity in the drawing through the command bar. Tablemod (6.29.5) - modifies the properties of a selection of table cells. Tabledit (6.29.3) - edits the cell content. TableStyle (6.29.6) - creates and edits table styles through the Drawing Explorer - Table Styles dialog box.

Figure 6.2349



Figure 6.2350



6.29.5 TableMod

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Formats the properties of table cells, and inserts blocks. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tablemod-withtoolbar-eg.png}) **Note:** To edit and format text in cells, use the TablEdit (6.29.3) command.

Accessing the Command

command bar: **tablemod** mouse: click a cell in a table menu bar: **Modify | Modify Table : TableMod Prompts you in the command bar:** Pick table cell to edit:**(Click the cell you want to modify or press and hold the left mouse button then define a rectangle to select multiple cells)** Displays a toolbar: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel
To exit the command, click the drawing outside of the table or press **Esc**.

Figure 6.2351



Toolbar Options

Option	Description
Insert row above	Inserts an empty row above the selected cell
Insert row below	Inserts an empty row below the selected cell
Delete row	Deletes the row(s) of the selected cell(s) without warning; use the U command to reserve the change
Insert column left	Inserts an empty column at the left of the selected cell
Insert column right	Inserts an empty column at the right of the selected cell
Delete column	Deletes the column(s) of the selected cell(s) without warning; use the U command to reserve the change
Merge cells	Merges the selected cells; click the icon and then choose an option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C • **Merge All** - merges all selected cells into a single cell • **Merge by Row** - merges the selected cells into rows, each with one cell • **Merge by Column** -merges the selected cells into columns, each with one cell **Caution**: When cells are merged, all the their contents are erased, except for that of the first cell.
Unmerge cells	Splits previously merged cells by rows and columns; the program uses the row numbers (1, 2, 3, etc) and column letters numbers (A, B, C, etc)to know how to split the merged cells into the original set. When cells are unmerged, the content erased by the Merge Cells option is not returned.
Border style	Sets the border style of the selection; displays the Cell Border Properties dialog box: see below
Text alignment	Sets the text alignment of text in the selected cells; click the icon and then chooseoption: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C The content of each selected cell is aligned inside the cell. For instance, select four cells, and then choose Bottom Right, the content in each of the four cells is justified bottom-right.

Lock / unlock

Locks and unlocks the content and/or the format of selected cells; click the icon and then choose and option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tablocked-mnu.png})

- **Unlocked** - the selected cells are unlocked, which means that they can be edited; the program responds in the command line, 'Cell contents unlocked'
- **Locked Content** - the content of the the selected cells is locked, but the format of the cells can still be modified; the program responds in the command line, 'Cell contents locked'
- **Locked Format** - the format of the the selected cells is locked, but the content of the cells can still be edited; the program responds in the command line, 'Cell format locked'
- **Locked Format and Content** - the content and the format of the the selected cells are locked; the program responds in the command line, 'Cell contents and format locked'

It is common to lock header cells.

Cell format

(Not implemented) Sets the cell format of the selection

Inert block reference

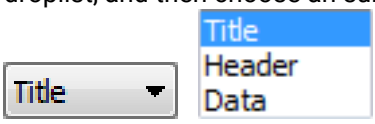
Inserts a block or a drawing in the selected cell; click the icon to display the Insert Block in Cell dialog box: see below **Caution**: placing a large drawing in a small cell can lead to program instability, never mind that you can't even see the drawing. You can have both text and blocks in a single cell: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tableblocktext-eg.png}) Use the Manage Cell Content dialog box to arrange the text and blocks; see below.

Manage cell content

Manages blocks and drawings in cells; click the icon to display the Manage Cell Content dialog box: see below

Cell style

Sets the sub-style of the selected cells; click the droplist, and then choose an sub-style name:



- **Title** - the title style is usually applied to the topmost row of the table, but doesn't need to be!
- **Header** - the Header style is usually applied to the first row of a column
- **Data** - the Data style is usually applied to all other rows in the table

The properties of the Header, Border, and Cell sub-styles are defined by the table style; see the TableStyle (6.29.6) command.

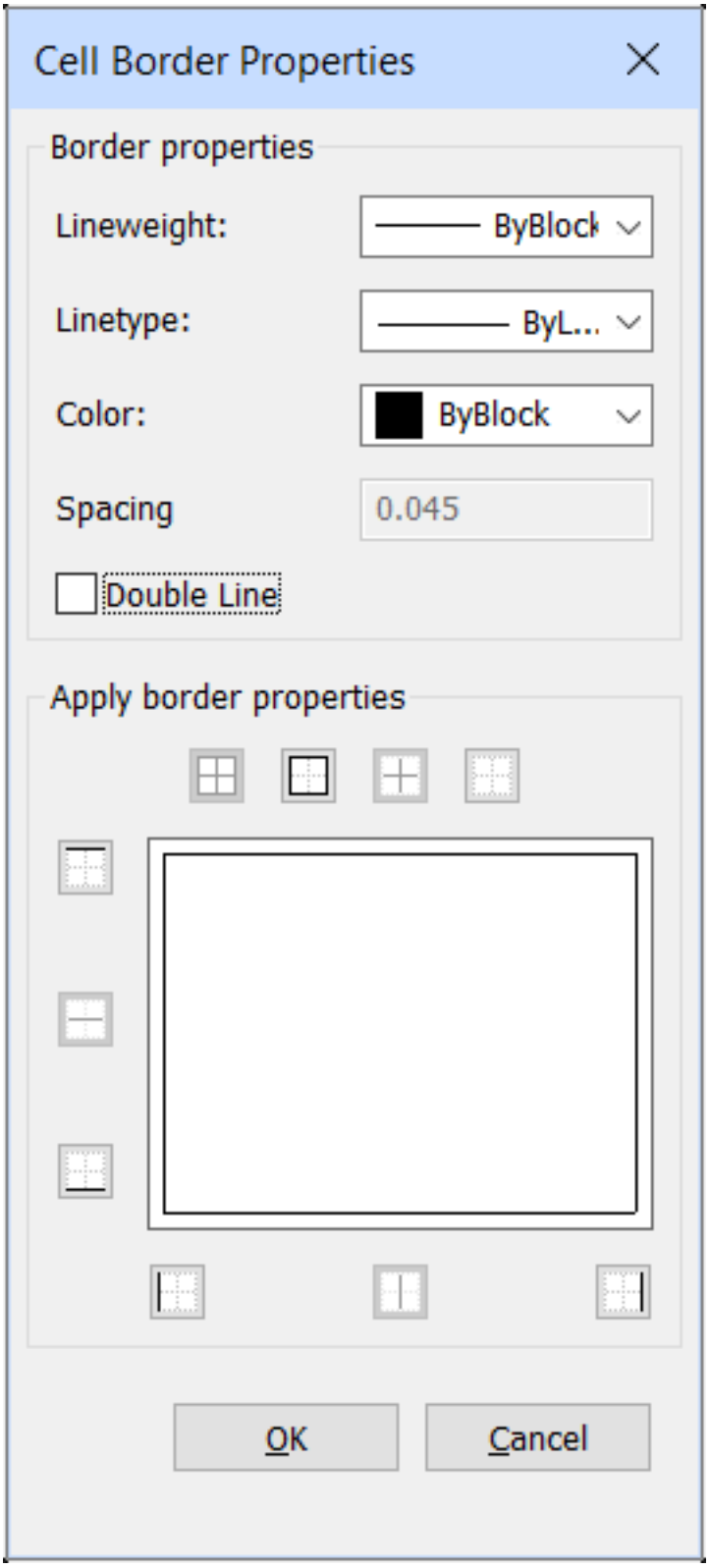
Cell background	Sets the background color of the selected cell(s): 1. Click the droplist 2. Choose a color. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C Or in the droplist click **Select Color**to open the Color (6.9.33) dialog box.
Esc	Press **Esc** to exit this command.

Cell Border Properties Dialog Box

To access this dialog box, click the **Border Style** button: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1
cellboderproperties-dlg.png})

Option	**Description**
Border Properties	Determines the look of the borders that surround each cell: **Lineweight** - changes the lineweight of the cell border(s); click the droplist and then choose a different lineweight **Linetype** -changes the line type of the cell border(s); click the droplist and then choose a different one, or else click **Load** to access additional line types **Color** - changes the color of the cell border(s); click the droplist, and then choose a different color, or else click **Select Color** to access all colors (Not implemented) **Spacing** - changes the spacing between the cell borders and the content of the cell; enter a number in drawing units **Double Line** - doubles the lines make up the borders of the cell(s)
Apply Border Properties	Applies the properties to only certain borders: click a button to see the effect in the preview window; going counter clockwise: • **No borders** - the properties are applied to no borders (strange but true) • **Inside borders** - the properties are applied only to the inner borders (you have to have selected more than one cell for this one to work) • **Outsideborders** - and so on • **Allborders** • **Top borders** • **Middleborders** • **Bottom borders** • **Left borders** • **Center borders** • **Right borders** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tab allboarders.png}) Although the preview window shows only the double-line effect, the other properties are applied to the specified cell border lines.
OK	Exits the dialog box, applies the properties, and returns to the Table toolbar.

Figure 6.2352



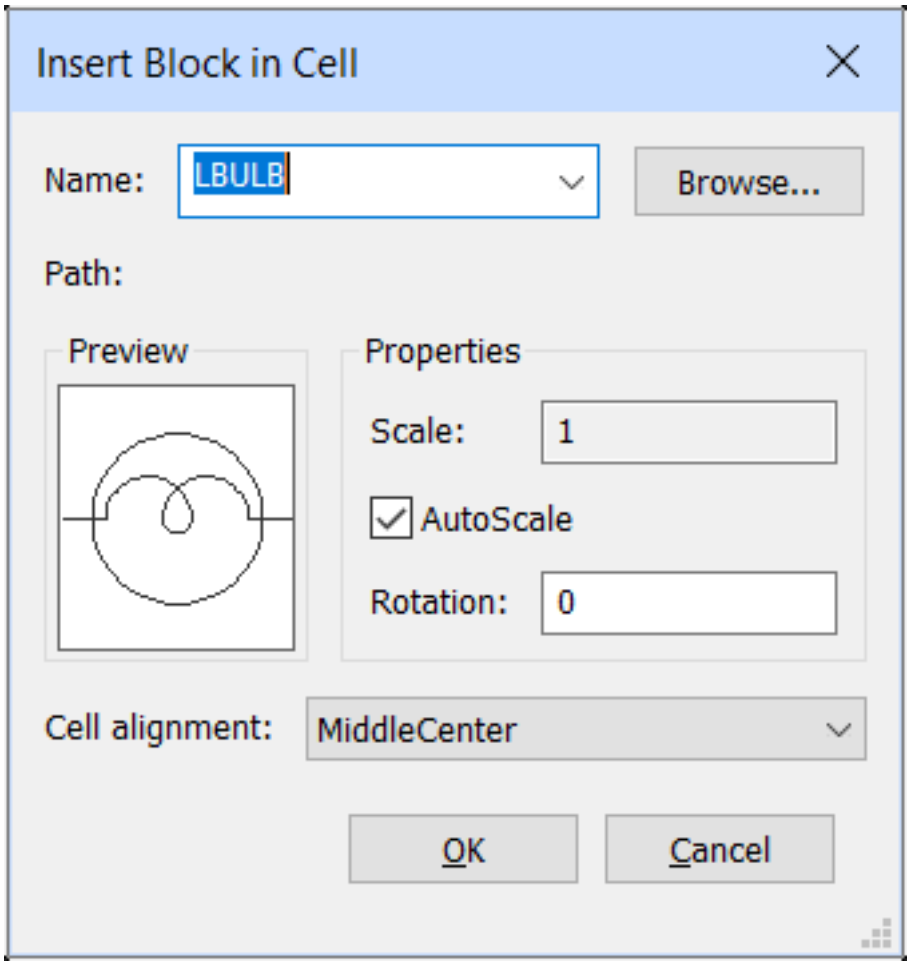
Insert Block in Cell Dialog Box

To access this dialog box, click the **Insert Block Reference** button: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help insertblockincell-dlg.png})

Option	Description
Name	Specifies the name of the block or drawing to insert. • To insert a block: 1. Click the droplist 2. Choose the name of a block already defined in the drawing • To insert a drawing: 1. Click **Browse**. 2. In the Open Drawing File dialog box, choose a folder 3. From the **Files of Type** droplist, choose the file format, DWG or DXF 4. Select a drawing file 5. Click **Open** When the cell is small, the block or drawing may be barely perceptible. You may need to increase the size of the row or column to accommodate the blocks. A cell can hold text and one or more blocks. When a 3D model is pasted into the cell, the 2D planview is displayed: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tab 3das2d-eg.png})
Properties	Determines the size of the block in the cell: **Scale** - specifies the size of the block (unavailable when AutoScale is turned on): • Numbers larger than 1.0 make the block larger • **1** - inserts the block at its actual size • Numbers smaller than 1.0 make the block smaller • Negative numbers flip the block upside down **AutoScale** - size the block to fit the cell: • **On** - the block is resized to fit the cell • **Off** - the cell is resized to fit the block **Rotation** - rotate the block about its center point (and not its insertion point); enter an angle: • Positive numbers rotate the block counter clockwise • **0** - does not rotate the block • Negative numbers rotate the block clockwise

Cell Alignment	Aligns the block in the cell: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tab cellalignment.png}) 1. Click the droplist 2. Choose a justification mode
OK	Exits the dialog box, inserts the block, and returns to the Table toolbar

Figure 6.2353

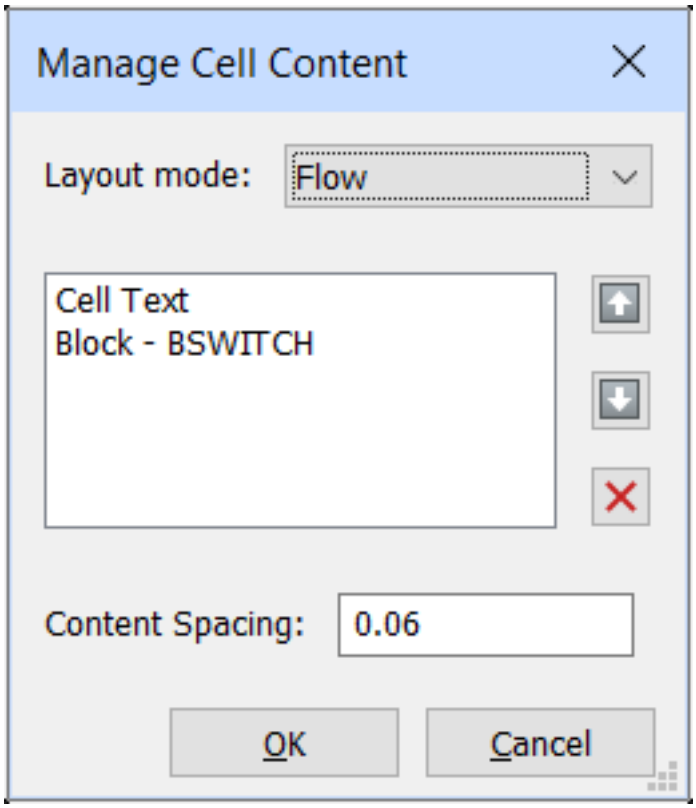


Manage Cell Content Dialog Box

To access this dialog box, click the **Manage Cell Content**button: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/e managecellcontent-dlg.png})

Option	Description
Layout Mode	When a cell contains more than one element, this option determines how they relate to each other; click the droplist and then choose an option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tab layoutmode.png}) • **Flow** - arranges the elements to fit the cell • **Stacked horizontal** - forces the elements to be arranged horizontally • **Stacked vertical** -forces the elements to be arranged vertically
Move Up	Moves the selected element up the list, so that it appears above other elements in the cell
Move Down	Moves the selected element down the list, so that it's below other elements in the cell
Delete	Deletes the selected element from the cell
Content Spacing	(This option is inoperable) Specifies the distance between elements in the cell; enter a number in drawing units
OK	Exits the dialog box, changes the arrangement of the cell's contents, and then returns to the Table toolbar

Figure 6.2354



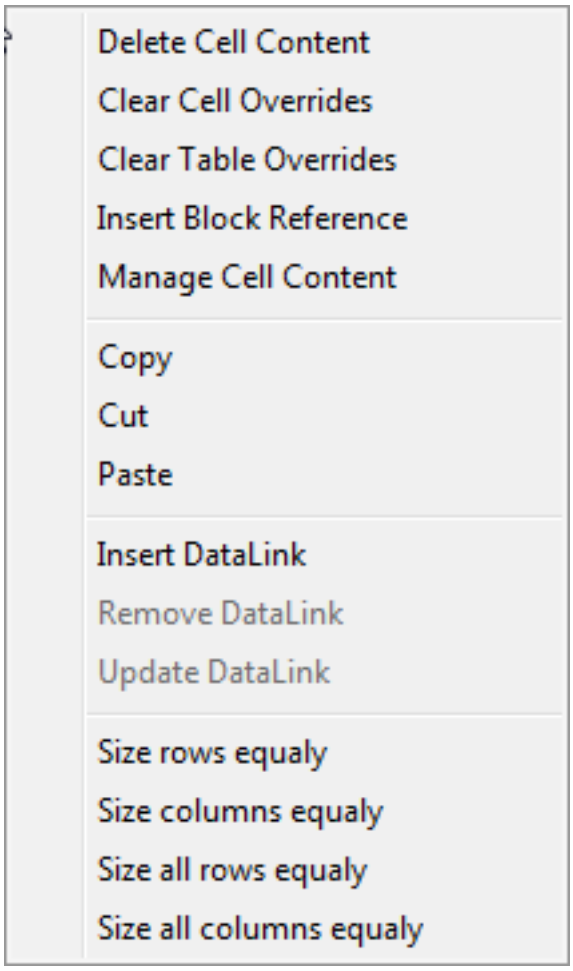
Context Menu

While the TableMod command is running, right-click a cell to see this context menu: (\vref {fig:C:/Dev/Task-2448/images/static rtclk.png})

Option	Description
Delete Cell Content	Deletes the text and blocks in cells
Clear Cell Overrides	Resets the properties of cells to the Cell sub-style settings defined in the table style
Clear Table Overrides	Resets the table properties to those defined in the table style
Insert Block Reference	Inserts blocks and drawings; seeInsert Block in Cell dialog box
Manage Cell Content	Manages the text and blocks in cells; seeManage Cell Content dialog box
Copy	Copies the content of the cell(s) to the Clipboard for pasting into other cell(s)
Cut	Cuts the content of the cell(s) to the Clipboard for pasting into other cell(s); existing content is erased
Paste	Pastes cell content from the Clipboard into cell(s), with these caveats: • All content in the cell is replaced by the pasted item • When more than one cell is selected for pasting, content is pasted only in the first cell • Non-cell data is not pasted, such as entities from the drawing or text from another application

Insert DataLink	Adds data from an external file to the end of the table; see theDataLink (6.10.2) command. Acceptable file formats are • **CSV** - ASCII files in which fields are separated by a deliminter character, such as a comma or tab • **XLS** - older Excel spreadsheet files • **XLSX** -new Excel spreadsheet files
Remove DataLink	(Available only when the table has a datalink) Removes the linked to the table; the data remains in the table
Update DataLink	(Available only when the table has a datalink) Updates the data in the table from the external file
Size Rows Evenly	Sizes the selected rows to match the size of the tallest cell in the selection set
Size Columns Evenly	Sizes the selected columns match the size of the widest cell in the selection set
Size All Rows Evenly	Sizes all rows in the table to match the tallest row
Size All Columns Evenly	Sizes all columns in the table to match the widest column

Figure 6.2355



Procedures

- Editing a table using grips (1.21.25)
- Editing a table using the Table toolbar (1.21.25)
- Editing a table in the Properties bar (1.21.25)
- Editing a cell in the Properties bar (1.21.25)

Related Commands

Table (6.29.1) - creates a table entity in the drawing through a dialog box. -Table (6.29.2) - creates a table entity in the drawing through the command bar. Tabledit (6.29.3) - edits the cell content. TableExport (6.29.4) - exports the content of a table entity in CSV file format. TableStyle (6.29.6) - creates and edits table styles through the Drawing Explorer - Table Styles dialog box.

Figure 6.2356

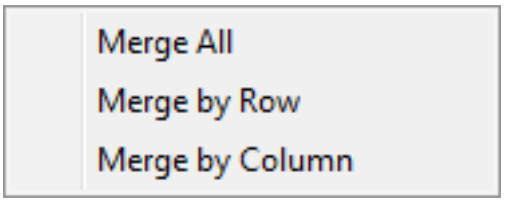


Figure 6.2357

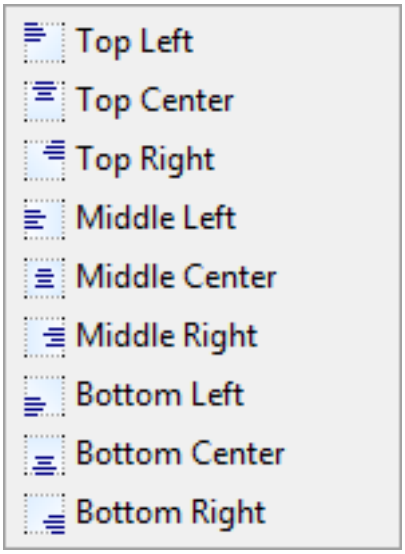


Figure 6.2358

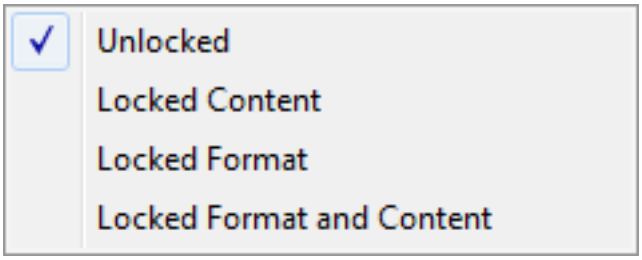


Figure 6.2359

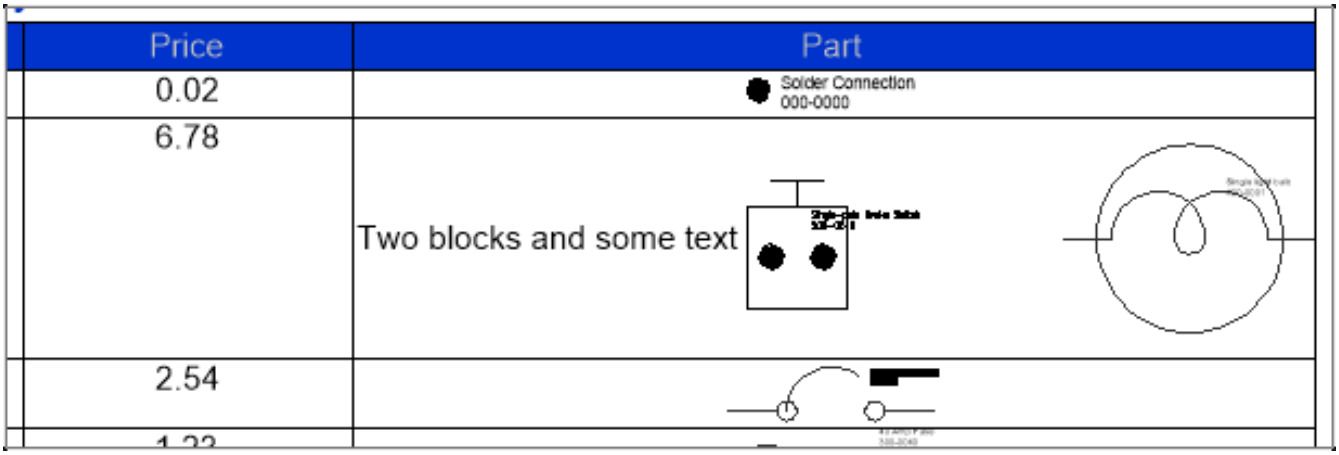


Figure 6.2360



Figure 6.2361

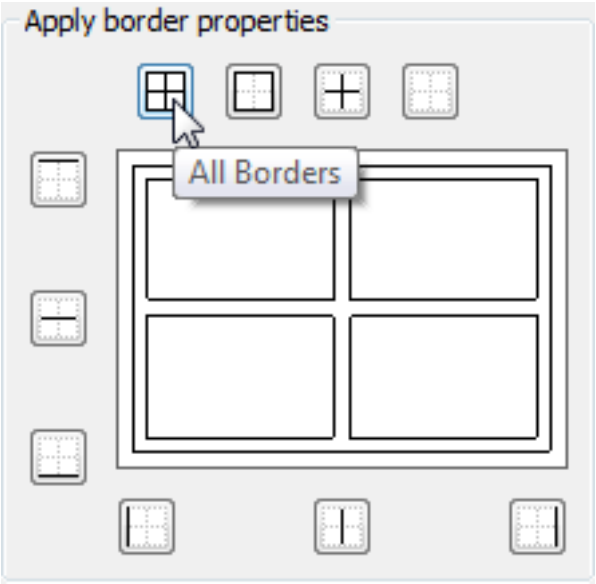


Figure 6.2362

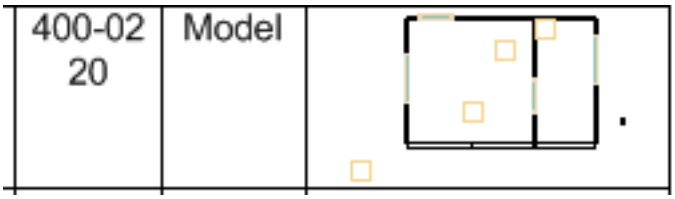


Figure 6.2363

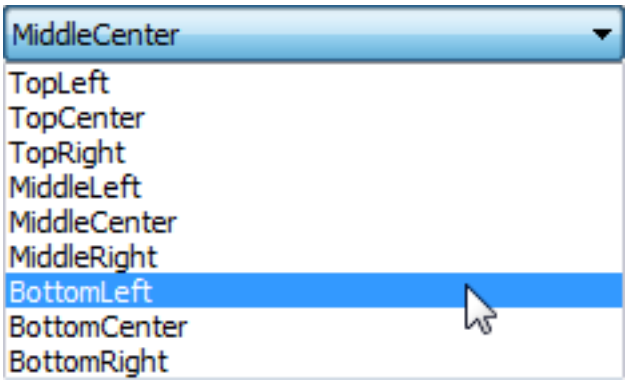


Figure 6.2364

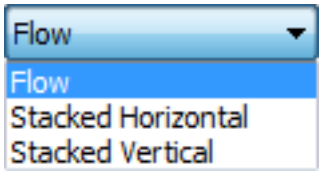


Figure 6.2365



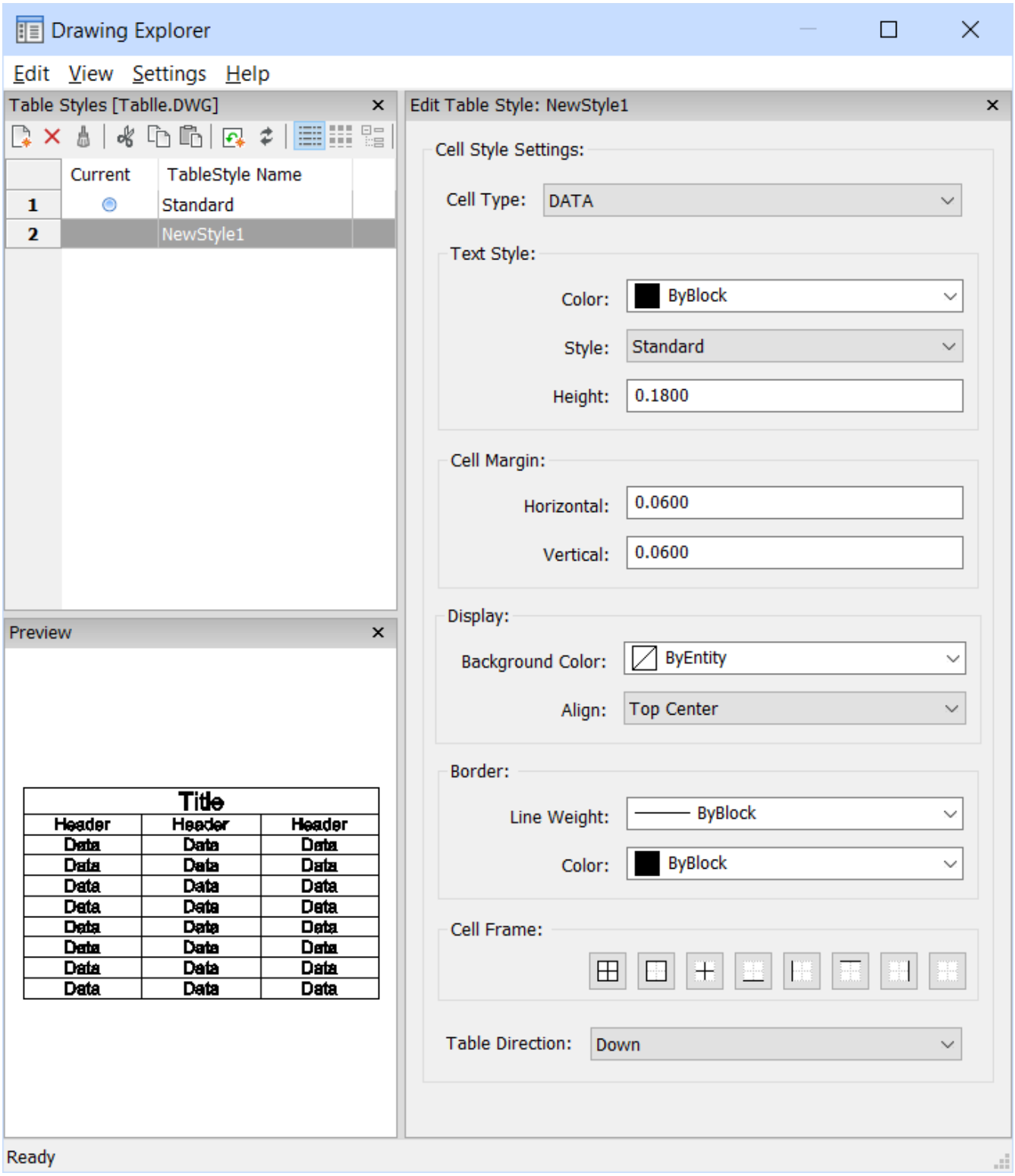
6.29.6 TableStyle

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and edits table styles through the Drawing Explorer.

Accessing the Command

command bar: **tablestyle** menu bar: **Tools | Drawing Explorer | Table Styles : TableStyle** Displays the Table Styles section of the a Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commar dlg.png}) Makes changes to a style, or else create a new one, and then click **X** to close the dialog box.

Figure 6.2366



Toolbar Options

Option	Description
 New	Creates a new table style, edit the default name 'NewStyle1' to make it your own. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N
 Delete	Deletes the currently selected table style without warning. You cannot deleted these style: • Table styles in use • 'Standard' stable style

Edit Table Style Options

Option	Description
Cell Style Settings	Defines the cell style settings for each cell style: Title, Header, and Data: **Cell Type** - click the droplist, and hen choose a cell style: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C
Text Style	Sets the **Color** , **Style** and **Height** for the selected cell style: • **Color** - specifies the color of the text in the cell • **Style** - selects the text style for the text • **Height** - specifies the initial height for text in the cell
Cell Margin	Defines the spacing between the cell border and the content of the cell. • **Horizontal** - specifies the distance between the cell content and the vertical cell borders • **Vertical** - specifies the distance between the cell content and the horizontal cell borders
Display	Defines the background color and the text alignment: • **Background Color** - pick a background color from the droplist, or else choose **Select Color**to display the **Color** dialog bop • **Align** - click the droplist, and then choose an alignment: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou
Border	Sets the lineweight and color of the cell borders. • **Line Weight** - click the droplist, and then choose a line weight; line wights aredisplayed only when the **LWT**option is active on the status bar • Color - pick a border color from the droplist, or else choose **Select Color**to display the **Color** dialog bop.

Cell Frame	Applies the border properties to specific lines that make up the cell frames. Click a button to choose an option, and then view the effect in the Preview window: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C
Table Direction	Determines the direction in which the table grows when you add and remove rows: • **Down** - the title and header cells at the top, and the table grows downwards • **Up** -the title and header cells at the bottom, and the table grows upwards

Procedures

- To open the Table Styles Explorer (1.21.26)
- To create a table style (1.21.26)
- To edit a table style (1.21.26)
- To apply a style to a table (1.21.26)
- To delete a table style (1.21.26)

Related Commands

-Table (6.29.2) - creates a table entity in the drawing through the command bar. Table (6.29.1) - creates a table entity in the drawing through a dialog box. Tablemod (6.29.5) - modifies the properties of a selection of table cells. Tabledit (6.29.3) - edits the cell content. TableExport (6.29.4) - exports the content of a table entity in CSV file format. Color (6.9.33) - specifies the current color, through a dialog box. Style (6.27.53) - creates and modifies text styles with the Drawing Explorer. -Style (6.27.54) - creates and modifies text styles in the command bar. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.2367

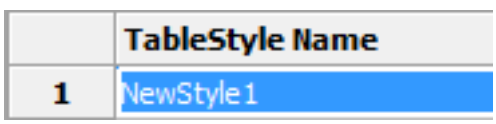


Figure 6.2368

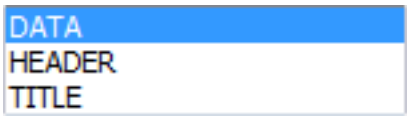


Figure 6.2369

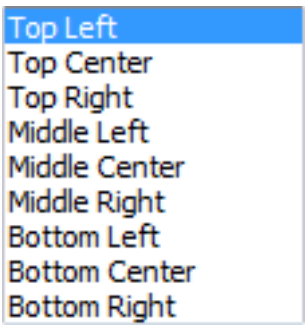


Figure 6.2370

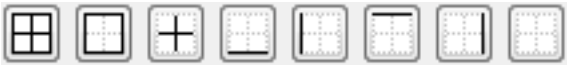


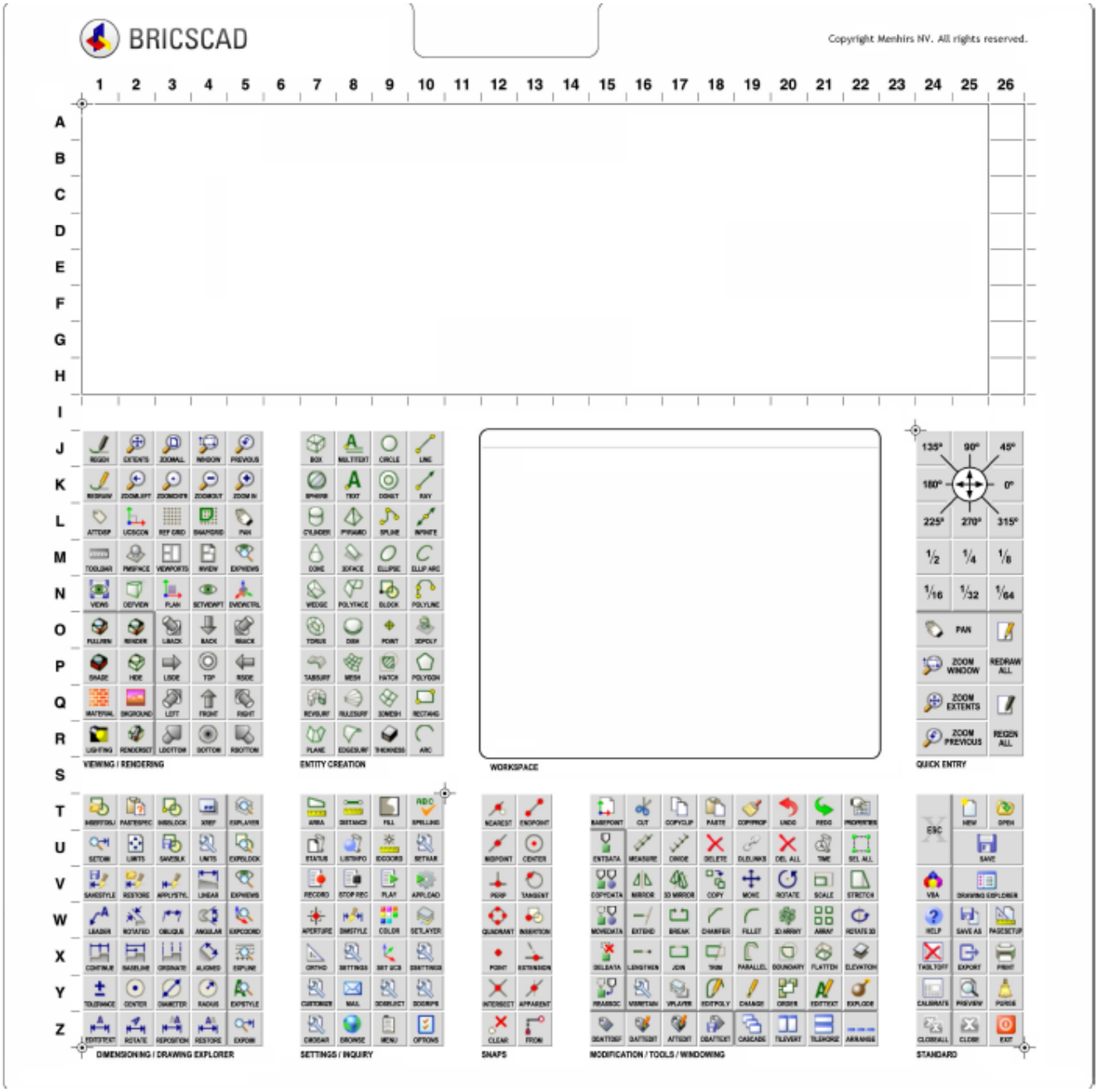
Figure 6.2371



6.29.7 Tablet

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Configures and calibrates digitizing tablets, and toggles tablet mode. **Note:**This command requires that **Wintab32.dll** from Wacom Technology be installed in folder **C:\Windows\System32**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Command eg.png})

Figure 6.2372



Accessing the Command

command bar: **tablet** shortcut: **T** alias: **ta** menu bar: **Settings | Tablet** status bar: **TABLET : Tablet Prompts you in the command bar:** Tablet: [ON/OFF/CALibrate/ConFiGure]<On>: **(Enter an option)**

Command Options

Option	Description
ON	Turns on tablet digitizing mode.The entire surface of the table is used like a mouse for picking points and selecting entities.
OFF	Turns off tablet digitizing mode.The menu areas become available for selecting commands.
CALibrate	Calibrates the coordinates of the tablet.
ConFiGure	Configures the tablet's menu areas.

Status Bar Menu

Right-click TABLET on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image791.png})

Shortcut Menu	Description
ON	Turns on tablet digitizing mode.
OFF	Turns off tablet digitizing mode.
Settings	Opens the Coordinate Input section of the Settings dialog box. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Tablet mode	Toggles tablet mode: • **On** - switches tablet to digitizing mode • **Off** - switches tablet to command selection mode

Figure 6.2373

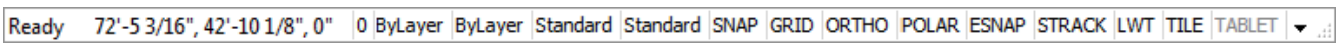
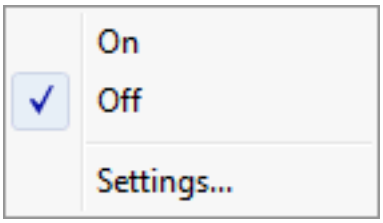


Figure 6.2374



Procedures

- To use a digitizing tablet in BricsCAD (1.12.8)
- To download the tablet overlay and CUI files (1.12.8)
- To load the overlay CUI file (1.12.8)
- To switch the tablet ON / OFF (1.12.8)
- To configure the tablet (1.12.8)
- To calibrate the tablet (1.12.8)

Related Commands

None.

Figure 6.2375

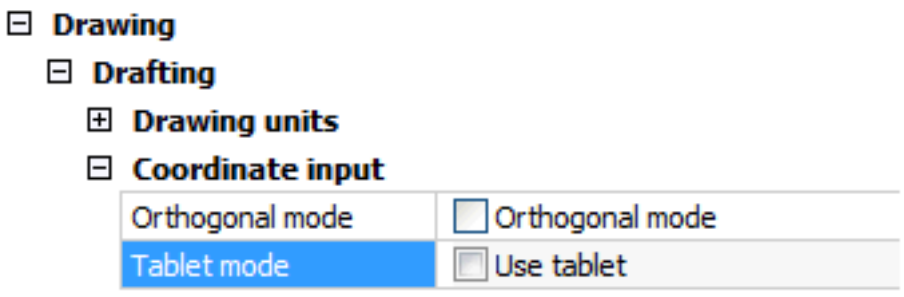
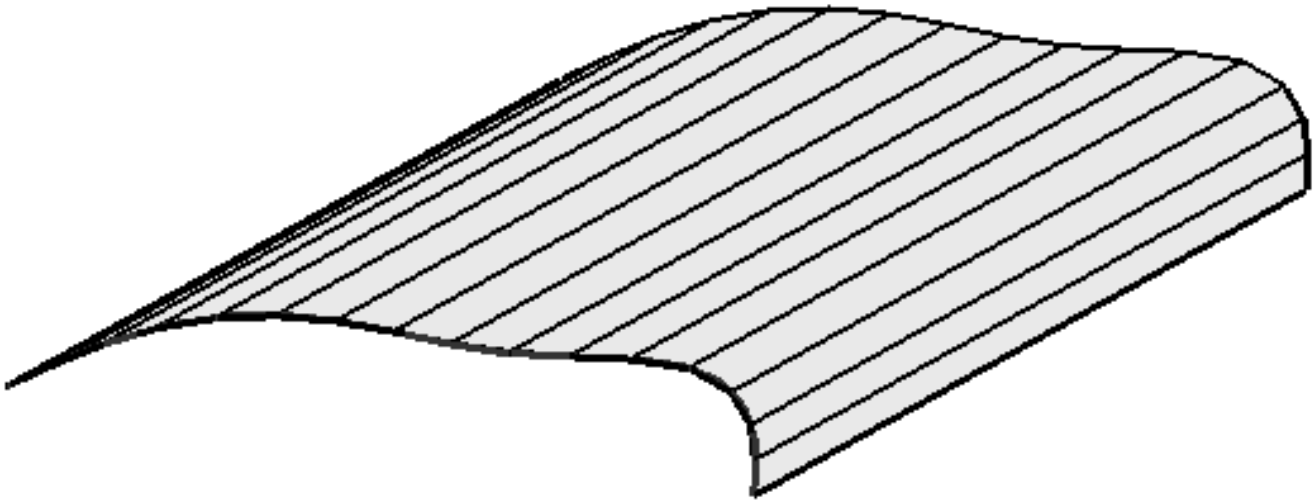


Figure 6.2376



Figure 6.2377



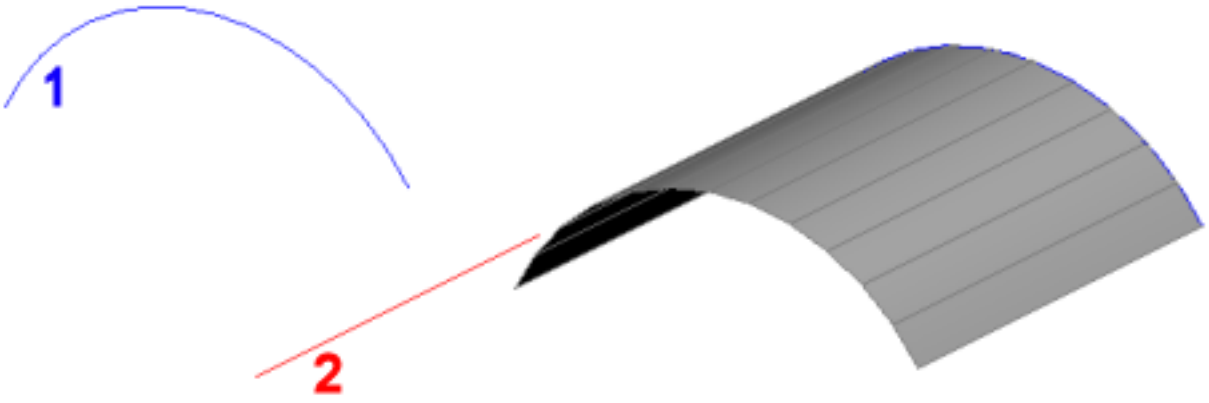
6.29.8 TabSurf

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates a 3D mesh surface by extruding an 2D entity along a straight path entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tabsurf-eg.png})

Accessing the Command


command bar: **tabsurf** menu bar: **Model | Meshes | Extruded Surface** ribbon: **Meshes | 3D Mesh Surfaces** toolbar: **Meshes | 3D Surfaces |**  **: TabSurf Prompts you in the command bar:** Select entity to extrude: **(Choose a single open entity, such as #1 in the figure below)** Select line or open polyline for extrusion path: **(Pick a line or polyline, such as #2)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/TabSurf.png}) Arc segments of the extruded entity are divided in a number of equal length intervals defined by the SURFTAB1variable.

Figure 6.2378



Command Options

Option	Description
Select entity to extrude	Specifies the cross-section shape of the extrusion.The entity to be extruded must be open and must be a single entity.
Select line or open polyline for extrusion path	Specifies the length and direction of the extrusion.The extrusion path can be a line or open polyline. The polyline need not be a line segment; if a polyarc or multisegment polyline, the extrusion, however, is always "straight," going from the start to the endpoint of the polyline. The path is a vector, which means its length specifies the length of the surface; its direction specifies the direction of extrusion. The extrusion starts at the entity to be extruded, but goes in the direction indicated by the path.The end selected on the polyline or line defines the direction of the extrusion.

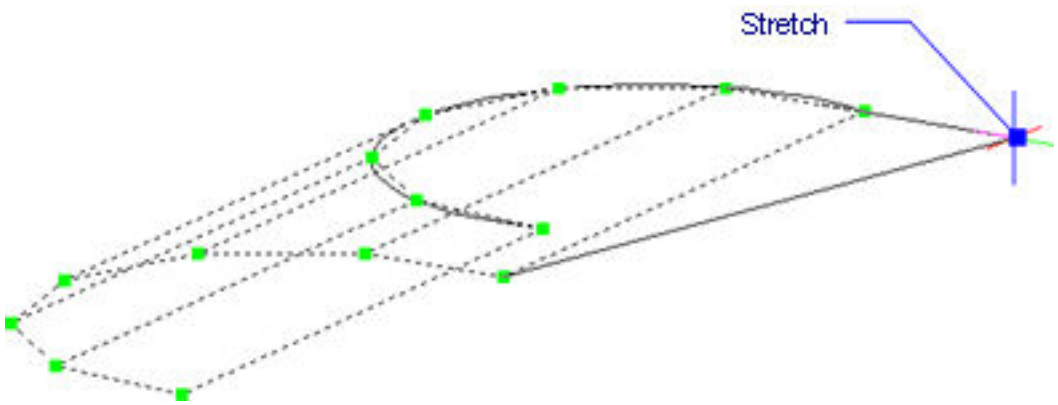
Grips Editing

Mesh-formed tabulated surfaces can be edited directly through grips:

1. Select the tabsurf. Notice that it has one grip at each vertex.
2. Drag the grip to move the adjacent faces.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image492.jpg})

Figure 6.2379



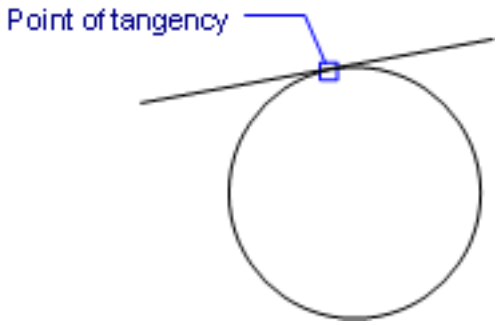
Related Commands

Ai_Box (6.5.3) - draws 3D boxes as mesh surfaces. Ai_Cone (6.5.4) - draws 3D cones as mesh surfaces. Ai_Cylinder (6.5.5) - draws 3D cylinders as mesh surfaces. Ai_Dish (6.5.7) - draws 3D dishes as mesh surfaces. Ai_Dome (6.5.8) - draws 3D domes as mesh surfaces. EdgeSurf (6.13.2) - draws edge surfaces as mesh surfaces. Ai_Pyramid (6.5.10) - draws 3D pyramids as mesh surfaces. RevSurf (6.26.32) - draws revolved surfaces as mesh surfaces. RuleSurf (6.26.45) - draws ruled surfaces as mesh surfaces. Ai_Sphere (6.5.13) - draws 3D spheres as mesh surfaces. Ai_Torus (6.5.15) - draws 3D donuts as mesh surfaces. Ai_Wedge (6.5.16) - draws 3D wedges as mesh surfaces.

Figure 6.2380



Figure 6.2381



6.29.9 Tangent

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles tangent entity snap; snaps to the tangent points of circles, arcs, ellipses and elliptical arcs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image605.png})

Accessing the Command

 command bar: **tangent** transparent: **tan** toolbar: **Entity Snaps** |  **: Tangent Displays no prompts in the command bar.** When tangent entity snap is on, the cursor snaps to points of tangency along curved entities, such as circles, arcs, ellipses and elliptical arcs.

Command Options

Option	Description
Tangent	Entering the command toggles Tangent entity snap. It snaps to points of tangency. This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

OSnap (6.23.12) - toggles entity snap modes in the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes in the command bar. Aperture (6.4.10) - sizes the aperture cursor for entity snaps. 2dIntersection (6.2.1) - toggles apparent intersection snap mode. 3dIntersection (6.3.6) - toggles intersection snap mode. Apparent (6.4.11) - toggles apparent intersection snap mode. Center (6.9.3) - toggles center snap mode. Endpoint (6.13.7) - toggles endpoint snap mode. **Extension (6.13.20)** - toggles extension snap mode. From (6.14.11) - allows to enter relative coordinates with respect to a reference point. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap. Insertion (6.17.16) - toggles insertion snap mode. Intersection (6.17.20) - toggles intersection snap mode. Midpoint (6.21.16) - toggles midpoint snap mode. Mtp (6.21.36) - allows to specify a point as the midpoint between two points. Nearest (6.22.2) - toggles nearest snap mode. Node (6.22.7) - toggles node (point) snap mode. None (6.22.8) - turns off all entity snap modes. Parallel (6.24.4) - toggles parallel snap mode. Perpendicular (6.24.22) - toggles perpendicular snap mode. Quadrant (6.25.7) - toggles quadrant snap mode.

Figure 6.2382



6.29.10 TConnect

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Connects solid faces to other solids. Connections can be made in any direction. If the DELETEINTERFERENCE system variable is ON, the connected solid is subtracted from interfering solids.

Accessing the Command



Keyboard: **tconnect** Menu: **BIM | BimConnect | TConnect** Toolbar:**BIM Connect** Quad Tab: **Model** Ribbon Tab | Panel:**Model | Modify 3D - Connect : TConnect Prompts you:** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help
Select face to connect: **(Select the face of a solid.)** Entities in set: 1 Select face to connect: **(Select more faces or press Enter.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/TConnectSelect1.png})
Select entities to connect to or [Connect to nearest] <Connect to nearest>: **(Select a solid or a solidface, or press Enter to connect to the nearest solid.)** Entities in set: 1 (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19
Select entities to connect to or [Connect to nearest] <Connect to nearest>: **(Select more faces or press Enter to create the connection.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/TConnectResult1.png

Figure 6.2383

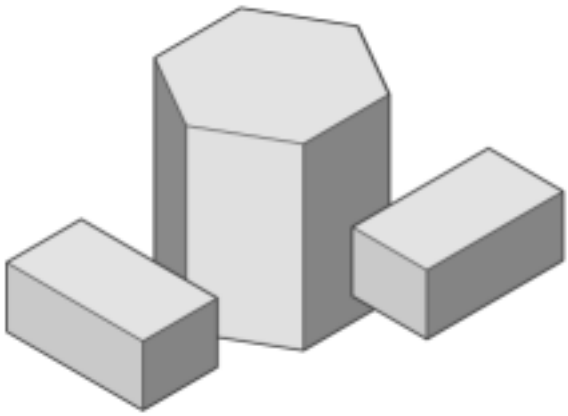


Figure 6.2384

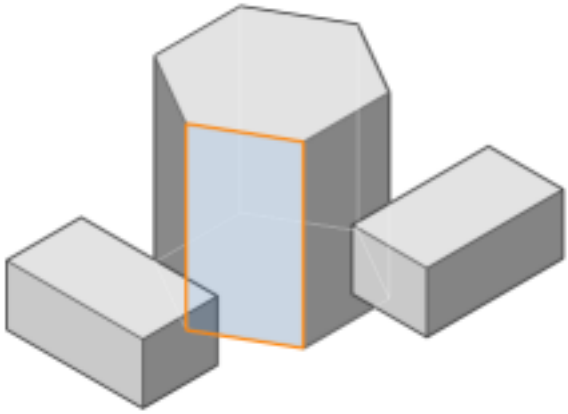
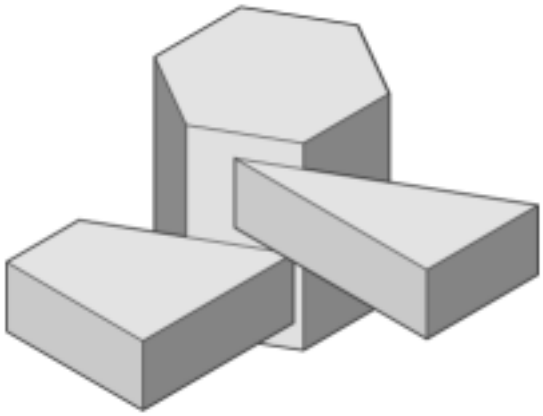


Figure 6.2385



Command Options

Option	Description
Connect to nearest	Connects the selected face(s) to the nearest face(s) of other solid(s). Select the faces to connect to the nearest solid. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/TConne Right click or press Enter to create the connection. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/TConne

Related Commands

LConnect (6.20.19) - creates an L-connection between solids. DmPushPull (6.12.20) - adds or removes volume from a solid by moving/offsetting a face. BimStructuralConnect (6.7.30) - connects structural profile solids such as beams and columns. BimFlowConnect (6.7.11) - creates a connection between flow segments, such as pipes or HVAC ducts.

Figure 6.2386

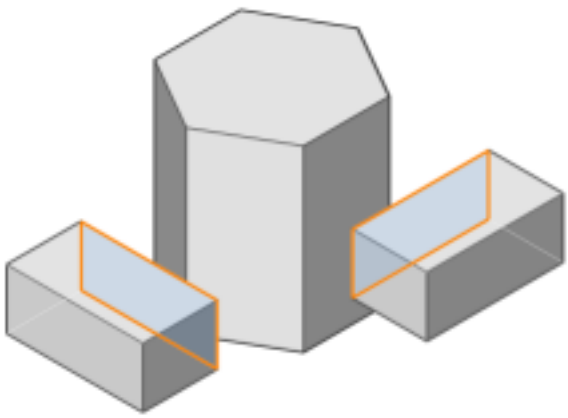


Figure 6.2387

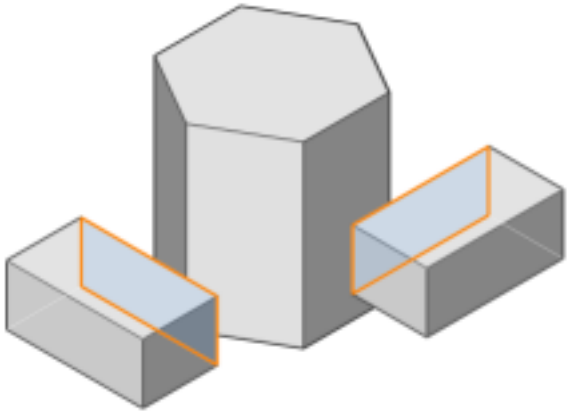


Figure 6.2388

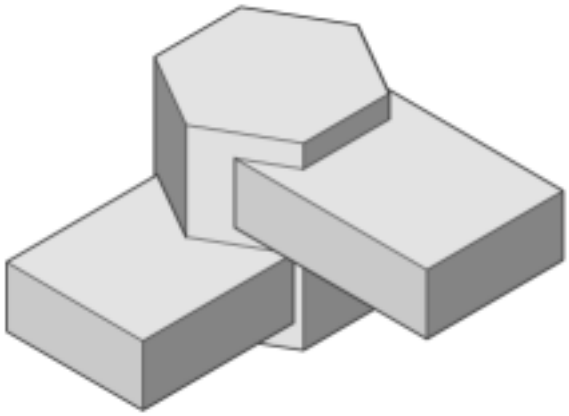


Figure 6.2389



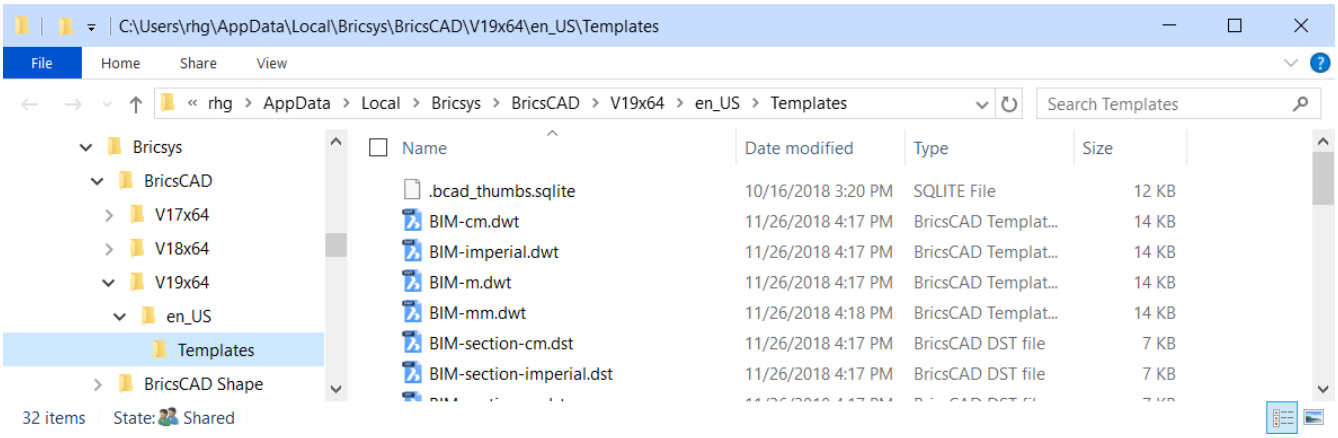
6.29.11 TemplateFolder

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Opens the Template folder in the operating systems file manager.

Accessing the Command

command bar: **templatefolder** transparent: **'templatefolder : TemplateFolder** Displays a file dialog with box the Template folder, which by default is **C:\Users\<user_name>\AppData\Local\Bricsys\BricsCAD\Vxx\en_US\Templates** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/templatefolder-dlg.png}) The template folder path is saved by the TemplatePath (1.13.11) user preference.

Figure 6.2390



Command Options

No Options

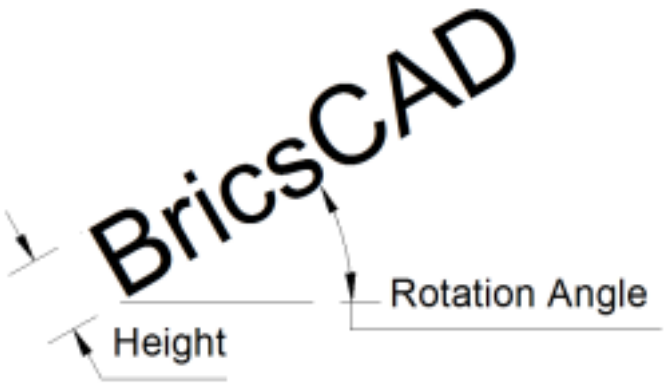
Related Commands

SupportFolder (6.27.58) - opens the main Support folder in a File dialog box.

Figure 6.2391



Figure 6.2392



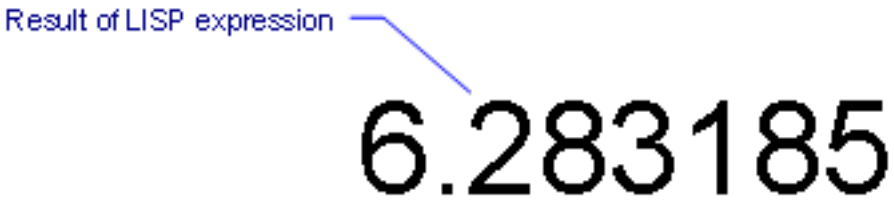
6.29.12 Text

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Places single lines of text in the drawing; optionally evaluates LISP expressions. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscon
Note: To enter paragraph text that can be extensively formatted, use the MText (6.21.35)command.

Accessing the Command

command bar: **text** aliases: **-t text tx : Text Prompts you in the command bar:** Text: Style/Align/Fit/Center/Middle/Right/Ju point>: **(Pick a point, or enter an option)** Height of text <0.2000>: **(Specify a height, or press Enter to accept the default**) Rotation angle of text <0>: **(Specify an angle, or press Enter to accept the default)** Text: **(Enter text, and then press Enter.) Text: (Enter more text, and else press Enter twice to end the command)** When the TextEval variable is set to 1, this command evaluates LISP expressions: Text: (* pi 2) The result of the equation (pi x 2) is placed as text: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image623.png})

Figure 6.2393



Command Options

Option	Description
Start point	Specifies the starting point of the text: • Pick a point • Enter x,y coordinates This actually is the text insertion point; the actual starting point is affected by the justification.
Style	Changes the text style to use for the text; prompts you: **Text style to use (or '?')<Standard>- (enter an option:)** • Enter the name of a text style • Press **?** to see a list of style names stored in the drawing • Press **Enter** to accept the default style See the Style (6.27.53)command.

?	Lists the names of styles found in the drawing; prompts you: T**ext style(s) to list** - **(specifies the styles to list:)**
	• ***** - lists all styles
	• **name*** - lists all names that begin with name
	• **name** - lists the style named name
	`Text styles:`
	`Style name: Standard Font typeface: Arial  Height: 0 Wid`
	`Current text style: Standard`

Align	Specifies align justification: aligns text between two points; prompts you: 1. **Text start point** - **(specify the staring point of the text.)** 2. **Text end point** - **(specify the end point for the text.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i This option suppress the height and angle prompts, because the height and angle are determined by the two pick points: • Rotation angle of text is determined by the angle between the two points • Height of text is determined proportions More characters in the same distance make the text shorter. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Fit	Specifies fit justification: fits text between two points and prompts for the height; prompts you: 1. **Text start point** - **(specify the staring point of the text.)** 2. **Text end point** - **(specify the end point for the text.)** 3. **Height of text** - **(specify the text height.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i This option suppresses the angle prompt, because the angle is determined by the two pick points.Entering certain heights can cause the text to be excessively elongated or squeezed. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Use this justification to force text to fit in an area, such as a closet.
Center	Specifies center justification: centers text in the middle of the baseline; prompts you: **Center point of text** - **(specify the center point on the baseline.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Middle	Specifies middle justification: centers text on the middle of the text; prompts you: **Middle point of text** - **(specify the middle point.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Right	Specifies right justification: right-justifies each line of text; prompts you: **Right point of text** - **(specify the right point on the baseline.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Justify	Displays all justification modes; prompts you: **Text: Style/ Align/ Fit/ Center/ Middle/ Right/ TL/ TC/ TR/ ML/ MC/ MR/ BL/ BC/ BR/<Start point>** - **(enter an option:)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • TL - top left • TC - top center • TR - top right • ML - middle left • MC - middle center (equivalent to Middle) • MR - middle right • BL - bottom left (equivalent to Start Point) • BC - bottom center (equivalent to Center) • BR - bottom right (equivalent to Right)
Height of text	Specifies the height of the text: enter a number or pick two points.This prompt is not displayed under the following conditions: • Style has set the text to a height other than 0 • Justification mode is Align If the text style is annotative the height value defines the paper space height of the text; the model space height depends on the current annotation scale as defined by the CANNOSCALE and CANNOSCALEVALUE system variables.

Rotation angle of text	Specifies the angle of the text: • Enter a number • Pick two points This prompt is not displayed when the justification is set to Align or Fit.
Text	Specifies the text: 1. Enter a line of text, and then press **Enter**. 2. The prompt repeats; enter more text; each additional line of text is positioned under the previous line. 3. Move the cursor to another location in the drawing, and then enter more text 4. Press **Enter** twice to end the command (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Grips Editing

Text can be moved directly through grips:

1. Select the text face. Notice that it has one grip at its insertion point.
2. Drag the grip to move the text.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image764.png}) Aligned and fitted text have two grips:

- One grip moves the text.
- The second grip stretches and rotates the text. Aligned text also changes its height.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image481.jpg})

Figure 6.2394

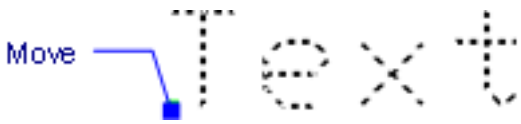


Figure 6.2395



Procedures

- To create single line text (1.21.20)
- To create multiline text (1.21.20)
- Defining text system variables (1.21.17)

Related Commands

Text (1.21.20) - places single-lines of text in drawings. MText (6.21.35) and -MText (6.21.33) - places paragraph text. AttDef (6.4.27) - defines attribute text. DdEdit (6.10.14) - edits text. Style (6.27.53) - creates and modifies text styles through the Drawing Explorer. -Style (6.27.54) - creates and modifies text styles through the command bar. QText (6.25.6) - toggles between text and rectangular representations. Change (6.9.10) - changes the position of text. TxtExp (6.29.27) - explodes text into polylines. Spell (6.27.63) - checks the spelling of text in drawings. Properties (6.24.49) - modifies the properties of text. Find (6.14.8) - finds and replaces text.

Figure 6.2396



Figure 6.2397

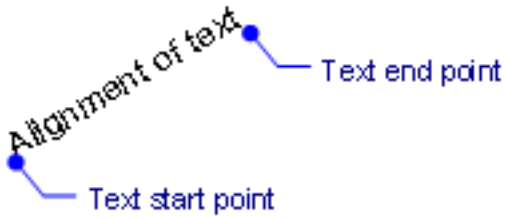


Figure 6.2398



Figure 6.2399



Figure 6.2400



Figure 6.2401



Figure 6.2402



Figure 6.2403



Figure 6.2404

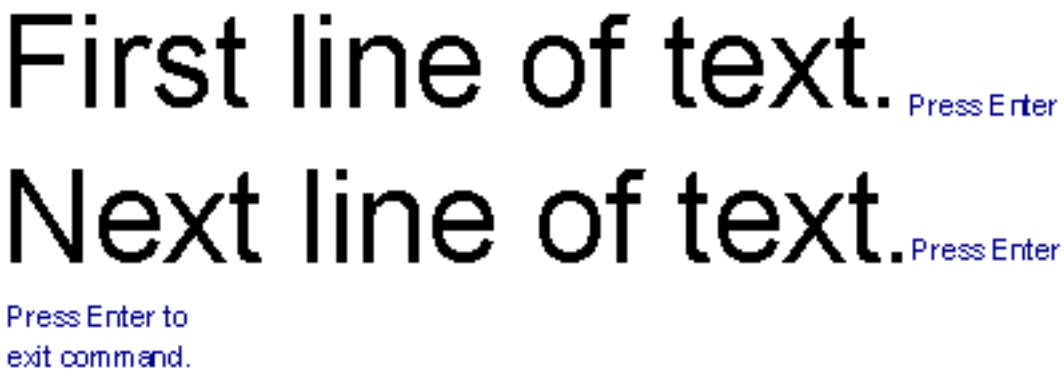


Figure 6.2405



6.29.13 TextScr

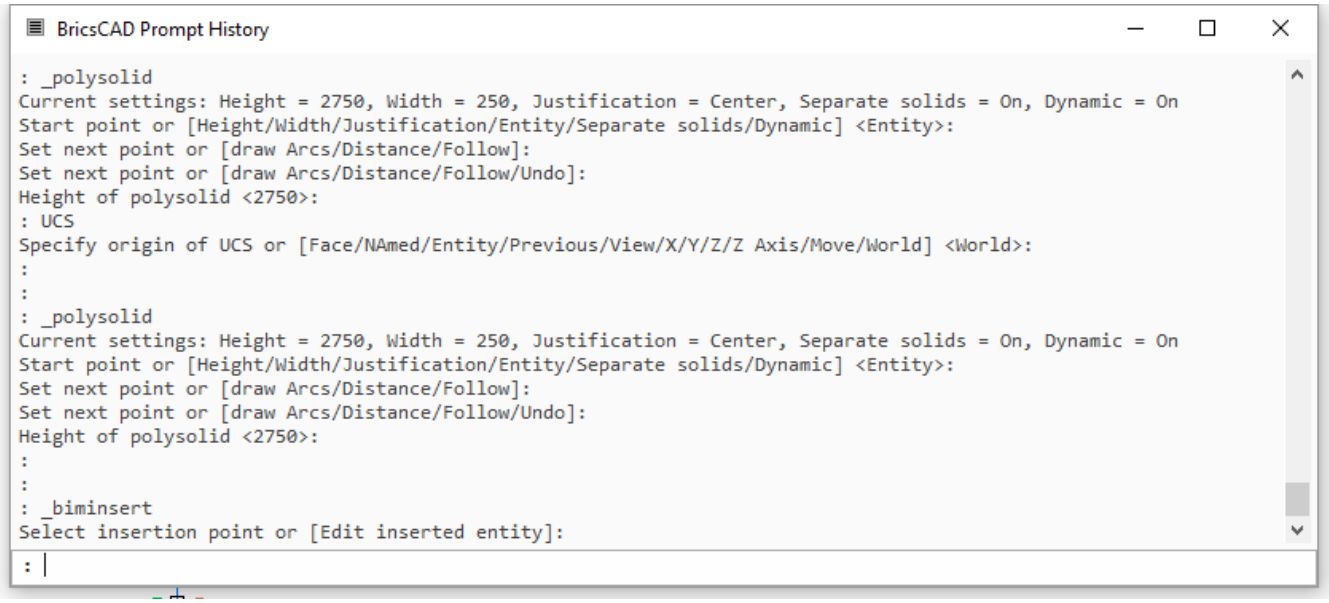
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Displays the Prompt History window (short for "text screen"); this window displays the last 256 lines of command and prompt text. **Note:** To display greater or fewer lines of command history, change the value of the ScrlHist variable.

Accessing the Command

command bar: **textscr** transparent: **'textscr** shortcut: **F2** menu bar: **View | Prompt History Window : TextScr** Displays window: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/PromptHistoryWindo
To close the window:

- Click the **X**in upper right corner
- Press the **F2** function key
- Enter the GraphScr (6.15.20) command

Figure 6.2406

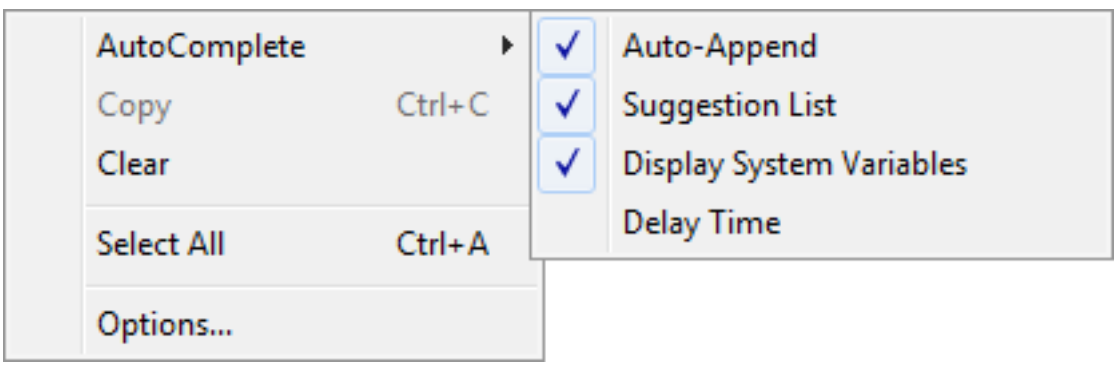


Shortcut Menu Options

Right-click the Prompt History window for the following shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom

Option	Description
AutoComplete	Sets the autocomplete options; see the AutoComplete (6.4.33) command
Copy	Copies selected text to the Clipboard: 1. Select text by highlighting with the mouse, or else using the **Select All** option 2. Choose the **Copy** option
Clear	Clears the history, after you answer **Yes** to the warning dialog box. **Caution**: this option clears the history from all open drawings.
Select All	Selects all of the text in the history window; useful before using teh Copy command
Paste	Pastes text in the Command Bar.This option is available only when the Clipboard contains text.
Options	Displays the Display Section of the Options dialog box: **Scroll History** - specifies the number of lines of text to remember

Figure 6.2407



Related Commands

AutoComplete (6.4.33) - sets the command line autocomplete mode. CommandLine (6.9.34) - displays the command bar. CommandLineHide (6.9.36) - hides the command bar. GraphScr (6.15.20) - brings the graphics window to the foreground.

Figure 6.2408



6.29.14 TextToFront

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the draw order of all text and/or dimensions in the drawing to display on top of all other entities. **Note:** To bring hatches to the back, use theHatchToBack (6.16.6) command; to bring any other entity to the front or back, use theDrawOrder (6.10.38) command.

Accessing the Command



command bar: **texttofront** menu bar: **Tools | Draw Order | Bring Text to Front Tools | Draw Order | Bring Dimensions to Front Tools | Draw Order | Bring Text and Dimensions to Front** ribbon:**Draw | Dimensions | Bring Dimensions to Front Draw | Text | Bring Text and Dimensions to Front** toolbar: **Draw Order | : TextToFront Prompts you in the command bar:** Bring to front [Text/Dimensions/Both] <Both>: **(Enter an option, or press Enter to bring both texts and dimensions to front)**

Command Options

Option	Description
Text	Brings text entities only to front
Dimensions	Brings dimension entities only to front
Both	Brings both texts and dimensions to front

Related Commands

DrawOrder (6.10.38) - changes the display order of overlapping entities. DrawOrderByLayer (6.10.37) - sets draw order of overlapping objects through layer names. HatchToBack (6.16.6) - sets the draw order of all hatch entities in the drawing to display behind all other entities.

Figure 6.2409



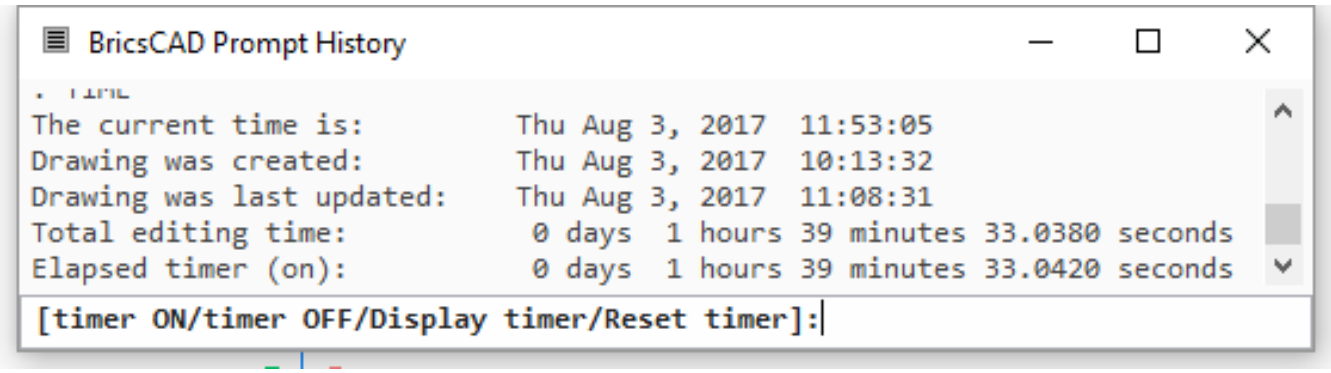
6.29.15 Time

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Reports on time in the drawing.

Accessing the Command

 command bar: **time** transparent: **'time** alias: **ti** menu bar: **Tools | Time Variables** ribbon: **Tools | Inquiry | Time Variables** toolbar: **Inquiry |**  **: Time** Reports the time in the Prompt History window: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/TimePromptHistoryWindow.png}) **Then prompts you:** [timer ON/timer OFF/Display timer/Reset timer]: **(Enter an option, or else press Enter to exit the command)** Press the F2 function key to close the Prompt History window. This command relies on the accuracy of the computer's built-in clock.

Figure 6.2410



Command Options

Option	Description
timer ON	Turns on the user timer.This can be used to time operations in BricsCAD.
timer OFF	Turns off the user timer.
Display timer	Displays the time again.Updates the time.
Reset timer	Sets the user timer to zero.

Related Command

Status (6.27.46) - reports the status of the drawing.

6.29.16 Tips

   Display of the Tips panel, which provides helpful advice on using certain commands. (There is no command named Tips; to display this panel, right-click a toolbar or the ribbon, and then choose Tips from the context menu.)

Accessing the Command

Context menu: **right-click a toolbar or the ribbon, and then choose Tips** Displays a panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tips-panel.png}) When you start certain commands, the panel displays a short animation and additional information on using the command. Shown below is the result for the PolySolid command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tips-example.png})

Figure 6.2411

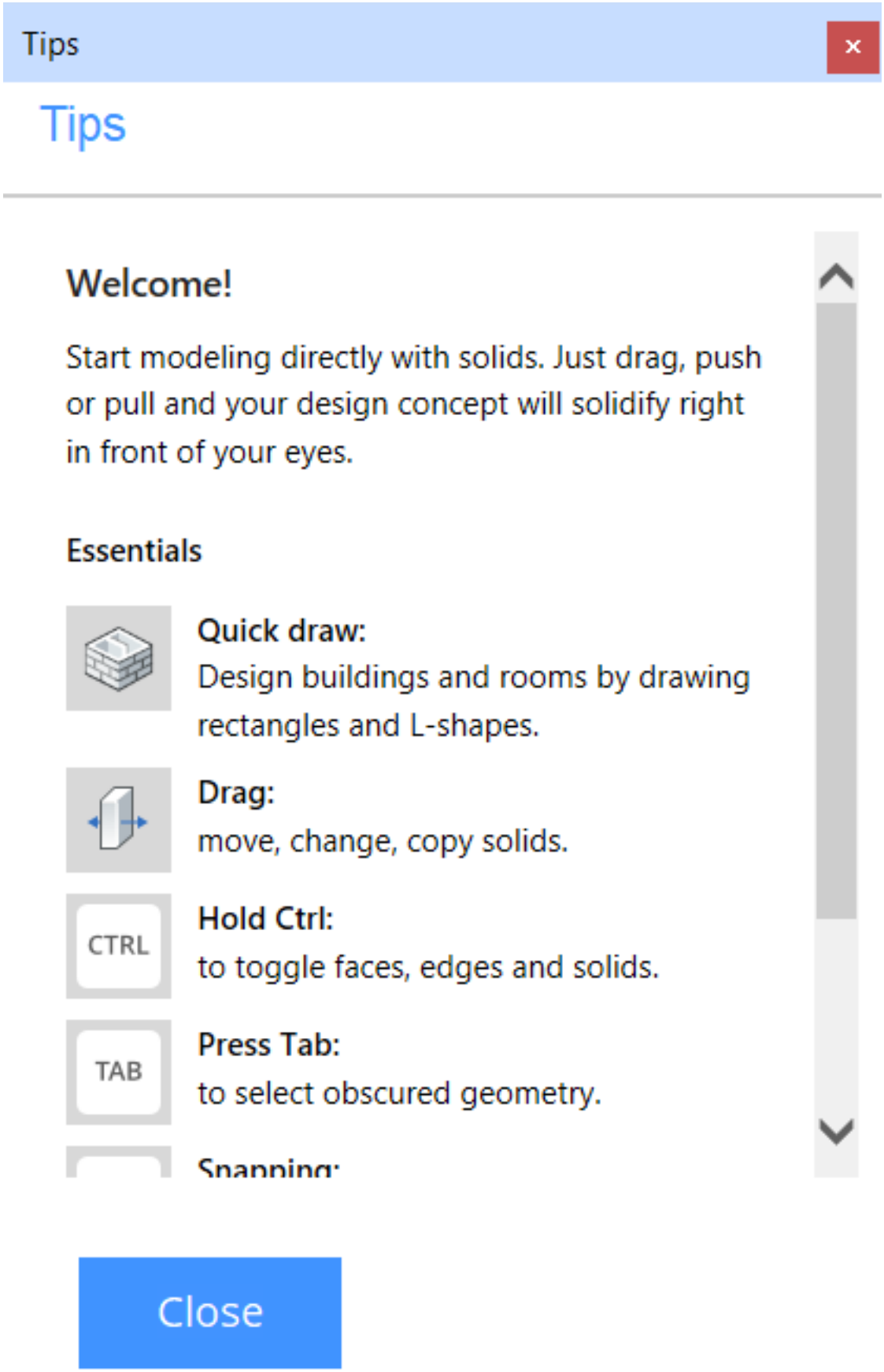
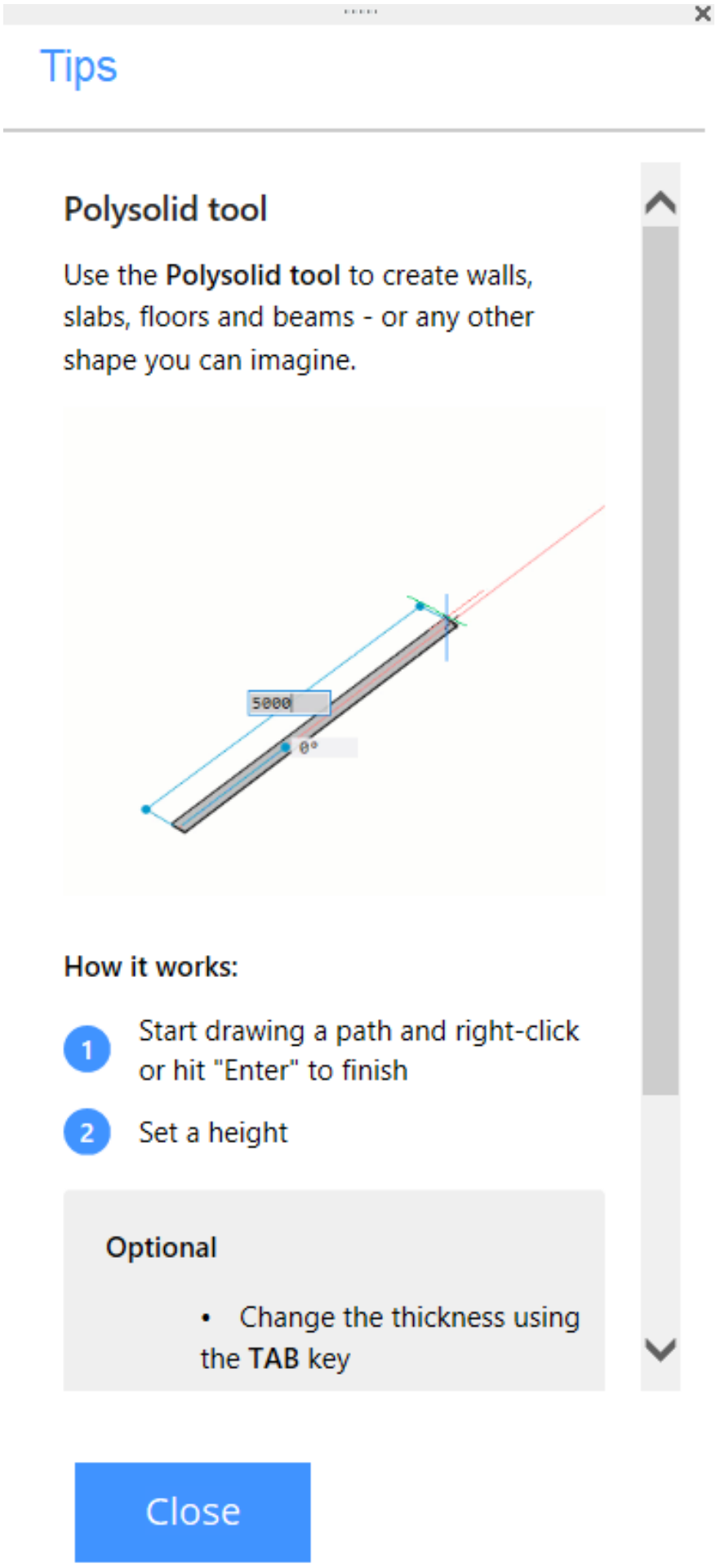


Figure 6.2412



Command Options

Option	Description
Close	Closes the panel

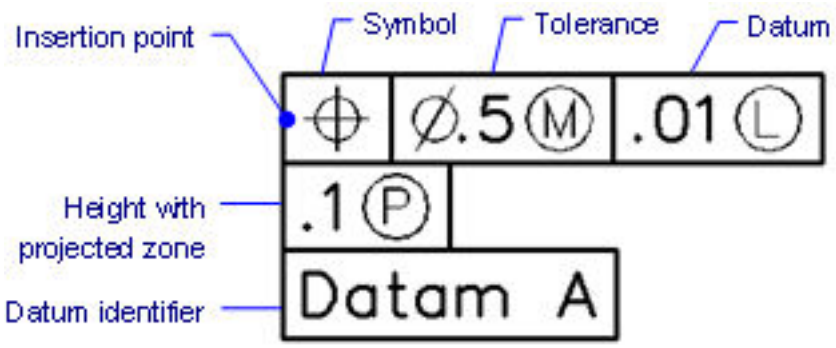
Related Commands

Help (6.16.8)- displays online help for the current command when you press **F1**

Figure 6.2413



Figure 6.2414



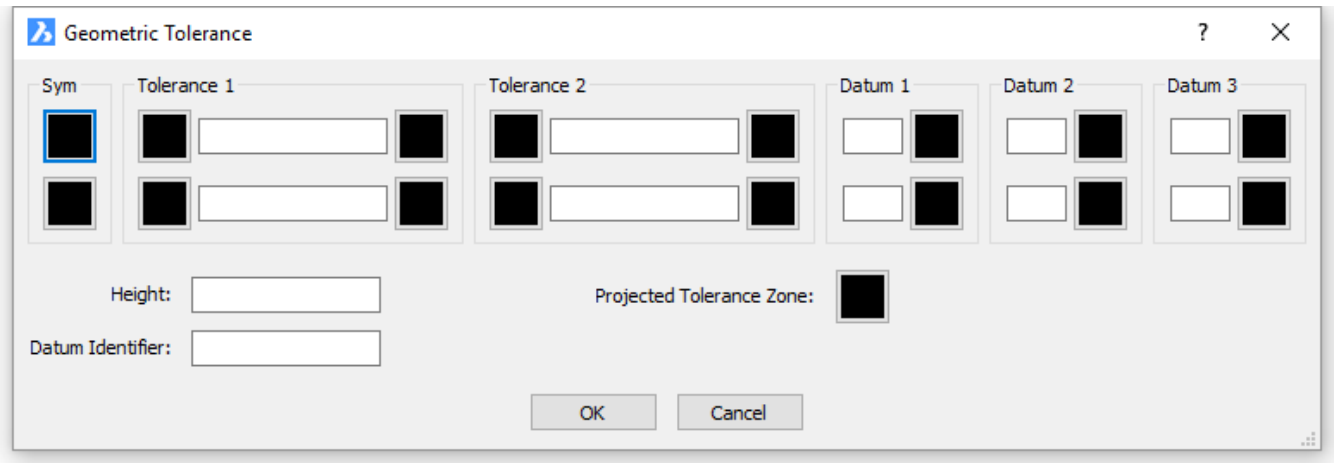
6.29.17 Tolerance

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Places tolerance symbols in drawings. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom
Note: Tolerance symbols are also placed by the QLeader (6.25.1) command.

Accessing the Command

command bar: **tolerance** alias: **tol** menu bar: **Dimensions | Tolerance** ribbon: **Draw | Dimensions | Tolerance** toolbar: **Dimensions |** **: Tolerance** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/
Select one or more tolerance symbols and enter values, and then click OK.Prompts you in the command bar: Select location for tolerance: **(Specifies the insertion point for the tolerance symbol)**

Figure 6.2415



Geometric Tolerance Dialog Box Options

Option	Description
Symbol	Inserts a tolerance symbol; displays dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S 1. Choose a symbol 2. Click **OK** 3. Repeat to add more symbols
Tolerance 1 **and** Tolerance 2	Specifies tolerance specification: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Diameter** - toggles the diameter symbol • **Value** - specifies the tolerance value • **Material condition** - displays dialog box when clicked:
Datum 1,Datum 2, **and**Datum 3	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M Specifies the datum reference: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Value** - specifies the datum value. • **Material condition** - click to display the Material Condition dialog box.
Height	Specifies the height of the tolerance symbols.
Datum Identifier	Specifies datum indentifier, such as Datum A.
 Projected Tolerance Zone	Toggles the projected tolerance zone symbol

Tolerance Symbols

Symbol	Characteristic	Type
	Position	Location
	Concentricity or Coaxiality	Location
	Symmetry	Location
	Parallelism	Orientation
	Perpendicularity	Orientation
	Angularity	Orientation

	Cylindricity	Form
	Flatness	Form
	Circularity or Roundness	Form
	Straightness	Form
	Profile of a surface	Profile
	Profile of a line	Profile
	Circular Run-out	Run-out
	Total Run-out	Run-out

Material Condition Symbols

Symbol	Definition
	At maximum material condition (MMC), a feature contains the maximum amount of material stated in the limits.
	At least material condition (LMC), a feature contains the minimum amount of material stated in the limits.
	Regardless of feature size (RFS) indicates that the feature can be any size within the stated limits.

Grips Editing

Tolerances can be edited directly through grips:

1. Select the tolerance. Notice that it has one grip.
2. Drag the grip to move the tolerance.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image482.jpg})

Figure 6.2416



Procedures

- To create a geometrical tolerance frame (1.21.16)

Related Commands

Leader (6.20.20) - attaches tolerances to leader lines. DimOrdinate (6.11.15) - draws x and y ordinate dimensions.

Figure 6.2417

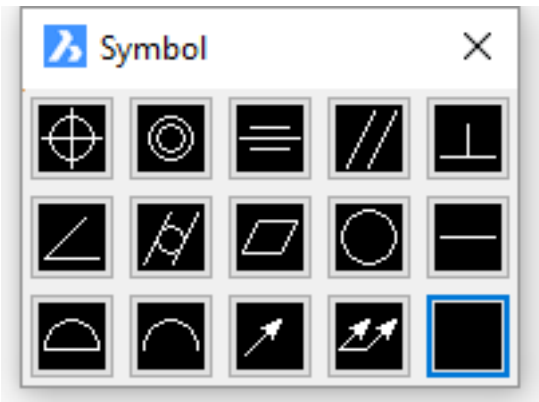


Figure 6.2418



Figure 6.2419

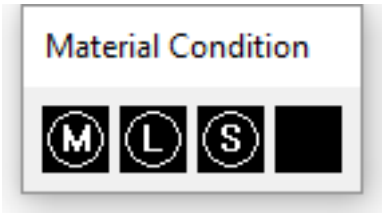


Figure 6.2420



Figure 6.2421



6.29.18 Toolbar

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the display of toolbars at the command line.

Accessing the Command

command bar: **toolbar** alias: **-toolbar : Toolbar Prompts you in the command bar:** Enter Toolbar name or [All] <All>: **(Enter the name of a toolbar, or press Enterto turn all toolbars on or off)** Enter an option [Show/Hide] <Show>: **(Choose an option)**

Command Options

Option	Description
Enter toolbar name	Specifies the name of a toolbar.
ALL	Specifies all toolbars.
Show	Displays the specified toolbar(s).
Hide	Hides the specified toolbar(s).

Procedures

- Toolbars (1.12.3)
- Toolbar Flyouts (1.12.3)
- Opening a toolbar (1.12.3)
- Closing a toolbar (1.12.3)
- To restore the default toolbars (1.12.3)
- To set the icon size (1.12.3)
- To set the position and visibility property of a toolbar (1.12.3)

Related Commands

CommandLine (6.9.34) - shows the command line. CommandLineHide (6.9.36) - hides the command line. Customize (6.9.58) - customizes toolbars. CuiLoad (6.9.56) and MenuLoad (6.21.14) - load CUI files that define toolbars and other user interface elements. CuiUnload (6.9.57) and MenuUnload (6.21.15) - load CUI files that define toolbars and other user interface elements. GraphScr (6.15.20) - brings the graphics window to the foreground. Scrollbar (6.27.10) - toggles the display of the scroll bars. StatBar (6.27.45) - toggles the display of the status bar. TextScr (6.29.13) - brings the text window to the foreground.

Figure 6.2422



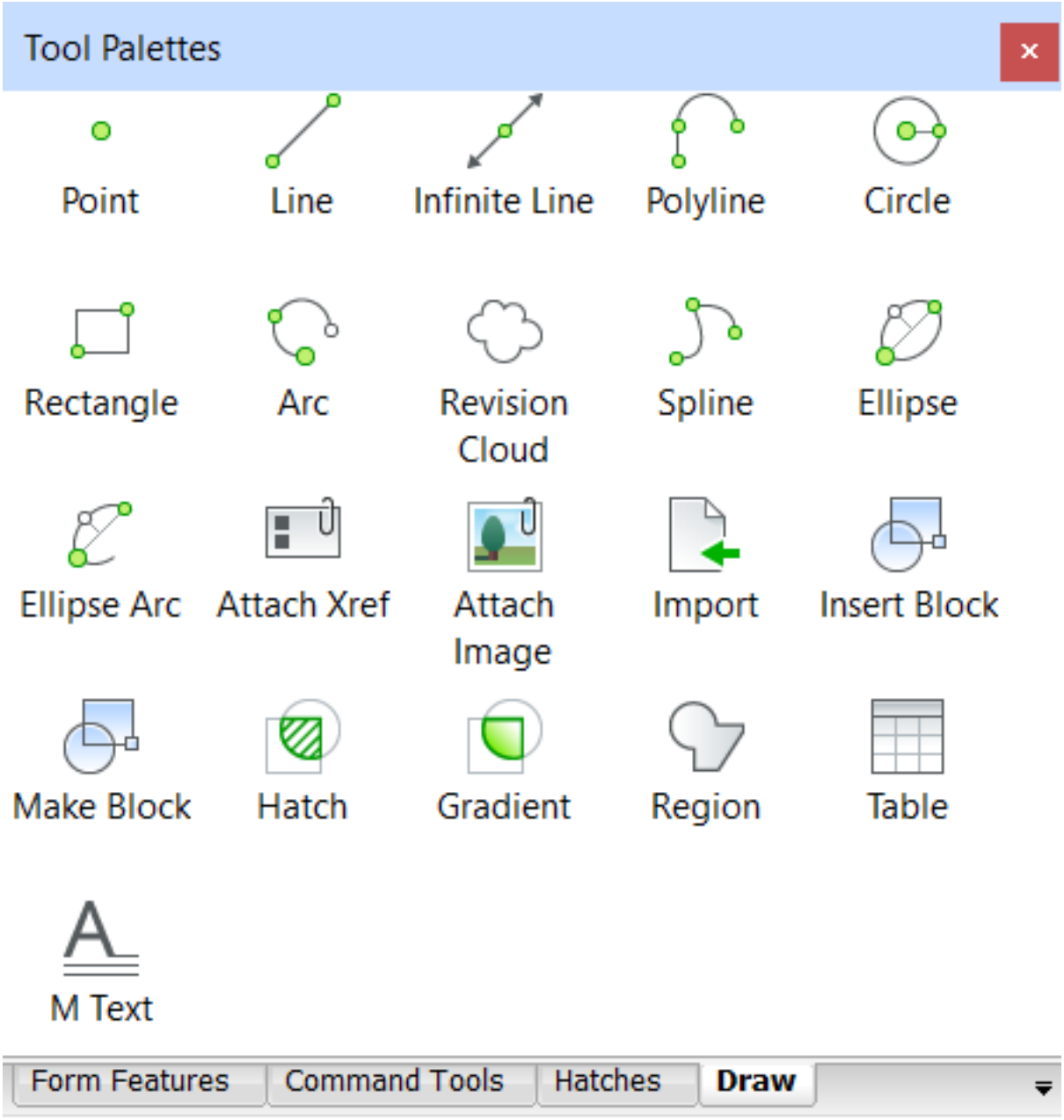
6.29.19 ToolPalettes

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Tool Palettes panel.

Accessing the Command

command bar: **toolpalettes** context menu: right-click a toolbar or the ribbon, and then choose**Tool Palettes : Tool-Palettes** Displays the Tool Palettes panel: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands panel.png}) Drag an icon from the panel into the drawing.

Figure 6.2423

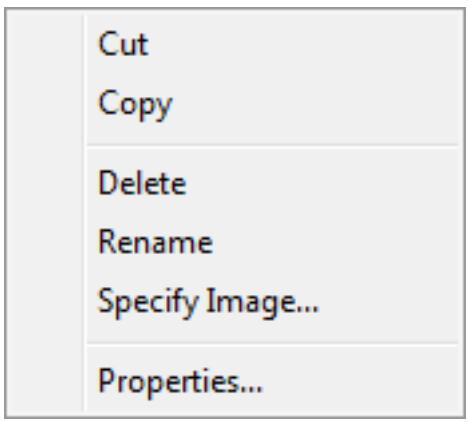


Icon Context Menu Options

Right-click an icon to see this context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Comman
context-mnu-1.png})

Option	Description
Cut	Removes the icon and places it in the Clipboard; to paste, right-click a blank area of a palette
Copy	Copies the icon and places it in the Clipboard;to paste, right-click a blank area of a palette
Delete	Removes the icon and its content from the palette
Rename	Renames the icon
Specify Image	Replaces the icon with a raster image in one of the following formats: • BMP • GIF • JPEG and JPG • PNG • TIFF 1. Opens the Select Image File dialog box; choose a file 2. Click **Open**
Properties	Displays the Tool Properties dialog box; the content varies according to the type of tool selected; see below

Figure 6.2424

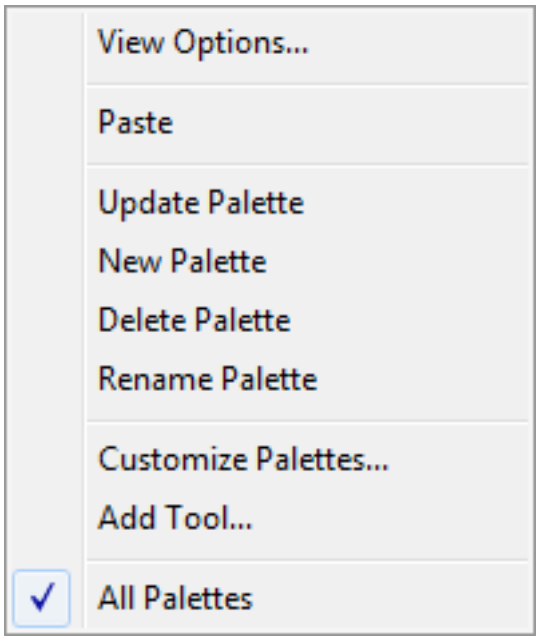


Palette Context Menu Options

Right-click a blank part of a palette to see this context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS
context-mnu-2.png})

Option	Description
View Options	Displays the View Options dialog box for determining how the icons are displayed by the palette; see below
Paste	Pastes the content of the Clipboard, when the Cut or Copy options were used previously to copy an icon and its content to the Clipboard; does not paste any other kind of content
Update Palette	Refreshes icons
New Palette	Creates a new, blank palette
DeletePalette	Removes the current palette, after you acquiesce to a warning dialog box
Rename Palette	Renames the current palette; enter a new name: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tp-rename.png})
Customize Palettes	Determines which palettes are loaded into the program, and which ones are part of a group; displays the Customize dialog box, which has nothing to do with the other Customize dialog box; see below.
Add Tool	Displays the Workspaces tab of the real Customize dialog box
All Palettes	Toggles the display of all palettes: • **On** - displays all palettes • **Off** - displays only those palettes that are part of a group

Figure 6.2425

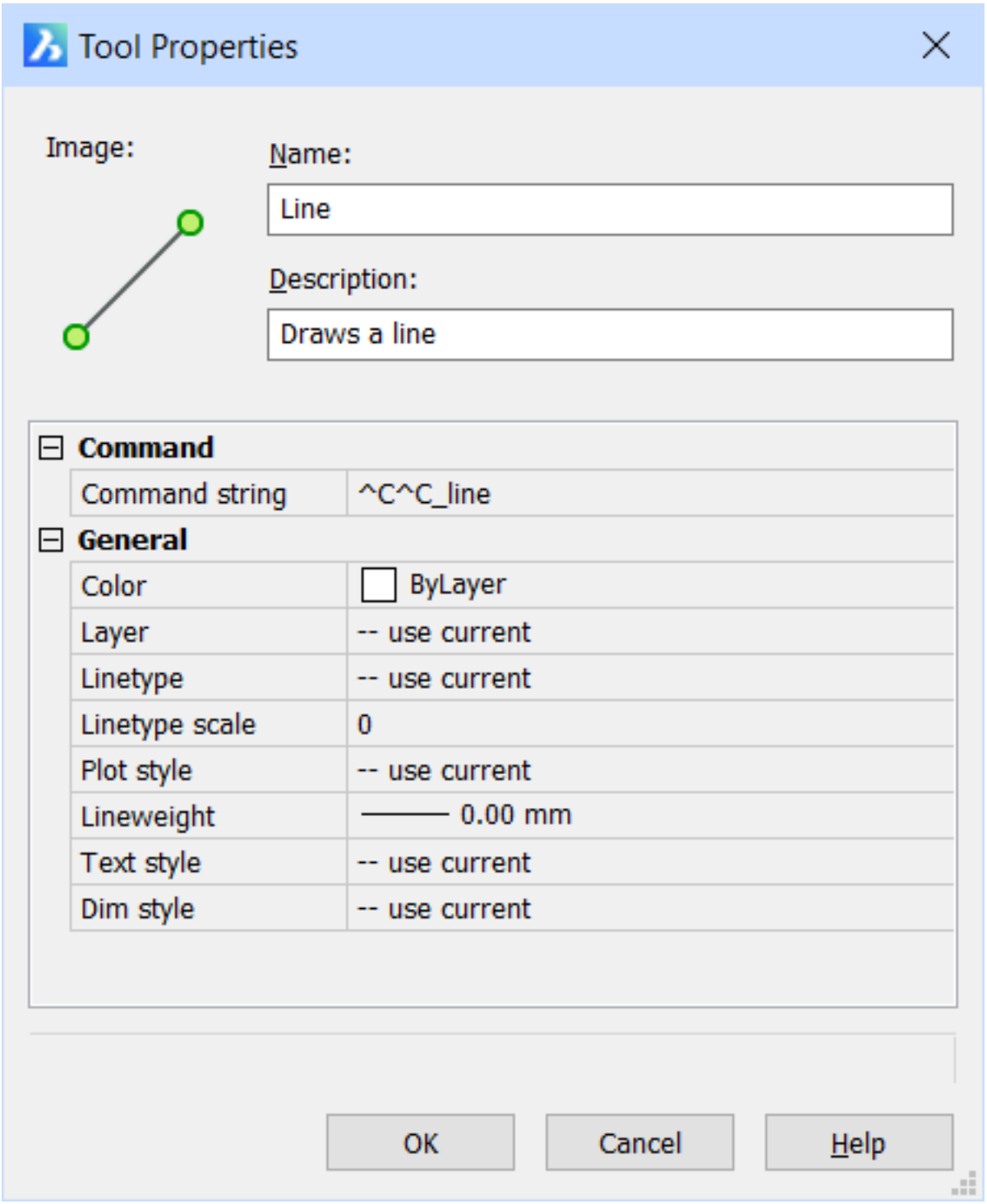


Properties Dialog Box

To access this dialog box, right-click an icon and then choose **Properties**: (\vref {fig:C:/Dev/Task-2448/images/staticbricsysco toolproperties-dlg.png})

Option	Description
Name	Names the icon
Description	Describes the function of the icon
Command String	Specifies the macro to run when you click the icon; you can use any macro characters used for menus and toolbar buttons
General	Specifies the properties of the entity; the properties vary, depending on the tool

Figure 6.2426

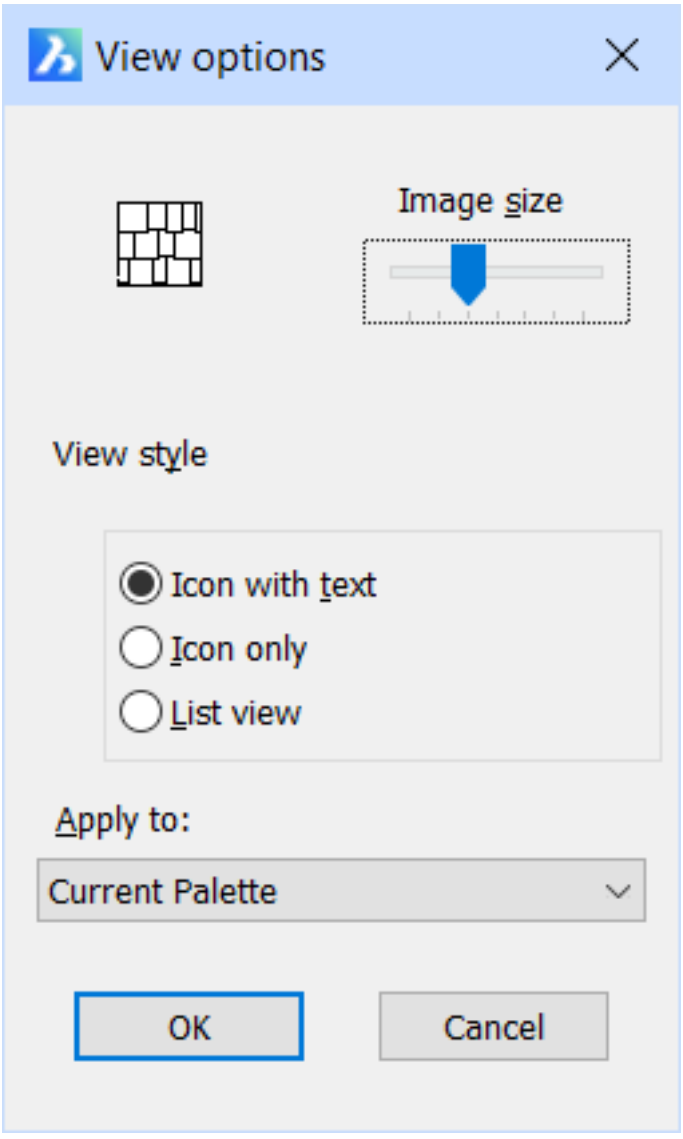


View Options Dialog Box

To access this dialog box, right-click a blank area and then choose **View Options**: (\vref {fig:C:/Dev/Task-2448/images/staticbri viewoptions-dlg.png})

Option	Description
Image Size	Specifies the size of the icons; drag the slider to make them larger and smaller
View Style	Determines how the icons and accompanying text are displayed • **Icon with text** - display icons with text • **Icon only** - display only icons, to save space • **List view** - display small icons with text in a list, as shown below: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/tp-list.png})
Apply To	Determines which palettes the changes should apply to: • **Current palette** - applies the change(s) to all icon on the current palette • **All palettes** - applies the changes to all palettes

Figure 6.2427

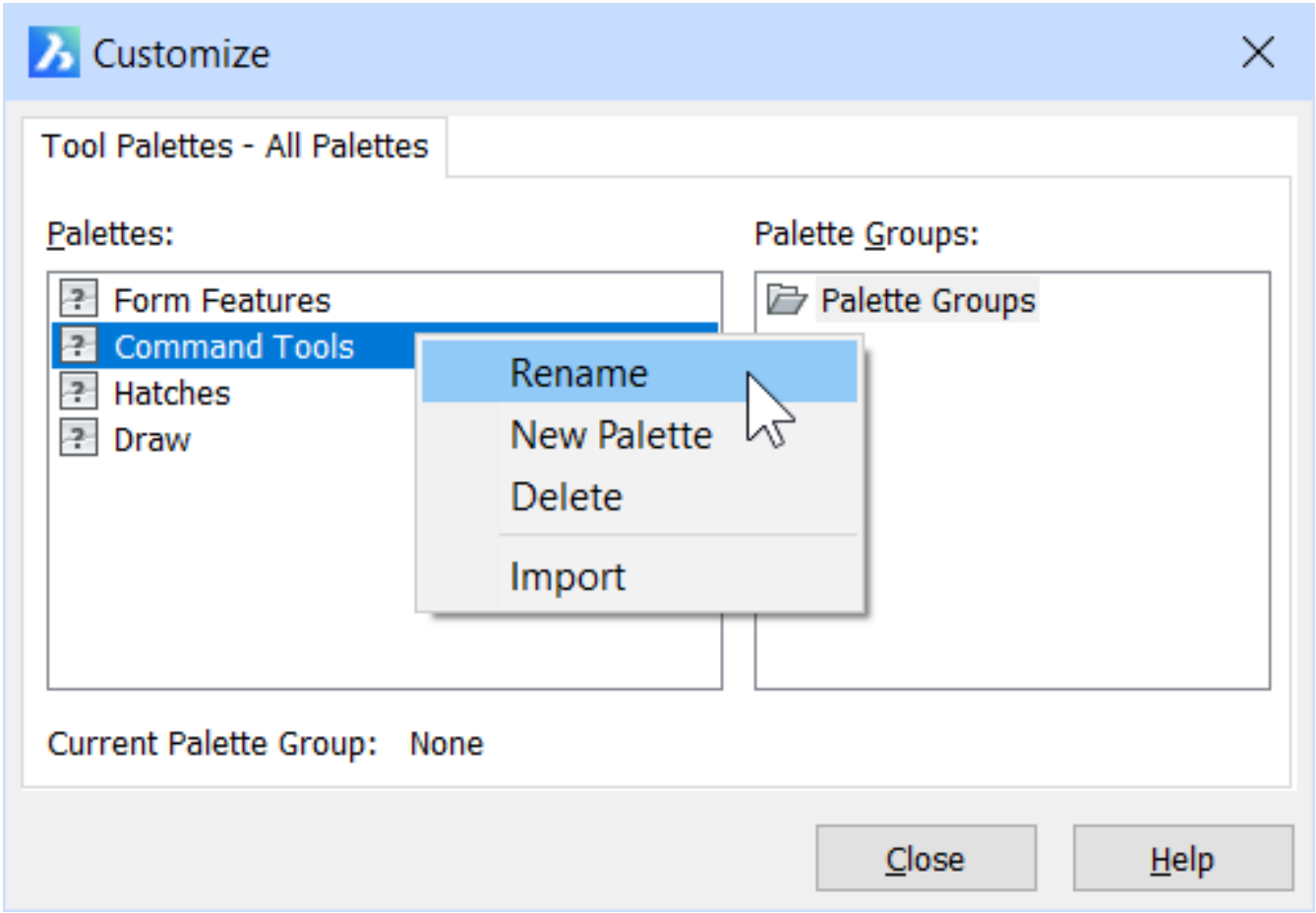


Customize Dialog Box

To access this dialog box, right-click a blank area and then choose **Customize**: (\vref {fig:C:/Dev/Task-2448/images/staticbrics customize-dlg.png}) To add a palette to a group, drag it form the Palettes pane to the Palette Groups pane. Once in a group, you can turn groups of palettes on and off by turning the associated group on and off.

Option	Description
Rename	Renames the palette
New Palette	Creates a new, blank palette
Delete	Removes the palette
Import	Imports tool palettes in the following formats: • **XTP** - XML-format files • **BTC** - BricsCAD tool palette format • **ATC** - AutoCAD tool palette format To export a palette, follow these steps: 1. Create a new group 2. Add one or more palettes to a group 3. You could rename the group ”Export” 4. Right-click the group, and then choose **Export**

Figure 6.2428



Procedures

- To set the view options (1.13.7)
- Adding a new tool palette (1.13.7)
- To open the customize dialog box (1.13.7)
- Importing a tool palette (1.13.7)
- Working with tool palette groups (1.13.7)
- To rename a tool palette (1.13.7)
- To delete a tool palette (1.13.7)
- Adding a hatch pattern (1.13.7)
- Adding a block (1.13.7)
- Adding a command (1.13.7)
- Editing a tool (1.13.7)
- Managing tools (1.13.7)

Related Commands

ToolPalettesClose (6.29.20) - closes the Tool Palettes panel. -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line. TpNavigate (6.29.23) - allows to load a tool palette or tool palette group by name. If not already open, displays the Tool Palettes panel.

Figure 6.2429



Figure 6.2430

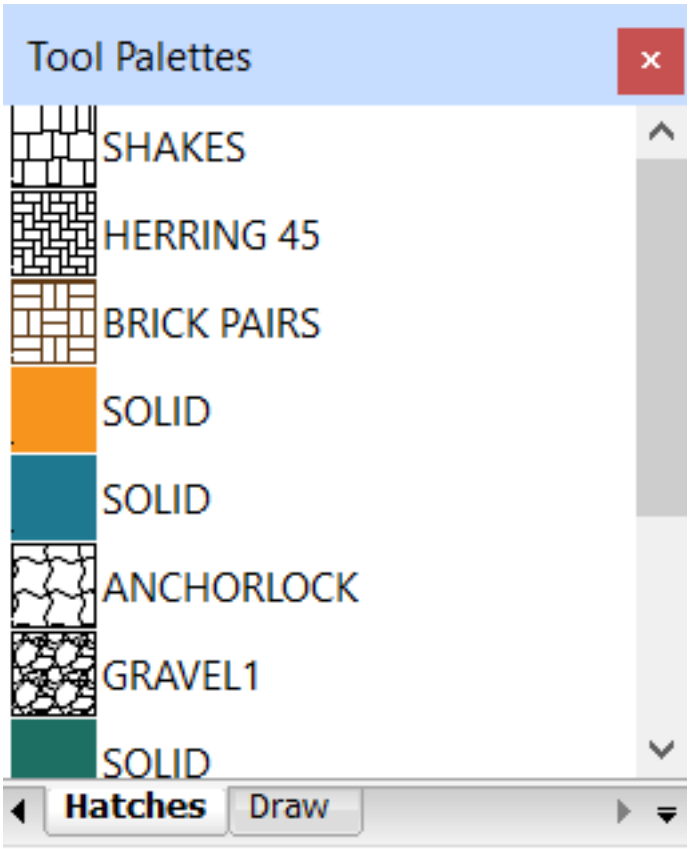


Figure 6.2431



6.29.20 ToolPalettesClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes the Tool Palettes panel.

Accessing the Command

command bar: **toolpalettesclose : ToolPalettesClose** Closes the Tool Palette panel. Use theToolPalettes (6.29.19) command to open it again.

Command Options

No Options

Related Commands

ToolPalettes (6.29.19) - opens the Tool Palettes panel. -ToolPanel (6.29.21) - allows to control the display of tool panels, such as the Properties Bar, the Ribbon or the Sheet Sets panel from the command line.

Figure 6.2432



6.29.21 -ToolPanel

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggle the display of panels -- Properties, Ribbon, Sheet Sets, and son --at the command line.

Accessing the Command

command bar: **-toolpanel : -ToolPanel Prompts you in the command bar:** Enter name of Tool Panel to show or hide or [? to list all panels]:**(Type the name of a tool panel, or press ?to list the names of panels)** Enter an option [Show/Hide/Toggle] <Show>: **(Choose an option, or else press Enter to accept the default option)**

Command Options

Option	Description
Enter name of Tool Panel to show or hide	Specifies the name(s) to panels to show or hide; enter the name of one panel.
? to list all panels	Lists the names in alphabetical order of the panels in this edition of the program: `Tool Panels: Mechanical Browser Attachments BIM Composi`
Show	Displays the tool panel
Hide	Hides the tool panel
Toggle	Toggles the display of the tool panel: if it is on, then this option turns it off -- and vice versa

Procedures

- To open/close a panel (1.12.2)

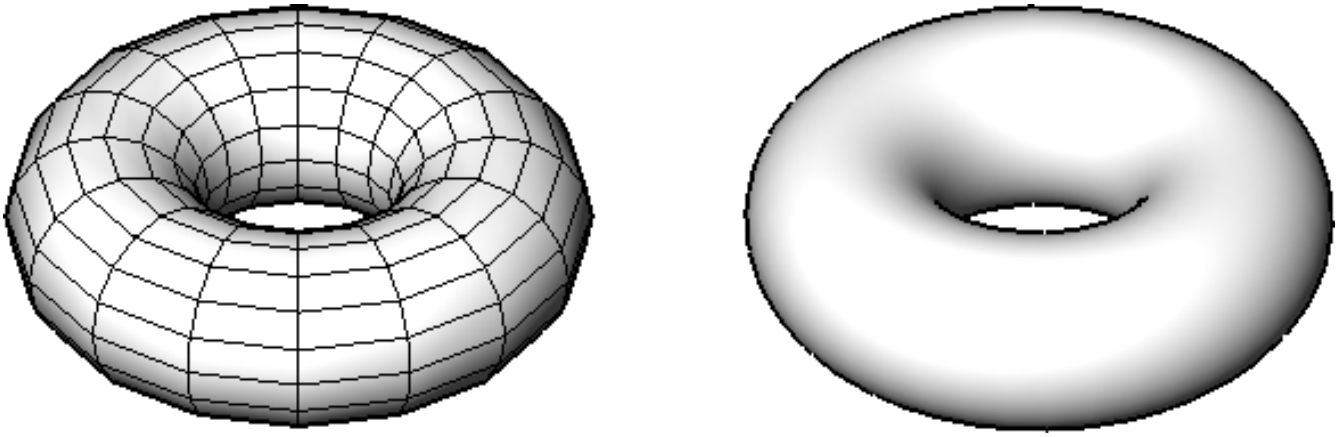
Related Commands

AttachmentsPanelOpen (6.4.24) - opens the Attachments panel. AttachmentsPanelClose (6.4.26) - closes the Attachments panel. ComponentsPanelOpen (6.9.61) - opens the Components panel. ComponentsPanelClose (6.9.38) - closes the Components panel. ContentBrowserclose (6.9.40) - closes the Content Browser panel. ContentBrowserOpen (6.9.41) - opens the Content Browser panel. CommandLine (6.9.34) - displays the command line. CommandLineHide (6.9.36) - hides the command line. LayersPanelClose (6.20.5) - opens the Layers panel. LayersPanelOpen (6.20.6) - closes the Layers panel. MatBrowserClose (6.21.5) - closes the materials browser. MatBrowserOpen (6.21.6) - opens the materials browser, which shows an overview of available high-resolution render materials. MechanicalBrowserClose (6.8.28) - closes the Mechanical Browser panel. MechanicalBrowserOpen (6.8.29) - opens the Mechanical Browser panel. ParametersPanelOpen (6.24.8) - opens the Parameters panel. ParametersPanelClose (6.8.30) - closes the Parameters panel. Properties (6.24.49) - displays the Properties panel with properties of selected entities. PropertiesClose (6.24.50) - closes the Properties panel. ReportPanelClose (6.8.31) - closes the Report panel. ReportPanelOpen (6.8.32) - opens the Report panel. Ribbon (6.26.31) - displays the ribbon. RibbonClose (6.26.33) - hides the ribbon. SheetSet (6.27.31) - Opens the Sheet Sets panel. SheetSetHide (6.27.33) - closes the Sheet Set panel. StructurePanel (6.27.50) - opens the Structure panel. StructurePanelClose (6.27.52) - closes the Structure panel. ToolPalettes (6.29.19) - opens the Tool Palettes panel. ToolPalettesClose (6.29.20) - closes the Tool Palettes panel.

Figure 6.2433



Figure 6.2434



6.29.22 Torus

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draw a torus (donut shape):

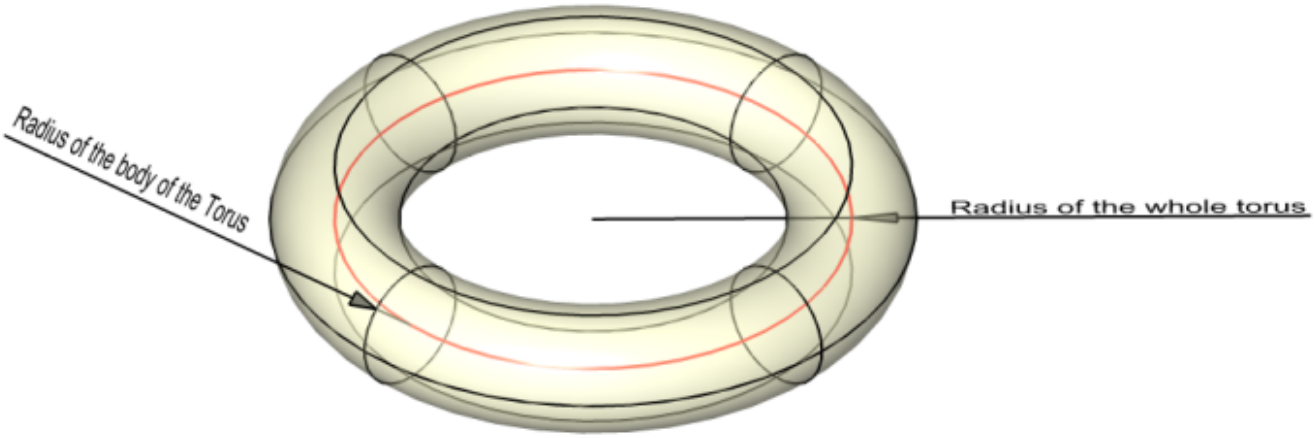
- Classic: torus is drawn from a 3D polyface mesh (left).
- Shape, Pro and Platinum: torus is drawn as a 3D solid (right).

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/T/torus-eg.png}) **Note:** Donuts drawn by this command look good enough to eat, but are not edible in fact.

Accessing the Command

 command bar: **torus** menu bar: **Model| 3D Solids | Torus** ribbon:**Solid | Create | Primitive | Torus** toolbar: **3D Solids |**  **: Torus Prompts you in the command bar:** Center of whole torus: <0,0,0> **(Pick a point, or enter x,y,z coordinates.)** Set Radius of whole torus or [Diameter]: **(Specify the outer radius, or press D)** Set radius of body of torus [Diameter]: **(Specify the tube radius, or type D)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/C

Figure 6.2435



Command Options

Option	Description
Center of whole torus	Specifies the center of the torus: • Pick a point in the drawing • Enter x,y,z coordiantes • Press **Enter** to accept the default coordinates
Radius of whole torus	Specifies the overall radius of the torus; note that the distance is measured from the center of the tube (body).
Radius of body of torus	Specifies the radius of the torus body, or tube. It is not necessary for the body radius to be smaller than the whole radius.
Diameter	Specifies the diameters of the whole torus and/or body.

Related Commands

Box (6.6.17) - creates 3D boxes as solid models. Cone (6.9.63) - creates 3D cones as solid models. Cylinder (6.9.60) - creates 3D cylinders as solid models. Dish (6.10.30) - creates the lower half of 3D balls as solid models. Dome (6.10.34) - creates the upper half of 3D balls as solid models. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Pyramid (6.24.58) - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic) Revolve (6.26.30) - revolves 2D entities into 3D solid models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Sphere (6.27.64) - creates 3D balls as solid models. Subtract (6.27.56) - removes one solid model from a second. Union (6.30.6) - joins two or more solid models into a single entity. Wedge (6.33.7) - creates 3D wedges as solid models.

Figure 6.2436



6.29.23 TpNavigate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Loads a tool palette or palette group by name. If not already open, displays the Tool Palettes panel (short for "tool palette navigation").

Accessing the Command

command bar: **tpnavigate : Tpnavigate Prompts you in the command bar:** Specify tool palette to display or [palette Group]: **(Type the name of a tool palette or group)**

Command Options

Option	Description
Specify tool palette to display	Specifies the tool palette to load: enter the name of a palette. When you enter the wrong name, the program responds: **Unable to find a palette or group by that name**
palette Group	Load a palette group;prompts you: **Specify palette group to display** - **(type the name of a palette group.)**

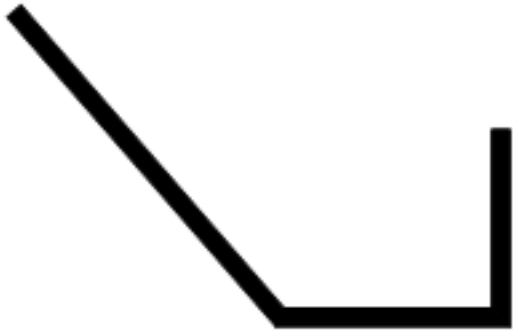
Related Commands

ToolPalettes (6.29.19) - opens the Tool Palettes panel.

Figure 6.2437



Figure 6.2438



6.29.24 Trace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws traces (like wide lines) with mitered vertices. **Note:**This command is rarely used as the PLine (6.24.27) command is more convenient. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

Accessing the Command

 command bar: **trace** menu bar: **Draw | Trace** ribbon: **Draw | Draw | Polygons | Trace** toolbar: **Draw | Polygons |**  **: Trace Prompts you in the command bar:** Width of trace <0.5000>: **(Enter a number, pick two points, or press Enter to accept the default)** Start of trace: **(Pick a point, or enter x,y coordinates)** Next point: **(Pick another point)** Next point: **(Pick another point, or else press Enter to exit the command)** This command draws the trace segments in an unusual manner:The trace is not drawn until the second pick point.The delay allows BricsCAD to determine the mitering needed at vertices, which this command does automatically. The FillMode variable affects the look of traces: (left) on and (right) off. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Trace (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/TraceFillmodeOff.png})

Figure 6.2439

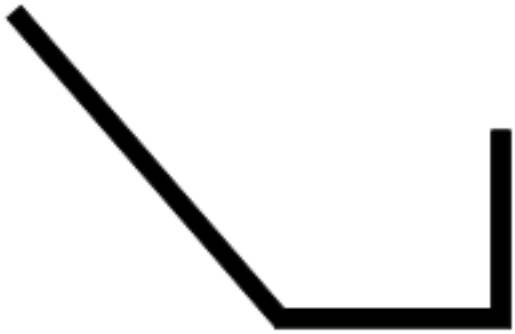
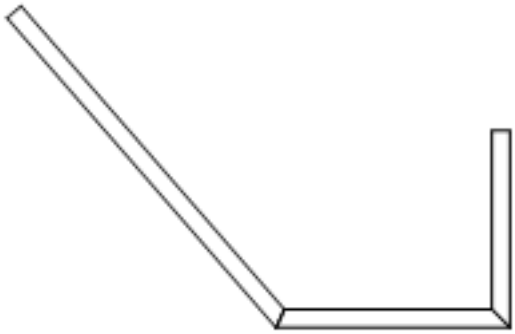


Figure 6.2440



Command Options

Option	Description
Width of trace	Specifies the width of the trace: • Enter a number • Pick two points
Start of trace	Specifies the start of the trace: • Pick a point • Enter x,y coordinates
Next point	Specifies the subsequent end points. • Pick a point Enter x,y coordinates • Press **Enter** to exit the command

Procedures

- Drawing traces (1.17.21)

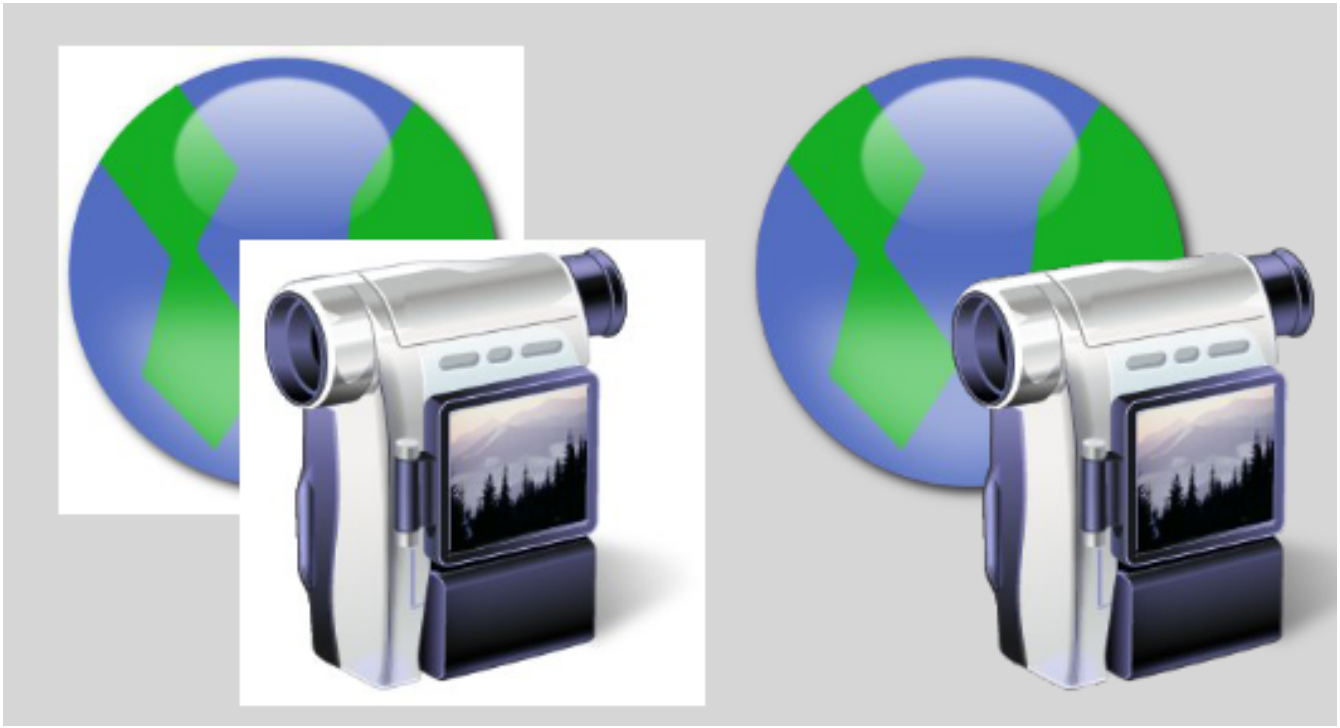
Related Commands

Line (6.20.31) - draws line segments. PLine (6.24.27) - draws polyline segments and arcs with constant and variable width. Solid (6.27.37) - draws 3- and 4-sided solid-filled areas. Donut (6.10.35) - draws wide and solid-filled circles. LWeight (6.20.41) - gives most entities visual width.

Figure 6.2441



Figure 6.2442



6.29.25 Transparency

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Toggles the transparency of monotone images. **Note:** To change the transparency of entities, use the Transparency property in the Layer (6.20.3) and Properties (6.24.49) commands. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help

Accessing the Command

 command bar: **transparency** transparent: **'transparency** ribbon:**Insert | Images | Image Transparency** toolbar: **Image** |  **: Transparency Prompts you in the command bar:** Enter transparency mode [OPN/OFF] <OFF>: **(Type ON or OFF)** When transparency is on, the background color of the image is transparent.This command works only with raster images attached to the drawing that have a monochrome palette.

Command Options

Option	Description
ON	Turns on transparency.
OFF	Turns off transparency.

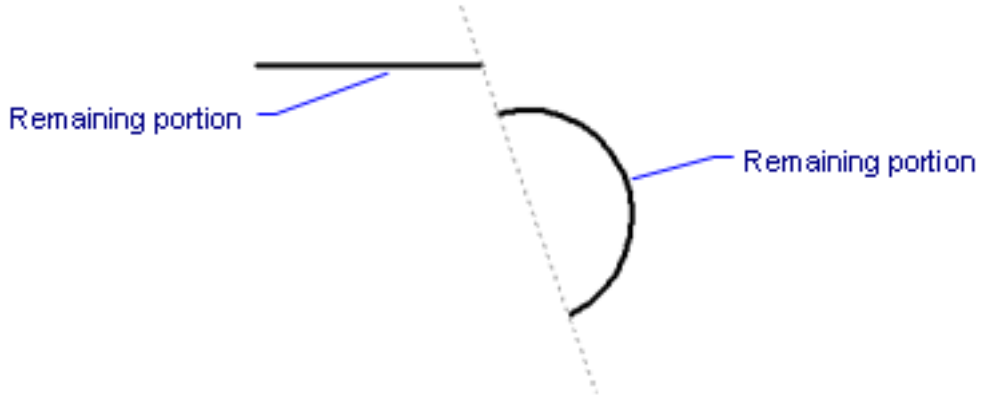
Related Commands

Image (6.17.3) - controls the parameters of the image file. ImageAttach (6.17.5) - places raster images in drawings. -ImageAttach (6.17.6) - attaches raster images to the drawing; displays prompts in the command bar. ImageAdjust (6.17.4) - changes the properties of images through the Properties palette. ImageQuality (6.17.9) - toggles the quality of the image.

Figure 6.2443



Figure 6.2444



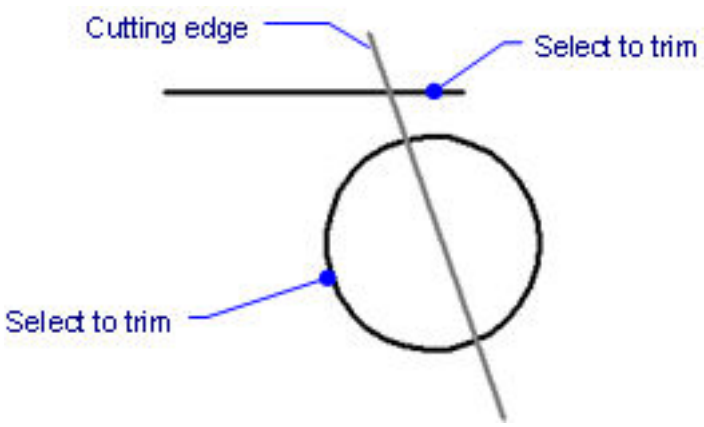
6.29.26 Trim

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Trims entities to a cutting line; extends entities when you hold down the **Shift** key. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsysc

Accessing the Command


command bar: **trim** alias: **tr** menu bar: **Modify | Trim** ribbon: **Change | Modify | Trim** toolbar: **Modify |**  **: Trim Prompts you in the command bar:** Select cutting entities for trim <ENTER to select all>: **(Choose one or more entities, or press Enter to select all entities in the drawing)** Entities in set: 1 Select cutting entities for trim <ENTER to select all>: **(Choose more entities, or else press Enter to continue the command)** (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/image356.jpg}) Select entity to trim or **s**hift-select to extend or [Fence/Crossing/Edge mode/ Projection/ eRase]: **(Pick a single entity to be trimmed, or enter an option, or press Enter to end the command)**

Figure 6.2445



Command Options

Option	Description
Select cutting entities for trim	Selects the entities to use as trimmers: • Choose one or more entities; you can even select text • Press **Enter** to select all entities in the drawing;entities can both be cutlines and be cut These entities define where crossing entities are trimmed.
Shift-select to extend	Extends the picked entities instead of trimming them: Hold down the **Shift** key while selecting entities to be extended
Select entity to trim	Chooses the entity to be trimmed.Select a single entity; only one entity can be trimmed at a time.Your pick point defines the portion of the entity to be removed: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Edge mode	Toggles edge mode; prompts you: **Extend/No extend - (type E or N.)** • **Extend** - trims entities that do not physically intersect the cutting entity. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ • **No extend** - trim only entities that intersect. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ This option is affected by the setting of the Projection option; see below.
Fence	Cuts entities interactively; prompts you: 1. **First point of fence** - **(specify the fence's starting point.)** 2. **Next fence point** - **(specify the next point; press Enter to exit this option.)** As you pick fence points in the drawing, BricsCAD immediately trims entities under the fence. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/ The portion of each entity between other entities is trimmed. Use this option together with ALL (select all entities to trim).
Crossing	Cuts entities interactively; prompts you: 1. **First corner of crossing window** - **(specify the first corner.)** 2. **Second corner of crossing window** - **(specify the second corner.)** The portions of the entities that cross the specified rectangle are trimmed.

Projection	Defines how intersections are projected; prompts you: **None/Ucs/View** - **(enter an option:)** • **None** - entities are not projected • **Ucs** - entities are projected according to the current UCS • **View** - entities are projected according to the current viewpoint This option affects the operation of the Edge option.
Erase	Deletes selected entities; prompts you: **Select entities to delete** - **(select the entities.)**
Undo	Undoes the last trim action.Repeat this option undo earlier trim actions.

Procedures

- To trim entities (1.23.21)

Related Commands

Break (6.6.18) - cuts parts out of entities. Chamfer (6.9.9) - chamfers intersections. Extend (6.13.19) - Extends open entities to a boundary. Fillet (6.14.7) - fillets intersections. Settings (6.27.23) - specifies parameters for trims.

Figure 6.2446

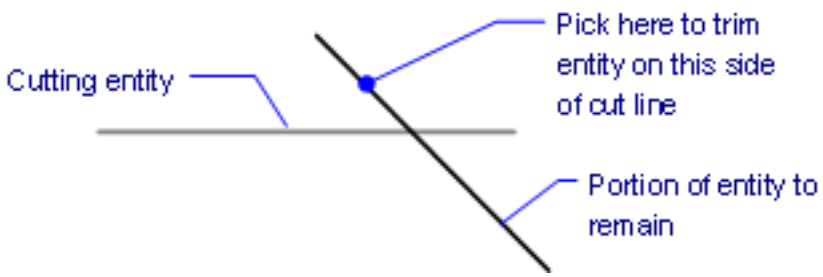


Figure 6.2447

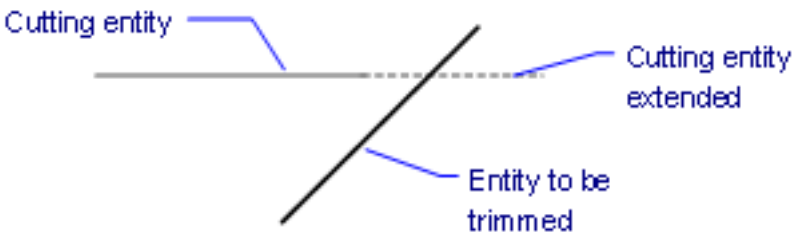


Figure 6.2448

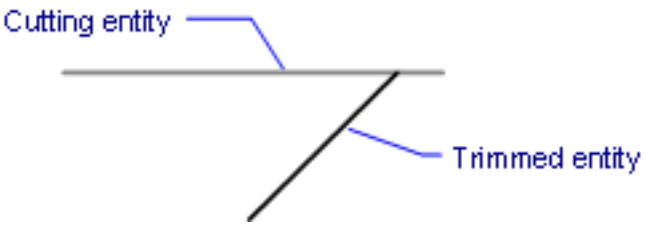


Figure 6.2449

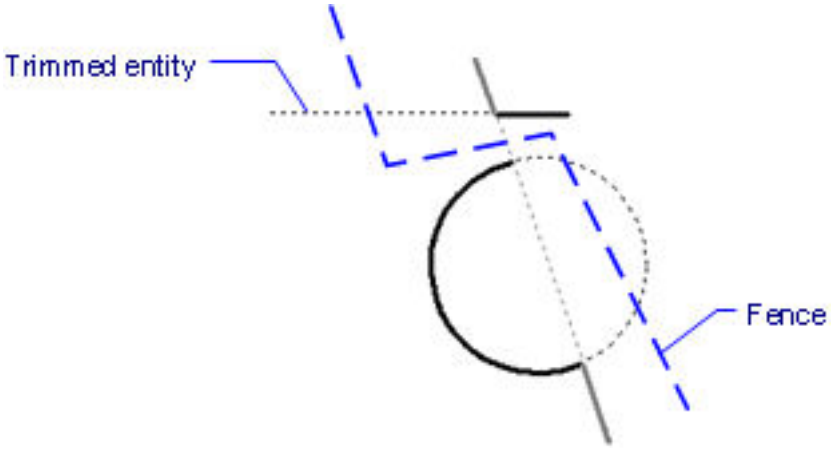
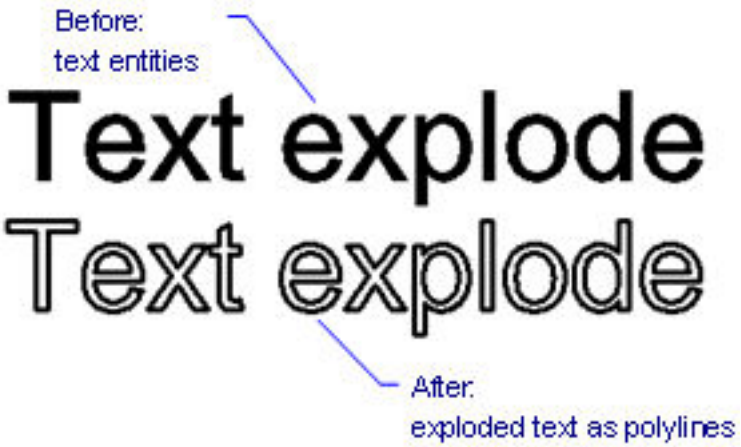


Figure 6.2450



Figure 6.2451



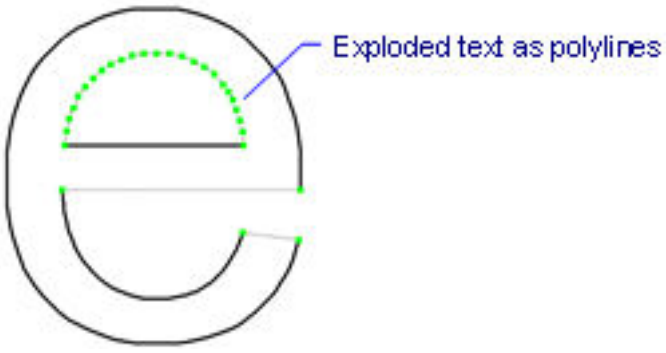
6.29.27 TxtExp

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Explodes text into polylines (short for "text explode"). (\vref {fig:C:/Dev/Task-2448/images/sta

Accessing the Command

command bar: **txtexp** menu: **Modify | Explode Text : TxtExp Prompts you in the command bar:** Select text to be exploded: **(Choose one or more text entities)** Select text to be exploded: **(Choose more text entities, or press Enter to end entity selection)** SHP and TTF fonts are exploded to individual polylines and polyarcs. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image359.jpg})

Figure 6.2452



Command Options

Option	Description
Select text to be exploded	Selects the text that you want exploded into polylines: • Choose more text entities • Enter **all** to select all text in the drawing; the program filters out non-eligible entities • Press **Enter** to end entity selection This command does not explode attributes in blocks or text in tables, or text in another space (model vs paper). Standalone attributes are, however, exploded.

Tutorial: Unlock BricsCAD's extra features Click here (1.9.1) to watch

Command Options

Option	Description
Select text to be exploded	Chooses the text to be exploded into polylines. Use any entity selection method. Press Enter to end entity selection.

Procedures

- To explode text (1.21.21)

Related Commands

Text (1.21.20) and -Text (6.29.12) - create text. MText (6.21.35) and -MText (6.21.33) - create paragraph text. Explode (6.13.13) - explodes non-text entities. U (6.30.1) - undoes the explosion.

6.30 U

6.30.1 U

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Undoes the last command that you entered.

Figure 6.2453



Accessing the Command

 command bar: **u** shortcut: **Ctrl+Z** menu bar: **Edit | Undo** ribbon: **Home | Edit | Undo** toolbar: **Standard |** 
: **U Reports to you in the command bar:** Undo: **(name of command)**

Command Options

No options.

Related Commands

Undo (6.30.5) - undoes one or more commands. Redo (6.26.8) - redoes the last undone command. Oops (6.23.7) - unerases the last erased object, including newly created blocks. Zoom (6.35.7) - utilizes the Previous option to show the previous view.

Figure 6.2454



6.30.2 Ucs

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates and displays UCSs through the command bar (short for "user-defined coordinate system").

Accessing the Command

command bar: **ucs** menu bar:**Model | Direct Modeling | Align UCS** ribbon:**Solid | Plane | Align UCS : Ucs Prompts you in the command bar:** Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/ Move/ World] <World>: **(Pick a point to specify the origin, enter an option, or else press Enter for the WCS.)**

Command Options

Option	Description
Specify origin of UCS	Specifies a new UCS through one, two, or three points in the drawing. Prompts you: 1. **Specify origin of UCS** - **(pick a point, or enter x,y,z coordinates.)** 2. **Point on X-axis/<Accept>** - **(enter an option)** • **Point on X-axis**: Pick a point to show the direction of the x axis; the program ghosts a line between the origin and the cursor • **Accept**: Press **Enter** tokeep the orientation of the x,y axes as they are; this moves the UCS to a new origin. 3. **Point on the XY-plane with positive Y value or <Accept>** - **(enter an option)** • **Point on the XY-plane with positive Y value:**Pick a point to show the direction of the y axis, which determines the angle of the x,y-plane • **Accept**: Press **Enter** to keep the x,y plane flat
Face	Aligns the UCS to a face of a 3D solid. This option works only with flat 3D faces, and not curved ones, such as on a sphere. Prompts you: 1. **Select entity** - **(click the face of a 3D solid; the x-axis of the UCS lies on the edge closest to you pick point.)** 2. **Enter an option [Flip/Xrotate/Yrotate] <accept>** - **(enter an option.)** • **Flip** - reverses the direction of the Z-axis • **Xrotate** - rotates the UCS 180° about the X-axis • **YRotate** - rotates the UCs 180° about the Y-axis • **Accept** - Press **Enter** to accept the UCS orientation picked by the program

NAmed

Creates, restores, and deletes named UCSs; prompts you: **Enter an option [Restore/Save/Delete/? for ucs list]** - **(enter an option:)**

- **Restore** - restores a named UCS; prompts you: **UCS name to restore** - **(enter the name of a UCS.)** As a shortcut, you can enter R directly at the 'Specify origin of UCS' prompt.
- **Save** - saves the current UCS by a name; prompts you: **Enter name for saving UCS, or ? for list** - **(enter an option:)**
  - Enter a name by which to save the UCS. As a shortcut, you can enter S directly at the 'Specify origin of UCS' prompt.
  - Press **?** to list UCS names already saved in this drawing
- **Delete** - erases a named UCS from the drawing; prompts you: **UCS name(s) to delete, or ? for list** -
  - Enter one or more names of UCSs; separate the names with commas. As a shortcut, you can enter D directly at the 'Specify origin of UCS' prompt.
  - Press **?** to list UCS names already saved in this drawing
- **? for ucs list** - lists the names of UCSs in the drawing; prompts you: **Name(s) of UCS to list <* for all>** - **(specify the names to list:)**
  - ***** - lists all UCS names
  - **name*** - lists names of UCSs that begin with **name**
  - **name** - lists the UCS specified by name If necessary, press **F (6.29.13)**2 (6.29.13) to view the history prompt window. As a shortcut, you can enter ? directly at the 'Specify origin of UCS' prompt.

Entity

Aligns the UCS to a selected entity; prompts you: **Select entity for UCS definition** - **(choose a single entity)**; the orientation and type of entity define the orientation of the UCS. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

- The UCS origin is located to the end point, center point, or vertex closest to the pick point.
- The x-axis is aligned with the entity or an edge
- The x,y-plane is aligned to the entity's plane
- For ambiguous objects with obvious orientation, like a circle, the orientation is maintained

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Previous

Changes the UCS to the previous one.

View	Sets the UCS to the current viewpoint: • The x axis and y axis are parallel to the view edges • The z axis is perpendicular to the view, with positive Z-axis pointing to the viewer. • The origin is copied from the previous coordinate system.
X	Rotates the current UCS about the x axis; prompts you: **Enter angle of rotation about X axis** - **(specify an angle or pick two points.)**
Y	Rotates the current UCS about the y axis; prompts you: **Enter angle of rotation about Y axis** - **(specify an angle or pick two points.)**
Z	Rotates the current UCS about the z axis; prompts you: **Enter angle of rotation about Z axis -(specify an angle or pick two points.)**
ZAxis	Sets the UCS relative to the z-axis; prompts you to define the Z-axis: 1. **Origin point** - **(pick a point, or enter x,y,z coordinates.)** 2. **Point on positive Z-axis** - **(pick another point, or enter x,y,z coordinates.)** • The second point defines the positive Z-axis. • The X • Y-plane is perpendicular to the Z-axis with the X-axis horizontal and the Y-axis pointing upwards.
Move	Moves the origin of the current UCS, keeping the orientation of the axes; prompts you: **Origin point <0,0,0>** - **(pick point, or enter x,y,z coordinates.)**
World	Switches to the WCS (world coordinate system).

Related Commands

ExpUcs (6.13.18) - creates, modifies, and deletes named UCSs through the Drawing Explorer. UcsIcon (6.30.3) - controls the appearance of the UCS icon. SetUcs (6.27.25) - sets the UCS to a specified viewpoint. Plan (6.24.25) - changes the viewpoint to the plan view of the WCS or current UCS. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Settings (6.27.23) - sets parameters of UCSs.

Figure 6.2455

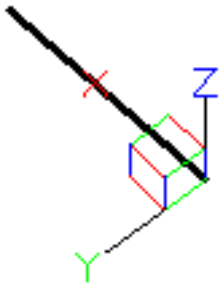


Figure 6.2456

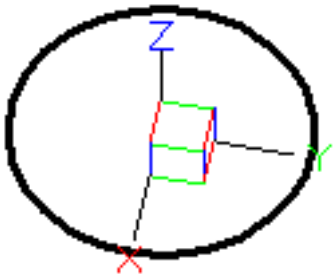


Figure 6.2457



6.30.3 UcsIcon

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Controls the display of the UCS icon (short for "user-defined coordinate system icon"). (\vref {fig:C:/Dev/Task-2448/images/staticb
Note: The colors denote the directions of the axes:

- **Red** - x axis
- **Green** - y axis
- **Blue** - z axis

Accessing the Command

command bar: **ucsicon : UcsIcon Prompts you in the command bar:** [ucs icon ON/ucs icon OFF/display in All views/display at ORigin/display in Corner] <ON>: **(Enter an option)** The properties of the UCS icon can be changed by the User Coordinate System section of the Settings (6.27.23) dialog box.

Command Options

Option	Description
ucs icon ON	Turns on the display of the UCS icon: • UCS icon in wireframe and hidden visual styles: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/U • UCS icon in all other visual styles: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/U • UCS icon in paperspace layouts: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/U The colors of the axes are controlled by the COLORX, COLORY and COLORZ system variables: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
ucs icon OFf	Turns off the UCS icon. This option is viewport-specific, meaning it turns off the icon in the current viewport only.
display in All views	Applies the changes to all viewports; repeats prompts: **[ucs icon ON/ucs icon OFf/display at ORigin/display in Corner] <ON>** - **(enter an option to apply to all viewports.)** This command normally applies only to the current viewport; use this option to apply changes to all viewports.
display at ORigin	Displays the UCS icon at the origin (0,0), when possible: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i When the origin is outside of the viewport, the UCS icon is displayed at the corner defined by the UCSICONPOS (1.13.11) system variable; see the Corner option, below. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

display in Corner	Displays the UCS icon in the viewport corner defined by the UCSICONPOS (1.13.11) system variable: • 0 - lower right corner (default) • 1 - lower left corner • 2 - upper right corner • 3 - upper left corner

Related Commands

Ucs (6.30.2) - creates and modifies user-defined coordinate systems. ExpUcs (6.13.18) - creates, modifies, and deletes named UCSs through the Drawing Explorer. SetUcs (6.27.25) - sets the UCS to a specified viewpoint. Plan (6.24.25) - changes the viewpoint to the plan view of the WCS or current UCS. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. Settings (6.27.23) - sets parameters of UCSs.

Figure 6.2458

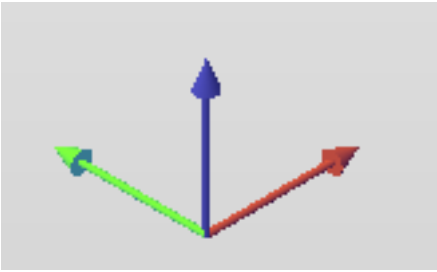


Figure 6.2459

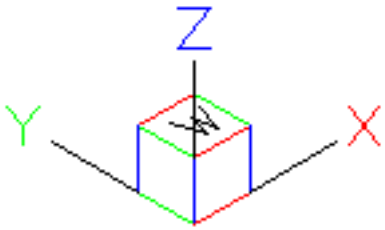


Figure 6.2460

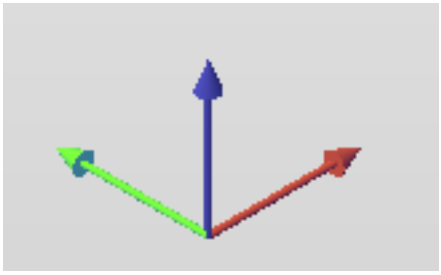


Figure 6.2461

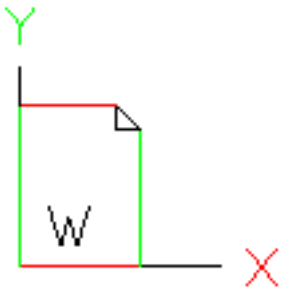


Figure 6.2462

X axis color	Red
Y axis color	Green
Z axis color	Blue

Figure 6.2463



Figure 6.2464

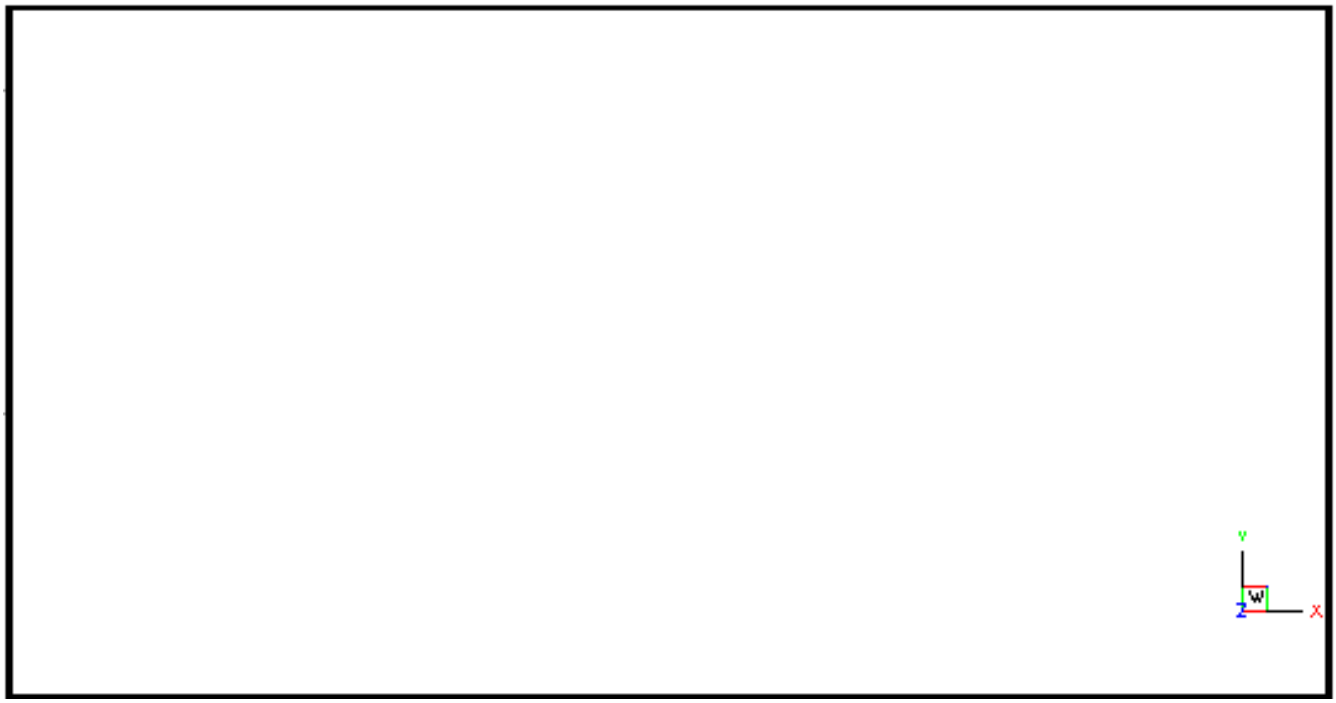


Figure 6.2465



6.30.4 Undefine

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Removes access to commands temporarily. **Note:** Use theRedefine (6.26.7) command to bring back undefined commands.

Accessing the Command

command bar: **undefine : Undefine Prompts you in the command bar:** Command to undefine: **(Enter the name of a command)** When you enter the name of an undefined command, BricsCAD responds: Unable to recognize command. Please try again. Undefined commands are accessed in these days:

- Prefix the command name with a dot, such as .CommandName
- Use the Redefine (6.26.7) command to recover the command name.

Command Options

Option	Description
Command to undefine	Enter the name of a command. Commands are undefined by developers who want to replace the command with a more extensive version they've written, or to prevent neophytes from using destructive commands, like Erase and Explode.

Related Commands

Commands (6.9.37) - lists the names of commands. Customize (6.9.58) - creates new commands through macros and LISP routines. Redefine (6.26.7) - activates commands that were deactivated with the Undefine command

Figure 6.2466



6.30.5 Undo

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Undoes one or more commands; some commands cannot be undone.

Accessing the Command

command bar: **undo : Undo Prompts you in the command bar:** Undo: Number of steps to undo or [Mark/Back to mark/BEgin set/End set/Control/Auto]: **(Enter the number of commands to undo, or enter an option)** BricsCAD reports the commands being undone. For example, enter the Customize and then the Units commands, followed by entering Undo 2: Undo: (UNITS) Undo: (CUSTOMIZE)

Command Options

Option	Description
Number of steps to undo	Specifies the number of commands to undo; snter a number. Commands are undone sequentially; you cannot skip over commands.Some commands cannot be undone, such as Save, Open, and Plot.
Mark	Sets a marker.The **Back to mark** option undoes all commands to the marker.
Back to mark	Undoes all commands back to when the marker was set.
BEgin set	Sets the start to a group that groups the following commands. The Undo command treats the commands in the set as a single undo.
End set	Ends the group.
Control	Specifies options; prompts you: **Set Undo control: None/One/<All>** - **(enter an option.)** • **None** - turns off the undo mechanism. This is useful when disk space is running low, because the undo mechanism uses disk space. • **One**- limits this command to a single undo. This option turns the Undo command into the U command. • **All**- turns on the undo mechanism
Auto	Considers all commands executed by a macro as a single undo.

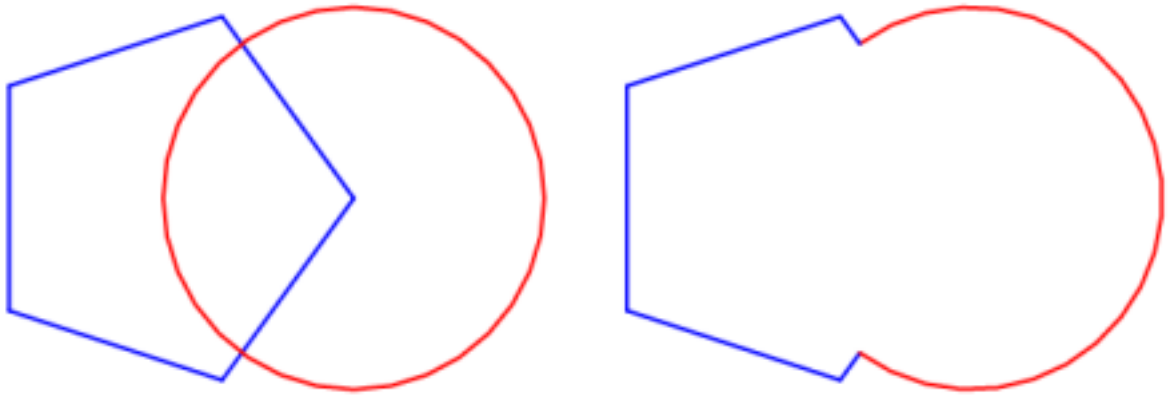
> Related Commands

U (6.30.1) - undoes the last command. Redo (6.26.8) - redoes the last undone command. Oops (6.23.7) - unerases the last erased object, including newly created blocks. Zoom (6.35.7) - utilizes the Previous option to show the previous view.

Figure 6.2467



Figure 6.2468



6.30.6 Union

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Performs Boolean union operations on 3D solids and 2D regions by adding set of entities to another set to form a single entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/

Accessing the Command

 command bar: **union** alias: **uni** menu bar: **Model | 3D Solid Editing | Union** ribbon: **Solid | Edit | Boolean | Union** toolbar: **3D Solid Editing |**  **: Union Prompts you in the command bar:** Select ACIS objects to union: **(Choose one or more 3D solids and/or 2D regions)** Entities in set: 2 **Select ACIS objects to union: (Choose more ACIS entities, or press Enter to complete the command)** 3D solids that are joined through the Union command often look no different after the operation: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/UnionSolids.p This command also applies to regions: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sourc

Figure 6.2469

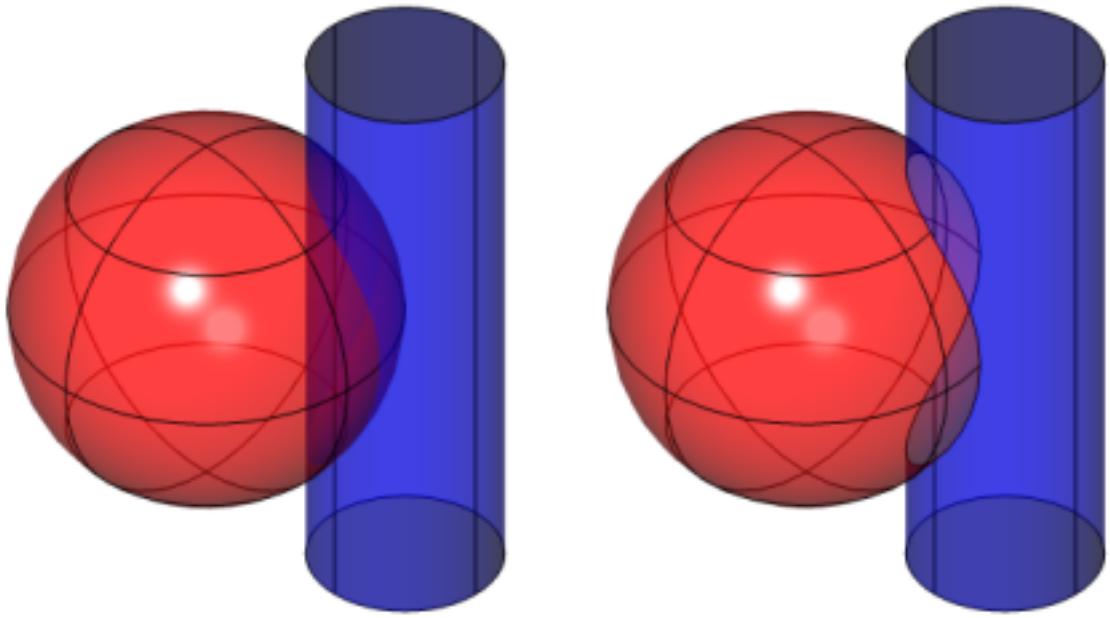
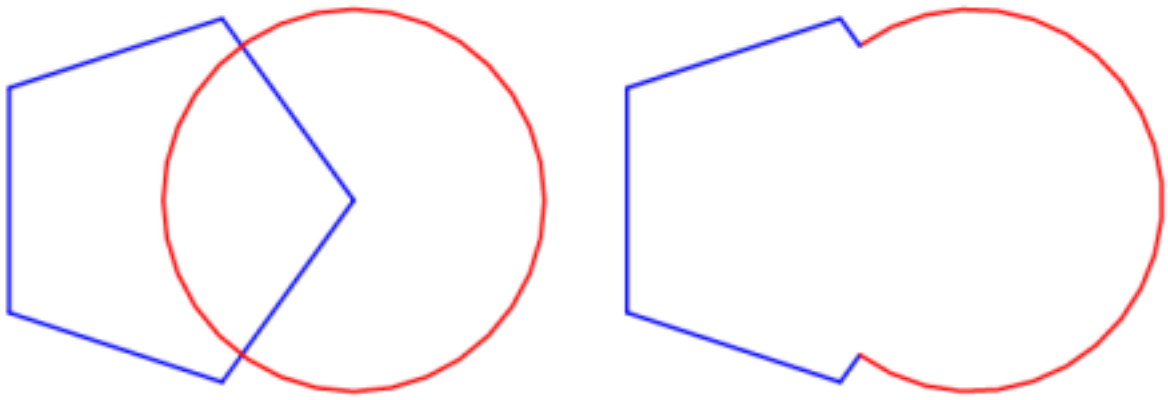


Figure 6.2470



Command Options

Option	Description
Select ACIS objects to union	Specifies the ACIS entities to join into a single entity.Choose one or more 3D solids and/or 2D regions. The resulting entity takes on the properties of the ACIS entity selected first.This command also joins regions and 3D solids into a single entity.

Related Commands

BimSplit (6.7.32) - splits segments of wall-like solids into separated solids. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. Revolve (6.26.30) - revolves 2D entities into 3D solid models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Subtract (6.27.56) - removes one solid model from a second.

Figure 6.2471



6.30.7 UnisolateObjects

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Unhides entities that were hidden by theHideObjects (6.16.11)andIsolateObjects (6.17.21) commands.

Accessing the Command

 command bar: **unisolateobjects** aliases:**unisolate**,**unhideobjects**,**unhide** ribbon: **Model | View | Show All** toolbar: **Hide/Show Entities** Quad: **Select : UnisolateObjects** The program displays entities that were previously hidden by theHideObjectsandIsolateObjects commands.

Command Options

No options.

Related Commands

HideObjects (6.16.11) - hides the selected entities. IsolateObjects (6.17.21) - hides all but the selected entities. LayOn (6.20.13) - turns on all layers that were turned off in the drawing, except for frozen layer .

Figure 6.2472



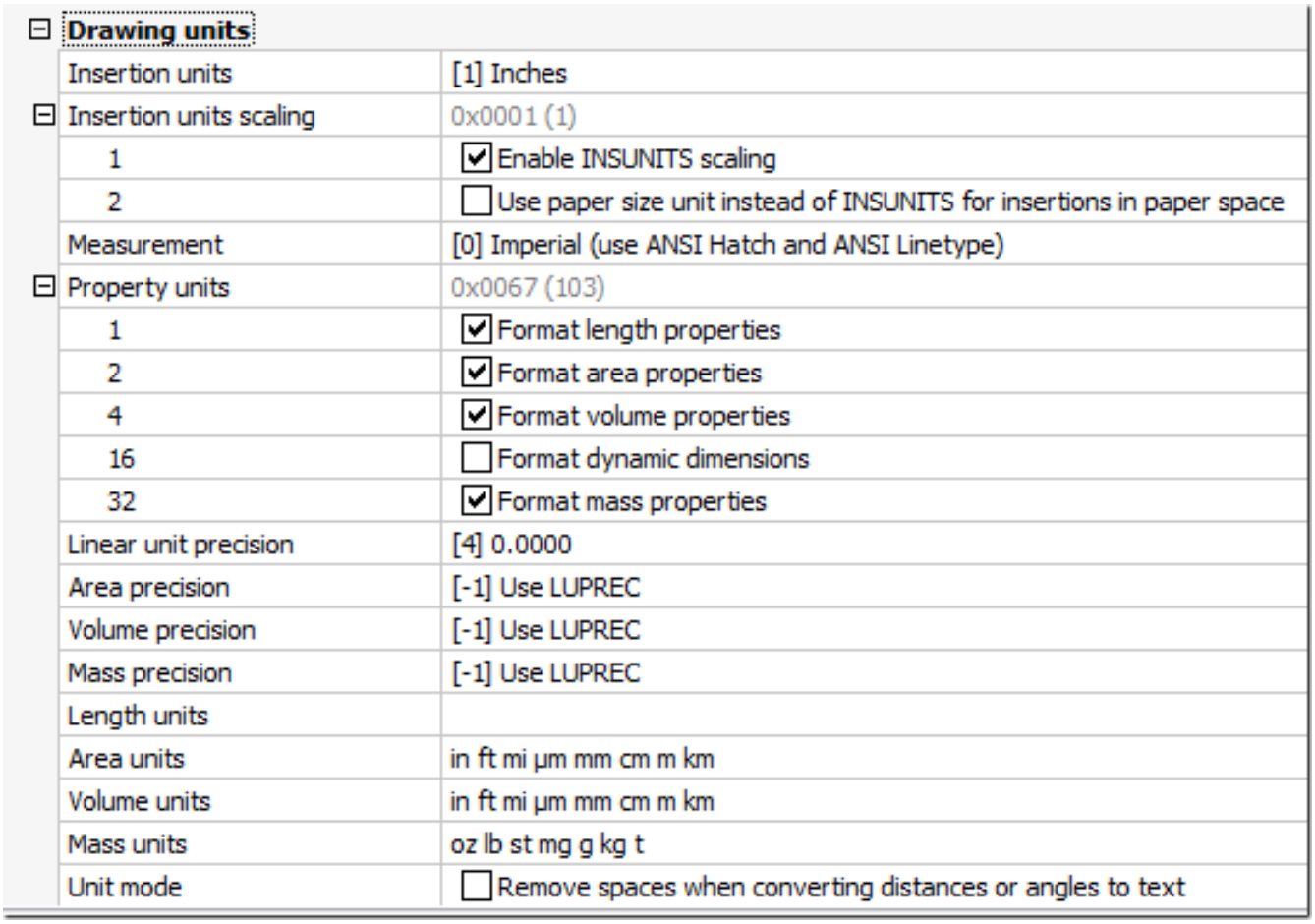
6.30.8 Units

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets units of linear and angular measurements through the Settings dialog box.

Accessing the Command

command bar: **units** transparent: **'units** aliases: **un ddunits** status bar: **X,Y,Z : Units** Displays the Drawing Units section of the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/U/unitdlg.png}) Change settings, and then click the **X** button.

Figure 6.2473



Command Options

Option	Description
Insertion Units	Defines a real world value of 1 drawing unit, which is used for automatic scaling when inserting or attaching blocks, images, and xrefs.When both INSUNITS and PROPUNITS are on, the Length, Area, Volume and/or Inertia variables will be formatted with a dedicated unit. 1. Click the **Insertion Units** droplist 2. Select a unit: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/I

Measurement	Determines which source files will be used when loading linetypes and predefined hatch patterns. 1. Click the **Measurement** droplist 2. Select an option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M • Imperial - uses **default.lin**for linetypes and **default.pat** for hatch patterns • Metric - uses **iso.lin** and **iso.pat** BricsCAD searches for the default hatch and linetypes files in the folders which are defined by the SRCHPATH variable, which by default is **C:\Users\<username>\AppData\Roaming\Bricsys\Bricscad\V..\e**
Property Units	Defines whether measurement values are formatted automaticallywhen INSUNITS is not 0 -- length, area, volume, and mass.The value is stored as a bitcode of the sum of selected options: • **1** - Formats length properties • **2**- Formats area properties • **4**- Formats volume properties • **16**- Formats dynamic dimensions • **32**- Formats mass properties
Linear unit precision	Specifies the number of decimal places displayed by linear measurements: 1. Click the **Linear Unit Precision**droplist 2. Select an option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P Although the droplist shows decimal precision, the option you choose also affects the precision of fractions used by architectural and fractional units, as shown below: • **0** - 0 fractional precision, such as 1 • **1** - 1/2 • **2** - 1/4 • **3** - 1/8 • **4** - 1/16 • **5** - 1/32 • **6** - 1/64 • **7** - 1/128 • **8** - 1/256 The value of Linear unit precision (LUCPREC) applies also to AREAPREC, VOLUMEPREC and MASSPREC unless these system variables are specified explicitly; see below.

Area,Volume, **and**Mass Precision	Specifies the number of decimal places for the display accuracy of areas, volumes, and masses. The display precision comes into play whenthe corresponding bit of the PROPUNITS (Property Units) variable is on. When its value is negative, the program instead uses the value of LUPREC (Linear Unit Precision). 1. Click a**Precision**droplist 2. Select an option:
Length Units	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/A Defines a list of units used for displaying lengths: • When this field is empty, all quantities display in the drawing units defined by the INSUNITS variable. • When one or more units are selected in the LengthUnits dialog box, the program uses the unit that is a best fit. The Format Unit Properties bit of the Property Units variable must be turned on. 1. Click the **Length Units** field. 2. Select one or more units in the dialog box. 3. Click **OK**.
Area Units	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L Defines the units used for displaying areas: • When this field is empty, all quantities display in the drawing units defined by the INSUNITS variable. • When one or more units are selected in the AreaUnits dialog box, the program uses the unit that is a best fit. The Format Area Properties bit of the Property Units variable must be turned on. 1. Click the **Area Units** field. 2. Select one or more units in the dialog box. 3. Click **OK**.
	(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/A

Volume Units	Defines a list of units used for displaying volumes: • When this field is empty, all quantities display in the drawing units defined by the INSUNITS variable. • When one or more units are selected in the VolumeUnits dialog box, the program uses the unit that is a best fit. The Format Volume Properties bit of the Property Units variable must be turned on. 1. Click the **Volume Units** field. 2. Select one or more units in the dialog box. 3. Click **OK**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/V
Mass Units	Defines the units used for displaying mass properties (weights): • When this field is empty, all quantities display in the drawing units defined by the INSUNITS variable. • When one or more units are selected in the MassUnits dialog box, the program uses the unit that is a best fit. The Format Mass /Properties bit of the Property Units variable must be turned on. 1. Click the **AMass rea Units** field. 2. Select one or more units in the dialog box. 3. Click **OK**. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/M
Unit mode	Specifies how imperial units are displayed by the program: • **0** - spaces: displays fractional, feet-and-inches, and surveyor's angles using spaces as delimiters • **1**- dashes: displays fractional, feet-and-inches, and surveyor's angles without spaces and in some cases substituting dashes for spaces
Suppress dim zeros	Suppress leading and trailing zeros in dimensions: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/S
Linear unit type	Specifies the units for linear measurement: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/L • **Scientific** - such as 4.225E+01 • **Decimal** - such as 42.25 • **Engineering** - such as 3'-6.25" • **Architectural** - such as 3'-6 1/4" • **Fractional** - such as 42 1/4 For metric units, choose 2 (decimal)

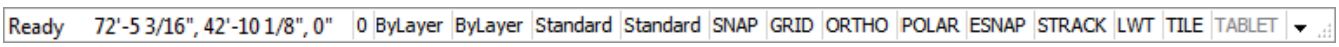
Angular unit type	Specifies the style of angular units: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/A • **Decimal degrees** - such as 90.0 • **Degrees/minutes/seconds** - such as 90d0’0 • **Gradians** - such as 100.00g. There are 400 grads in a circle • **Radians** - such as 1.57r. There are 2*pi radians (approximately 6.282) in a circle • **Surveyor’s units** - such as N 00d0’0”E. The N and E in Surveyor’s units refer to North and east
Angular unit precision	Specifies the precision for angular units: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/P
Angle base	Specifies the direction for 0 degrees: the default is the positive x axis. Enter an angle. You can rotate the drawing through the Snap (6.27.38) command’s Rotate option. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Angle direction	Toggles the direction in which angles are measured: • **No** - measures angles counterclockwise (default) • **Yes** - measures angles clockwise

Status Bar Menu

Right-click 72’,42’,0” (or other x,y,z coordinates) on the status bar for the shortcut menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image792.png}) (\vref {fig:C:/Dev/Task-2448/images/static contexetmenu.png})

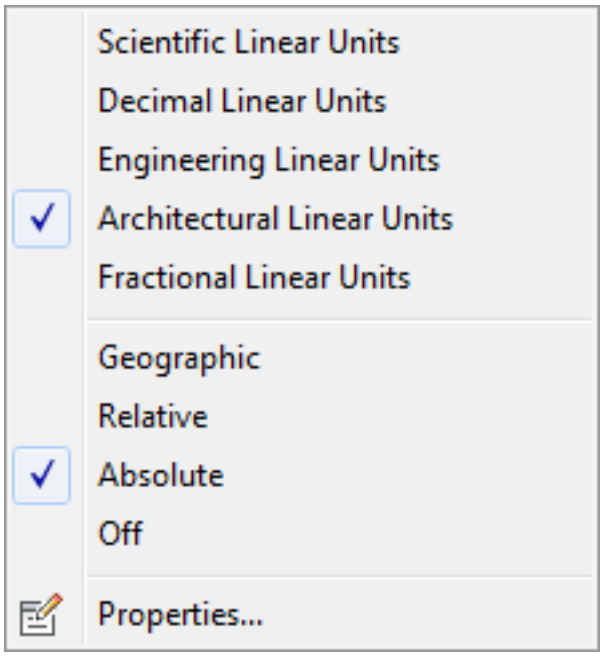
Shortcut Menu Option	**Description**
Scientific Linear Units	Chooses scientific as the working units
Decimal Linear Units	Chooses decimal (metric) as the working units
Engineering Linear Units	Chooses engineering as the working units
Architectural Linear Units	Chooses architectural as the working units
Fractional Linear Units	Chooses fractional as the working units.
Geographic	Displays geographic coordinates in the status bar
Relative	Displays relative coordinates in the status bar
Absolute	Displays absolute coordinates in the status bar
Off	Displays coordinates only when points are picked

Figure 6.2474



Properties	Opens the Coordinates section of the Settings dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Update coordinates only when selecting points** - equivalent to Off • **Coordinates always show pointer location** - equivalent to Absolute • **Coordinates in polar form for point, distance and angle selection** - equivalent to Relative

Figure 6.2475



Related Commands

-Units (6.30.9) - sets units through the command bar. NewWiz (6.22.6) - sets units and other parameters for new drawings. New (6.22.4) - starts new drawing in Imperial or metric units.

Figure 6.2476

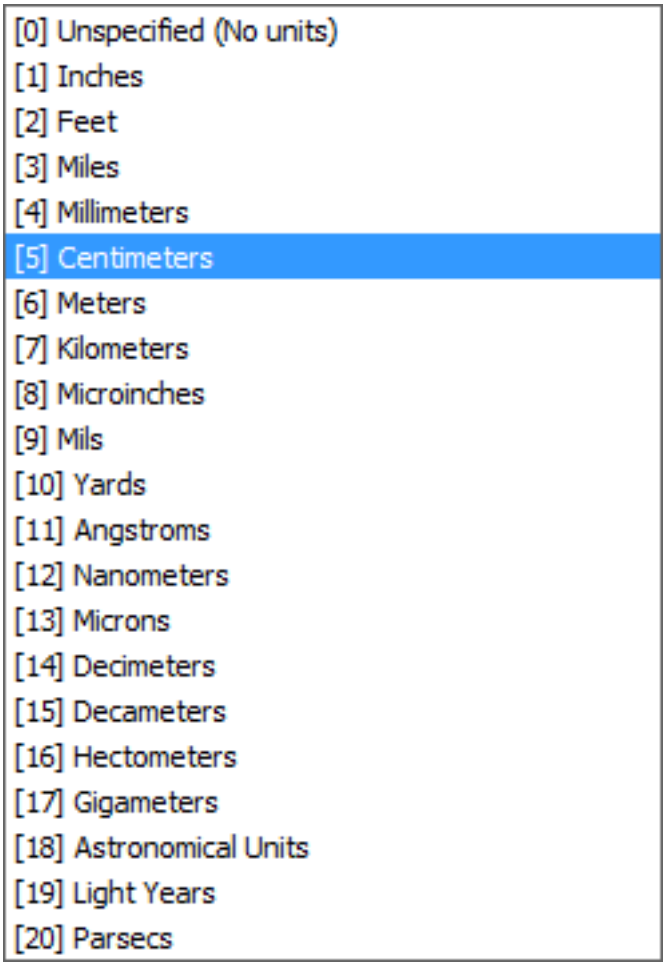


Figure 6.2477

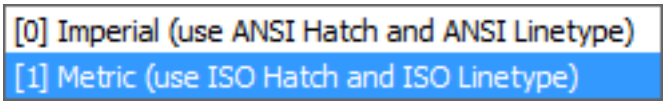


Figure 6.2478



Figure 6.2479



Figure 6.2480

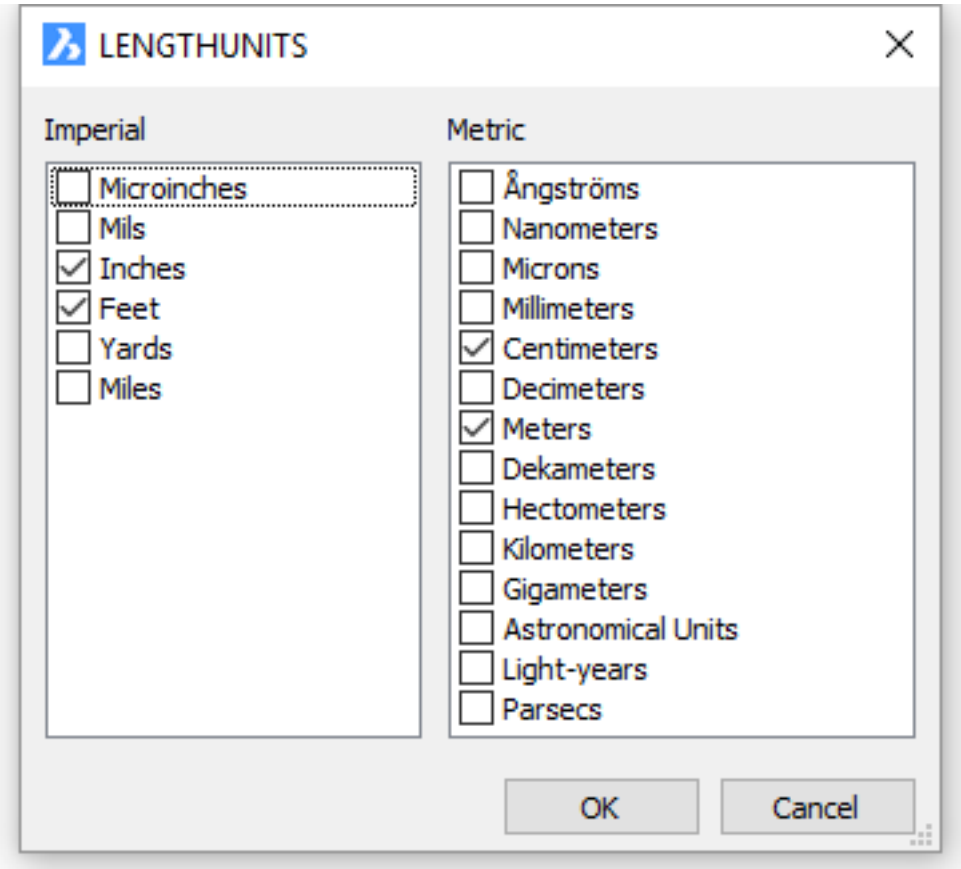


Figure 6.2481

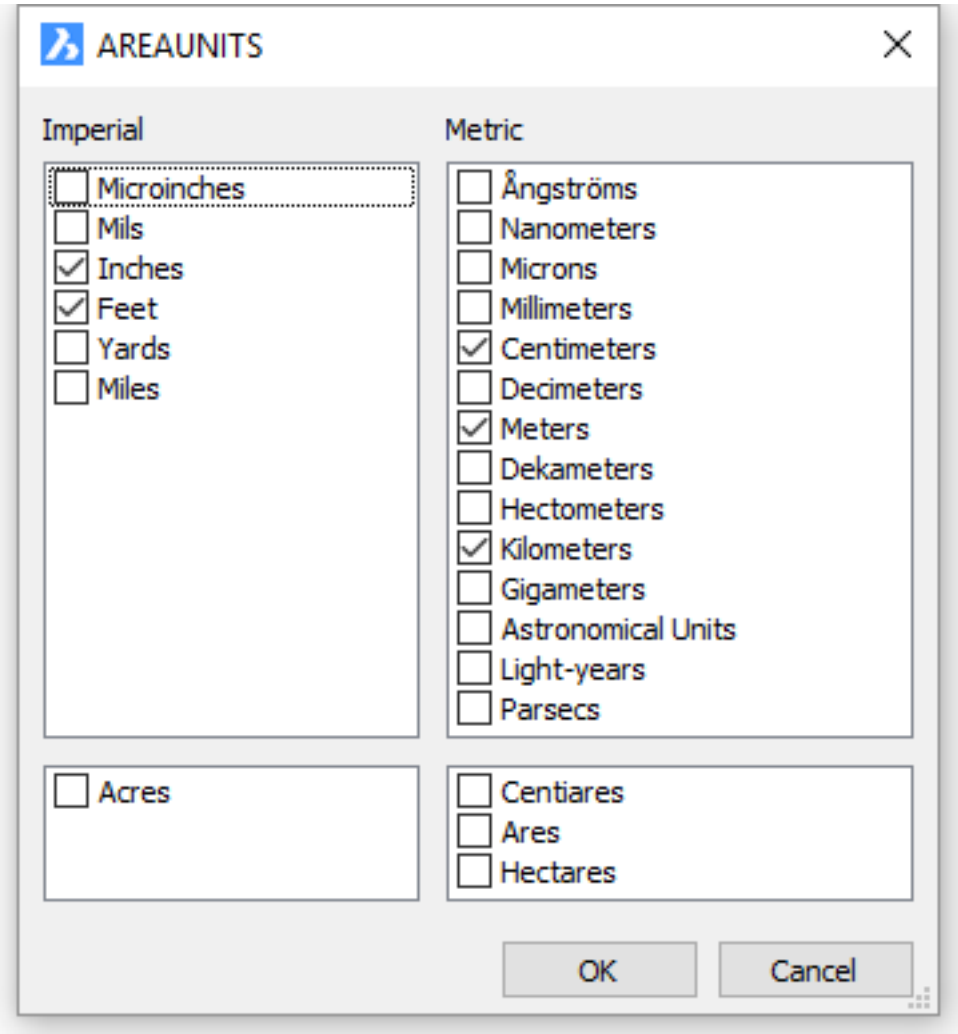


Figure 6.2482

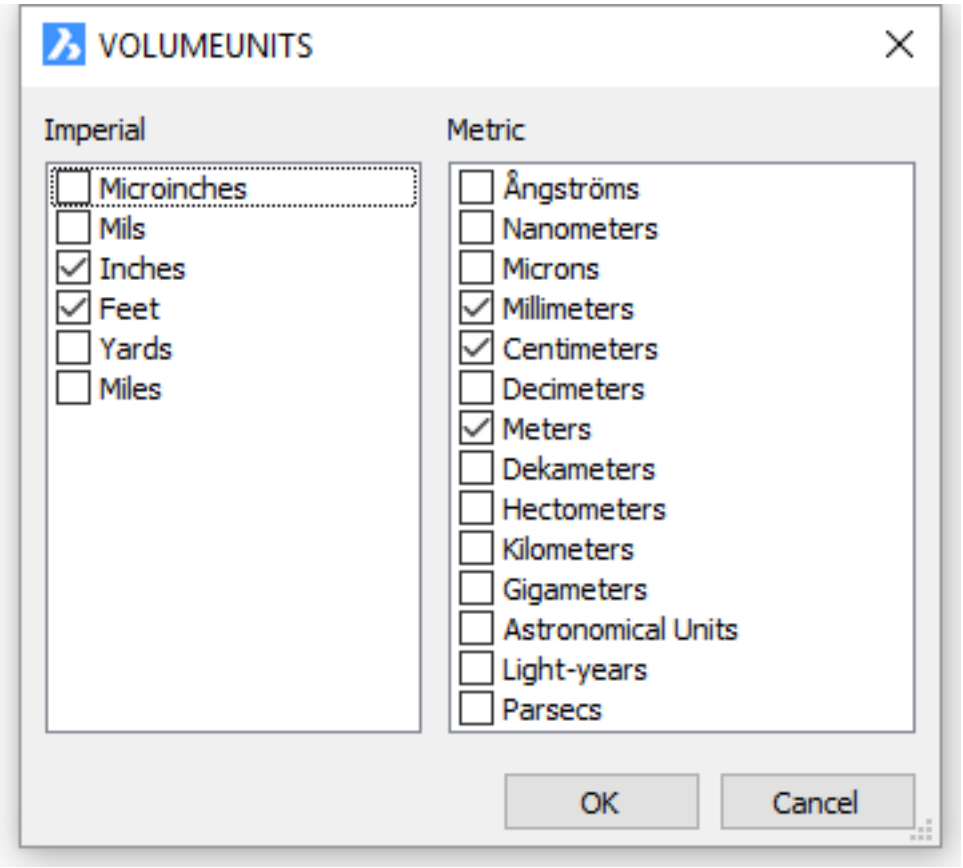


Figure 6.2483

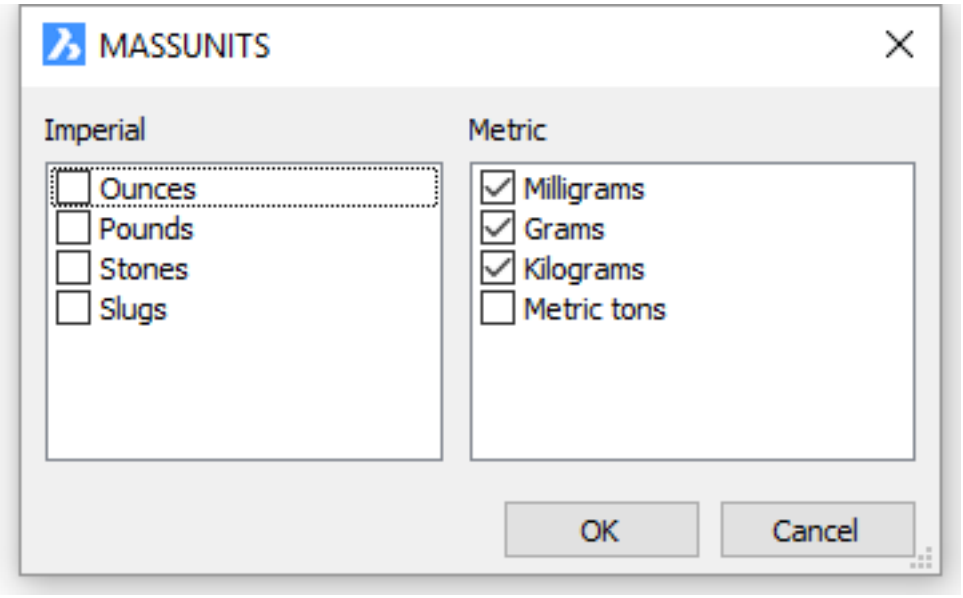


Figure 6.2484

Suppress dim zeros	0x0000 (0)
mask: 0x0003	[0] Suppress zero feet and precisely zero inches
4	Suppress leading zeros in decimal dimensions
8	Suppress trailing zeros in decimal dimensions

Figure 6.2485

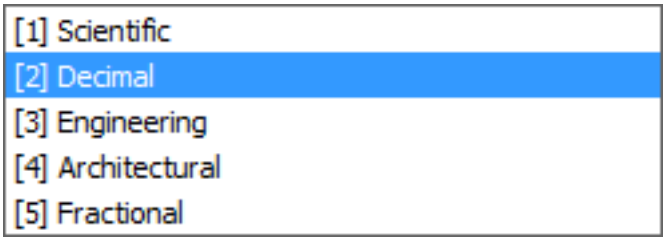


Figure 6.2486

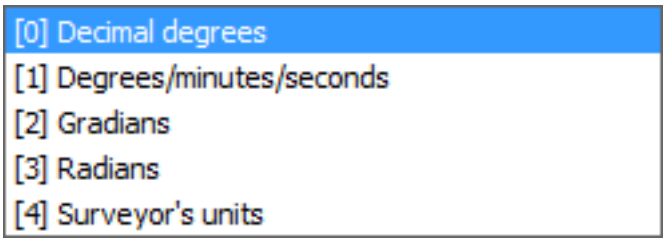


Figure 6.2487



Figure 6.2488

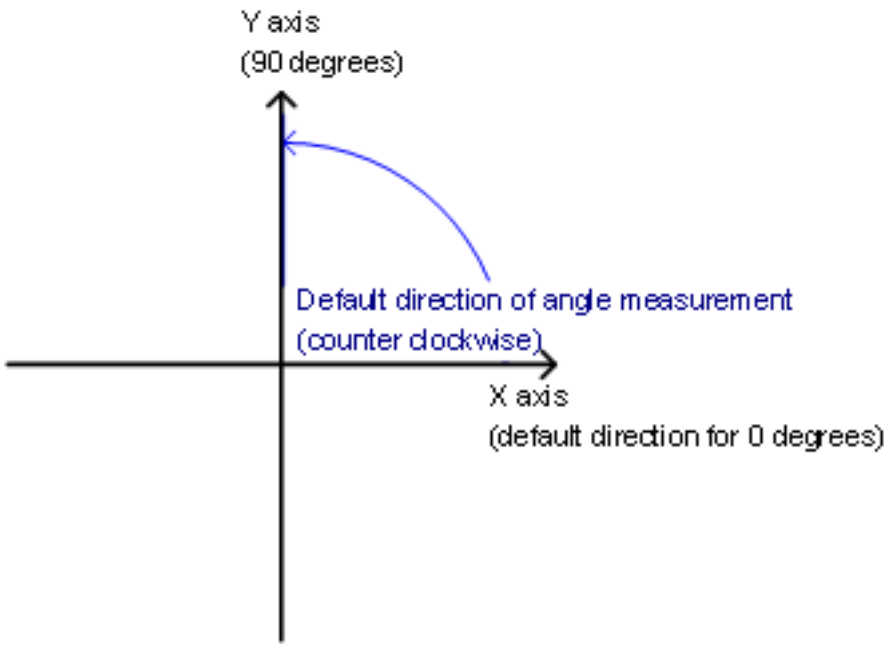


Figure 6.2489

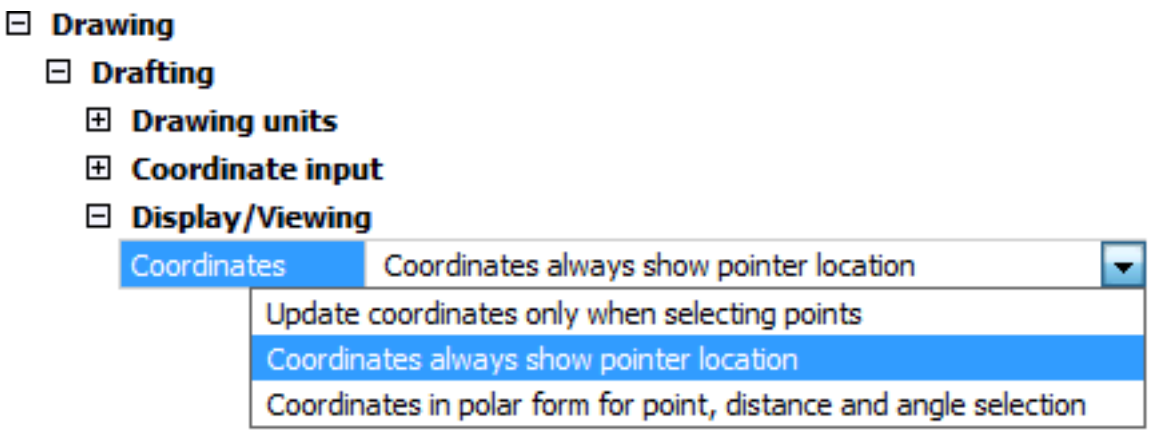


Figure 6.2490



6.30.9 -Units

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets units of linear and angular measurements in the command bar.

Accessing the Command

command bar: **-units** transparent: **'-units** alias: **-un** : **-Units Prompts you in the command bar:** Unit mode (LUNITS): 1. Scientific 4.225E+01 2. Decimal 42.25 3. Engineering 3'-6.25" 4. Architectural 3'-6 1/4" 5. Fractional 42 1/4 Unit mode 1-5 <2>: **(Enter a number, or press Enter for the default)** Number of decimal places for LUNITS 0-8 <4>: **(Enter a number, or press Enter for the default)** Angular unit mode (AUNITS): 1. Decimal degrees 90.0 2. Degrees/minutes/seconds 90d0'0 3. Grads 100.00g 4. Radians 1.57r 5. Surveyor's units N 00d0'0"E Angular unit mode 1-5 <1>: **(Enter a number, or press Enter for the default)** Number of decimal places for angular units 0-8 <0>: **(Enter a number, or press Enter for the default)** Angle 0 direction <0>: **(Enter a number, or press Enter for the default)** Do you want angles measured clockwise? <N>: **(Type Y or N)**

Command Options

Option	Description
Unit mode	Specifies the display style of linear measurement.Enter a number between 1 and 5 corresponding to the list: • **1** - Scientific, such as 4.225E+01 • **2** - Decimal, such as 42.25 • **3** - Engineering, such as 3'-6.25" • **4** - Architectural, such as 3'-6 1/4" • **5** - Fractional, such as 42 1/4 For metric units, choose 2 (decimal). LUNITS variable stores the style of linear units.

Number of decimal places for LUNITS	Specifies the display precision for decimal units.Enter a number between 0 and 8: • **0** - 0 decimal places, such as 0. • **1** - 1 decimal places, such as 0.0 • **2** - 2 decimal places, such as 0.00 • **3** - 3 decimal places, such as 0.000 • **4** - 4 decimal places, such as 0.0000 • **5** - 5 decimal places, such as 0.00000 • **6** - 6 decimal places, such as 0.000000 • **7** - 7 decimal places, such as 0.0000000 • **8** - 8 decimal places, such as 0.00000000 This option also affects the display precision of fractions used by architectural and fractional units: • **0** - 0 fractional precision, such as 1 • **1** - 1/2 • **2** - 1/4 • **3** - 1/8 • **4** - 1/16 • **5** - 1/32 • **6** - 1/64 • **7** - 1/128 • **8** - 1/256
Angular unit mode	Specifies the display style of angular units.Enter a number: • **1** - Decimal degrees, such as 90.0 • **2** - Degrees, minutes, and decimal seconds, such as 90d0’0 • **3** - Grads, such as 100.00g • **4** - Radians, such as 1.57r • **5** - Surveyor’s units, such as N 00d0’0”E There are 400 grads in a circle.There are 2*pi radians (approximately 6.282) in a circle.The N and E in Surveyor’s units refers to North and east. AUNITS variables stores the style of angular units.
Number of decimal places for angular units	Specifies the number of decimal places.Enter a number between 0 and 8.
Angle 0 direction	Specifies the direction for 0 degrees.Enter an angle, or pick two points in the drawing.The default is the positive x axis.You can also rotate the drawing through the Snap command’s Rotate option. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Do you want angles measured clockwise?	Specifies the direction in which to measure angles: • **Yes** - measures angles clockwise • **No** - measures angles counterclockwise (default)

Related Commands

Units (6.30.8) - sets units through the Settings dialog box. NewWiz (6.22.6) - sets units and other parameters for new drawings. New (6.22.4) - starts new drawing in Imperial or metric units.

Figure 6.2491

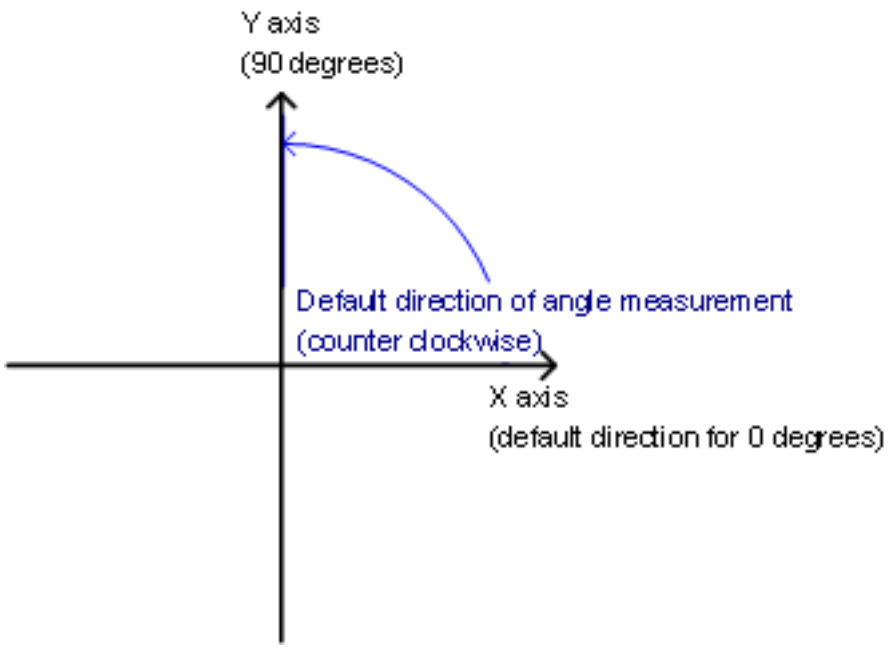


Figure 6.2492



Figure 6.2493



6.30.10 UpdateField

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Updates fields. **Note:**A field is text that update semi- and automatically, and is placed with Field (6.14.3)and other text-related commands. Fields are identified by their gray background color: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/U/field-eg.png})

Accessing the Command

 command bar: **updatefield** menu bar: **Tools | Update Fields** ribbon: **Draw | Text | Fields | Update Fields** toolbar: **Redraw/Regen |** **: UpdateField Prompts you in the command bar:** Select field(s) to update:**(Select one or more field entities)** Entities in set: 1 Select field(s) to update:**(Select more field entities, or else press Enter to end the command)** 1 field found, 1 updated. Command Options

Option	Description
Select field(s) to update	Select the field entities to update: • Select one or more field entities • Enter **all** to select all entities in the drawing; the program filters out non-eligible ones • Press **Enter** to finish the command

Procedures

- To update a field (1.21.23)

Related Commands

Mtext (6.21.35) - Places paragraph text. Field (6.14.3) - Inserts a field in a multiline text.

Figure 6.2494



6.30.11 Url

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Prompts for a location on the internet (short for "uniform resource locator").

Accessing the Command

command bar: **url** alias: **browser : Url Prompts you in the command bar:** Enter URL <http://bricsys.com>: **(Enter address of a Web page, or else press Enter to accept the default)**

Command Options

Option	Description
Enter URL	Specifies a location on the Internet: • Enter a URL, such as www.bricsys.com. • Press **Enter** to accept the default destination After you press **Enter**, your computer's default Web browser opens the page.URL is short for "universal resource locator."

Related Commands

Browser (6.6.19) - opens the default Web browser. OnWeb (6.23.6) - opens the Bricsys home page in your computer's default Web browser. Mail (6.21.1) - attaches the current drawing to a message in the default email client.

6.31 V

6.31.1 View

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and sets named 2D and 3D views through the Drawing Explorer.

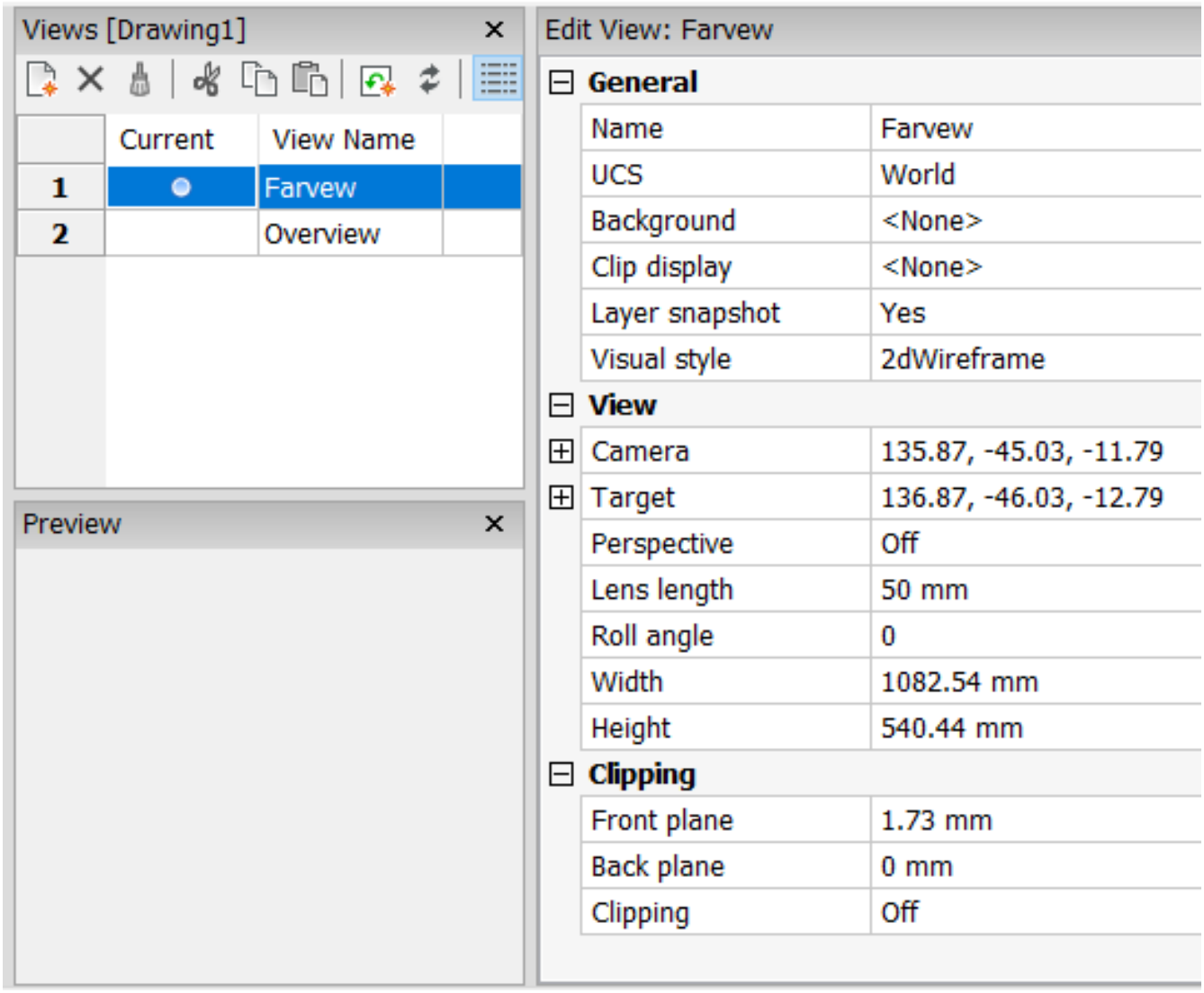
Figure 6.2495



Accessing the Command

 command bar: **view** aliases: **ddview v expviews** menu bar: **Tools | Drawing Explorer | Views** toolbar: View | ribbon:**Home | Settings | Drawing Explorer | Views : View** Displays the Views section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/view-dlg.png}) Create or select named views, and then click **X**. Press **F1** to access help.

Figure 6.2496



Toolbar Options

Toolbar Option	Description
Current	Designates the current view.
New	Creates new named views; prompts you: **View: ? to list saved views/Save/Window** - **(enter an option:)** • **? to list saved views** - lists the names of views saved in this drawing; prompts you: **View(s) to list <List all views>** - **(enter a name or press Enter for all)**; the program respond with a list: `Name of view: NewView1 Space: Model Height of view: 10.` Press F2 (6.29.13) to see the Prompt History window. • **Save** - saves the current viewpoint as the named view, and then returns to the Drawing Explorer • **Window** - saves a windowed area as the view; prompts you: 1. **First corner of view window** - **(pick a point, or enter x,y coordinates.)** 2. **Opposite corner** - **(pick another point, or enter x,y coordinates, and then the program returns to the Drawing Explorer.)**
Delete	Erases the named view from the drawing without warning.
Regen	Regenerates the current viewport.
Regen at each change	Toggles the regeneration of the current viewport after each change ON/OFF.

Edit View Options: General

View Name	Specifies the name of the view.Click to rename it.
UCS	Assigns a UCS: 1. Click the **UCS** droplist 2. Choose WCS or the name of a saved UCS When the drawing contains no saved UCSes, the droplist shows <None>.See the Ucs (6.30.2) and ExpUcs (6.13.18) commands.
Background	Sets the background for the view: 1. Click**Browse**; displays the Background dialog box 2. Choose a background style 3. Click **OK**. The program returns to the Drawing Explorer. See the Background (6.6.1) command.

Clip Display	Assigns a clip display: 1. Click the **Clip Display** droplist 2. Select a section plane
Layer Snapshot	Toggles whether to assigned the current on and off settings of layers to the view: • **Yes** - assigns the current visual state of layers, so that the next time you make this view current, the layers turn themselves on or off • **No** - doesn't do that
Visual Style	Sets the visual style: 1. Click the **Visual Style**droplist 2. Choose a visual style name See the VisualStyles (6.31.13) command.

Edit View Options: View

Camera	Specifies the camera point of the view in x,y,z coordinates in visual perspective mode.To edit the coordinates: 1. Click the **Camera** field 2. Enter new x, y, and/or z coordinates
Target	Specifies the target point of the view in x,y,z coordinates in visual perspective mode.To edit the coordinates: 1. Click the **Target**field 2. Enter new x, y, and/or z coordinates
Perspective	Toggles p**erspective**view: • **On** - view is displayed in visual perspective mode • **Off** - view displays in parallel perspective mode A camera glyph is displayed in the drawing for perspective views (see the Camera (6.9.2) command).
Lens Length	Sets the lens length of the camera.To edit the length: 1. Click the **Lens Length**field 2. Enter a new distance: • 50mm is normal view • 55mm or higher is a telephoto view • 45 or lower is a wide angle view

Roll Angle	Rotates the camera around the view axis. To edit the length: 1. Click the **Roll Angle**field 2. Enter a new angle
Width	Specifies the width of the view in current units.To edit the length: 1. Click the **Width**field 2. Enter a new width of view, in drawing units
Height	Specifies the height of the view in current units.To edit the height: 1. Click the **Height**field 2. Enter a new height of view, in drawing units

Edit View Options: Clipping

Front Plane	Sets the distance between the target point and the back clipping plane.To move the front plane: 1. Click the **Front Plane**field 2. Enter a new distance, in drawing units
Back Plane	Sets the distance between the target point and the back clipping plane.To move the back plane: 1. Click the **Back Plane**field 2. Enter a new distance, in drawing units
Clipping	Toggles the clipping planes, which remove from any entities that are outside the planes: 1. Click the **Clipping** droplist 2. Choose an option: • **Off** - both clipping plans are turned off • **Front on** - the front clipping plane is turned on • **Back on** -the back clipping plane is turned on • **Front and back on** - both the front and back clipping planes are turned on

Procedures

- Exploring views (1.16.10)
- Saving a view (1.16.10)
- Restoring a view (1.16.10)
- Defining the view background (1.16.10)
- To edit the view background (1.16.10)

Related Commands

-View (6.31.2) - controls named views through the command bar. Background (6.6.1) - sets the model space background for the current viewport and saved views. Camera (6.9.2) - creates perspective views. MatchPerspective (6.21.7) - matches the current model space perspective view with a background image by selecting 3 or more matching point pairs. Print (6.24.46) - plots named views. -Rename (6.26.21) - renames views through the Drawing Explorer. Zoom (6.35.7) - utilizes the Previous option to show the previous view. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.2497



6.31.2 -View

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Creates and sets named views through the command bar.

Accessing the Command

command bar: **-view** alias: -**v** menu bar: **View | Save/Restore View** ribbon:**View | View | Viewpoint | Save/Restore Views : -View Prompts you in the command bar:** View [? to list/Delete/Orthographic/Restore/Save/Window]: **(Enter an option, or else enter the name of a hidden option)**

- Hidden orthographic view options: Top/Bottom/Front/Back/Left/Right
- Hidden isometric view options: SWiso/SEiso/NEiso/NWiso

Command Options

Option	Description
? to list	Lists the names of views saved in this drawing; prompts you: **View(s) to list <List all views>** - **(enter a name or press Enter to list the names of all saved views)**; the programreports something like this: `Name of view: NewView1 Space: Model Height of view: 10.` Press **F2** to see the Prompt History window.
Delete	Erases named views from the drawing; prompts you: **View names to be deleted** - **(enter one or more names; separate multiple names with commas.)**
Orthographic*	Switches to an orthographic view; prompts you: **Enter an option: [Top/Bottom/Front/BAck/Left/Right]<Top>:(enter an option.)** If the UCSORTHO (1.13.11) system variable is ON the related orthographic UCS is restored automatically.
Restore	Restores a named view; prompts you: **View to restore** - **(enter the name of a view.)**
Save	Saves the current viewpoint as the named view; prompts you: **Save view as -(enter a name.)**
Window	Saves a windowed area as the view; prompts you: 1. **First corner of view windo**w - **(pick a point or enter x,y coordinates.)** 2. **Opposite corner** - **(pick another point or enter x,y coordinates.)**

*The orientation of the orthographic and isometric views is based on the coordinate system specified by the UCSBASE (1.13.11) system variable, which is set to the WCS (world coordinate system) by default. When one of the orthographic or isometric views is restored, BricsCAD zooms to the extents in the current viewport.

Related Commands

View (6.31.1) - controls named views through the Drawing Explorer. MatchPerspective (6.21.7) - matches the current model space perspective view with a background image by selecting 3 or more matching point pairs. Print (6.24.46) - plots named views. Rename (6.26.20) - renames views through the Drawing Explorer. Zoom (6.35.7) - utilizes the Previous option to show the previous view.

Figure 6.2498



6.31.3 ViewBase

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Generates associative orthographic and isometric views of 3D models in paper space layouts. **Note:** This command launches only in model space.

Accessing the Command

 command bar: **viewbase** transparent: **'viewbase** menu bar: **Model | Drawing Views | Generate Drawing Views** ribbon: **Annotate | Drawing Views | Generate Drawing Views** toolbar: **Drawing Views | : ViewBase Prompts you in the command bar:** Preset: "None", View scale: "Adapt to paper size" Select objects or [Entire model/preseTs/Special views] <Entire model>: **(Select one or more 3D solids in model tab, or else press Enter to select all of them; or else enter an option)** Enter new or existing layout name to make current <LayoutName>: **(Enter the name of a new or existing layout, or else press Enter to accept the current layout)** The command switches to the layout tab: Select position for base view [Scale/Hidden lines/Tangent lines/Orientation/Projection type/Isometric geometry/sElect]<Cancel>: **(Pick a point at which position the base view, or else enter an option)** Select position for projected view <Done>: **(Move the cursor left, right, above or below the base view to create other views, and click; continue until the final view is inserted or press Enter to finish early)**

Command Options

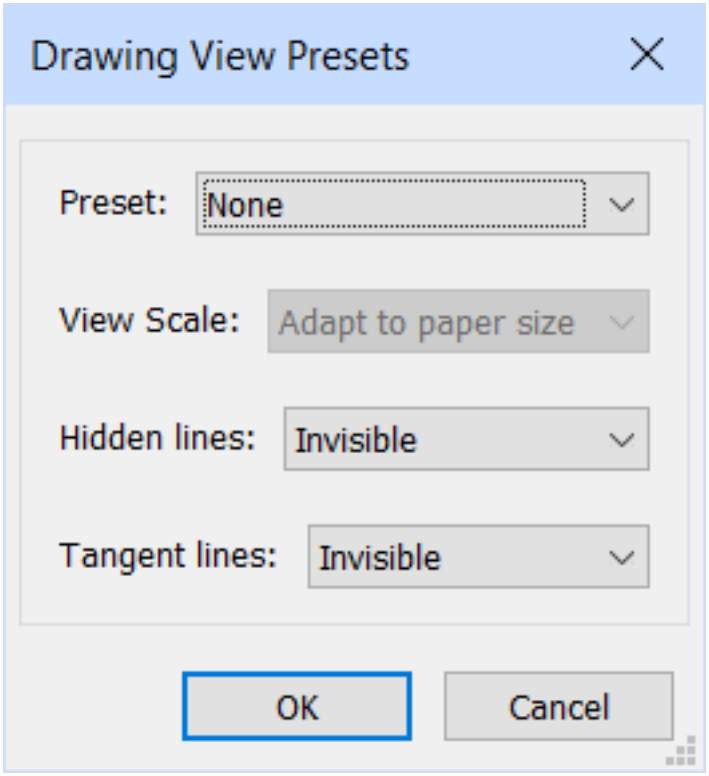
Option	Description
Select objects	Selects the entities in model space from which to generate the drawing views in a layout tab.
Entire model	Select all the 3D entities in model space.
preseTs	Specifies the types of generated drawings and their placement in the layout; displays the Drawing View Presets dialog box; see below. The selected preset is saved through the DRAWINGVIEWPRESET (1.13.11) system variable.
Special views	Select a style of exploded representations, if any exist in the drawing; prompts you: **Select view [Exploded view/Back] <Back>** - **(enter anoption:)** • **Exploded view** -displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/helpbricsyscom/hc/articleattachments/36001 From the droplist, choose a style of exploded representation • **Back** - returns to the previous prompt To create exploded views, use the bmExplode command; before this command can be used, the drawing must have inserts of mechanical components.

Scale	Sets the Scale property of the paper space viewports for the various views; prompts you: **Adjust view scale [fit 4 views/9 views/5 views/10 views/Standard scales/Custom] <fit 4 views>** - **(enter anoption:)** • **fit 4 views** - adjusts the scale is adjusted to fit the four standard orthographic views: Front, Top, Left, Right.The Front view (base view) is defined by the Orientation (6.31.3) option • **9 views** - adjusts the scale to fit five orthographic views and four isometric views. • **5 views** - adjusts the scale to fit five orthographic views: Front, Top, Left, Right, Back. • **10 views** - adjusts the scale to fit six orthographic views and four isometric views. < • **Standard scales** - displays the scale list as maintained by the ScaleListEdit (6.27.7) command; select a scale from the list. • **Custom** - prompts you to type a scale in the command bar.
Hidden Lines	Controls the visibility of hidden lines; prompts you: **Show hidden lines [Yes/No]** - **(enter an option:)** • **Yes** - displays hidden lines in the section view • **No**- turns off hidden lines When the BM_Ortho_Hidden and BM_Isometric_Hidden layers are off or frozen, hidden lines will not display. .
Tangent Lines	Toggles whether tangent edges between tangent faces are created.Visible tangent edges are created on the BM_Tangent_Visible layer. Hidden tangent edges are created on the BM_Tangent_Hidden layer.Prompts you: **Enable tangent edges on basic views? [Yes/No] <No>** - **(enter an option:) No**- does not display tangent lines: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N **Yes**- displays tangent lines: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N Tangent edges are always created for isometric views. Freeze or switch off the display of the BM_Tangent_Visible layer to hide tangent edges in isometric views. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/N
Orientation	Defines the base view. Rotates the 3D model so that the main view is projected on the vertical projection plane (V.P.). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/C Prompts you: **Select orientation [Front/bAck/Left/Right/Top/Bottom] <Front>** - **(enter an orientation to change the base view.)**

Projection type	Defines the layout of the views; prompts you: **Select the projection type [First angle/Third angle] <First angle>** - **(enter a projection type:)** • **First angle** - also known as European projection: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/V {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/V • **Third angle** - also known as American projection: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/V {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/V The default projection type depends on the value of the MEASUREMENT (1.13.11)variable: • **0** - (Imperial) generates the third angle projection • **1** - (Metric) generates the first angle projection
Isometric Geometry	Defines the style for the isometric views: rendered 3D view or 2D drawing.Prompts you: **Create isometric views as [2D views/3D views] <2D views>** - **(enter an option:)** • **2D views** - draws isometric views as 2D drawings: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou • **3D views** - draws isometric views as 3D solids with theConceptual visual style applied. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou
Select	Selects additional entities to include or exclude; prompts you: **Select objects to add to set or [Remove/Entire model/Layout] <Layout>** - **(enter an option:)** • **Remove** - removes entities from drawing views • **Entire model** - includes all entities from model space in the drawing views • **Layout**- opens the previous layout with updated drawing views
Select position for projected view	Depending on the position of the cursor with respect to the base view one of five orthogonal views (top, left, right, back and bottom) and four isometric views can be inserted. The views are aligned automatically. Tap the **Ctrl** key to toggle alignment on and off; when off, you can place the current view in any location.
Done	Concludes the command.

When theGENERATEASSOCIVIEWS (1.13.11)variable is on, associative dimensions are updated automatically when the 3D model is modified.

Figure 6.2499



Drawing View Presets Dialog Box

To access this dialog box, enter the **preseTs** option: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/C presets-dlg.png})

Dialog Box Option	Description
Preset	Select a preset view arrangement: 1. Click the **Preset** droplist 2. Choose an option • **None** - no preset affects the views generated • **Architectural Views** - generates – Five standard views: Front, Right, Back, Left (used for elevations) and Top views – Three section views: two horizontal sections (for plans) and 1 vertical section • **Mechanical Views** - generates – Three standard views: Front, Top and Left (first angle projection) or Right (third angle projection) The layout of the mechanical views is controlled by the projection type: first or third angle. The current projection type is saved in the PROJECTIONTYPE variable.
View Scale	Sets the default scale factor for the drawing views: 1. Click the **View Scale** droplist 2. Select a scale factor. The scale list can be edited by the ScaleListEditcommand.

Hidden Lines	Sets the default setting for hidden lines in drawing views: 1. Click the **Hidden Lines**droplist 2. Choose **Visible** or **Invisible**.
Tangent Lines	Sets the default for Tangent Lines in drawing views: 1. Click the **Tangent Lines**droplist 2. Choose **Visible** or **Invisible**.

Related Commands

Flatshot (6.14.9) - creates a hidden line representation of 3D solids in model space. MvSetup (6.21.39) - creates multiple paper space viewports; aligns, rotates and scales paper space viewports. In model space the command creates a rectangle, representing a paper sheet with respect to a specified scale. Solprof (6.27.62) - creates a hidden line representation of 3D solids in a layout viewport. ViewDetail (6.31.4) - creates a detail view of a portion of a standard generated drawing at a larger scale. ViewDetailStyle (6.31.5) - allows to specify the visual format of detail views and detail symbols. ViewEdit (6.31.6) - allows changing the scale and the hidden line visibility of drawing views. ViewExport (6.31.7) - Exports the content of drawing views obtained by ViewBase and ViewSection (6.31.10) to the Model Space of the drawing or to a new drawing. ViewProj (6.31.8) - generates additional projected views from an existing drawing view. ViewSection (6.31.10) - creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout. ViewSectionStyle (6.31.11) - allows to specify the visual format of section views and section lines. ViewUpdate (6.31.12) - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

Figure 6.2500

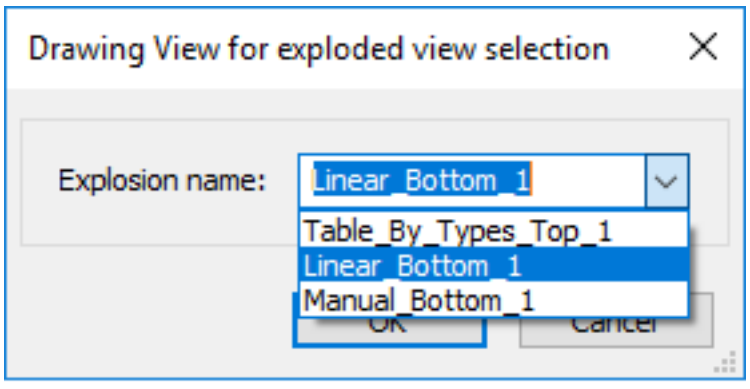


Figure 6.2501

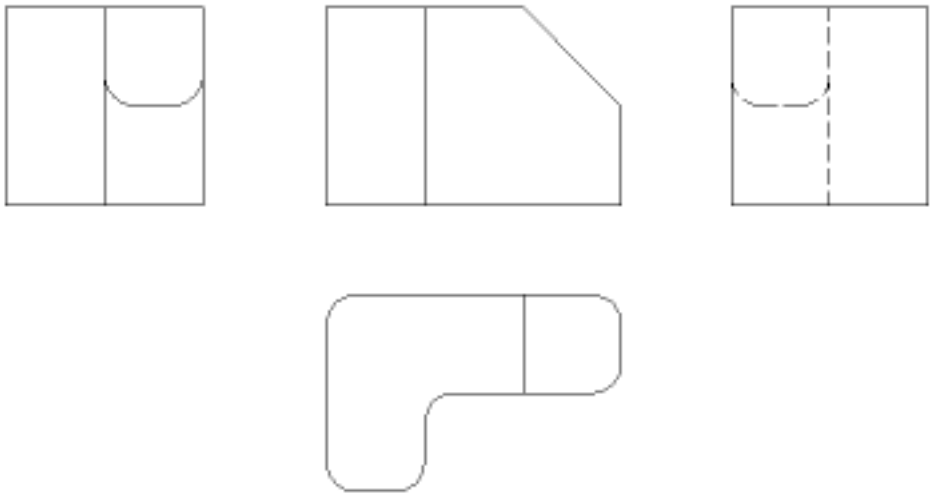


Figure 6.2502

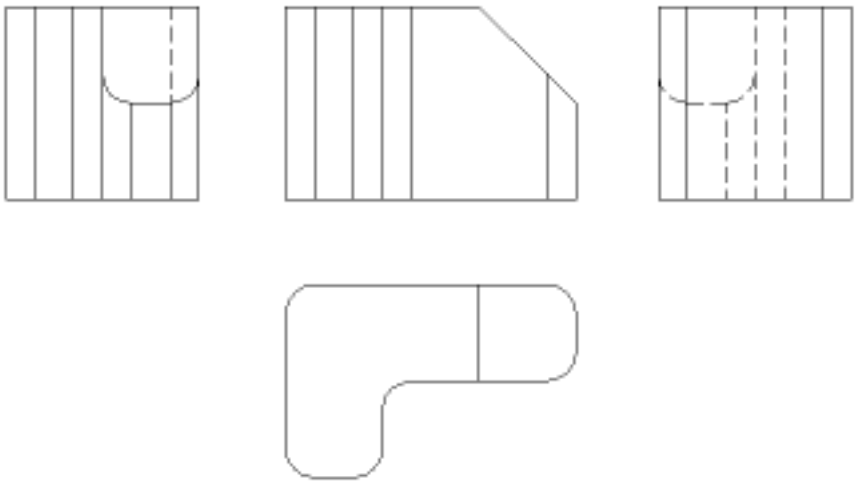


Figure 6.2503

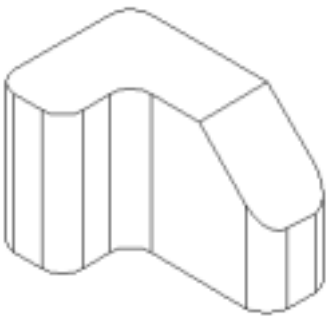


Figure 6.2504

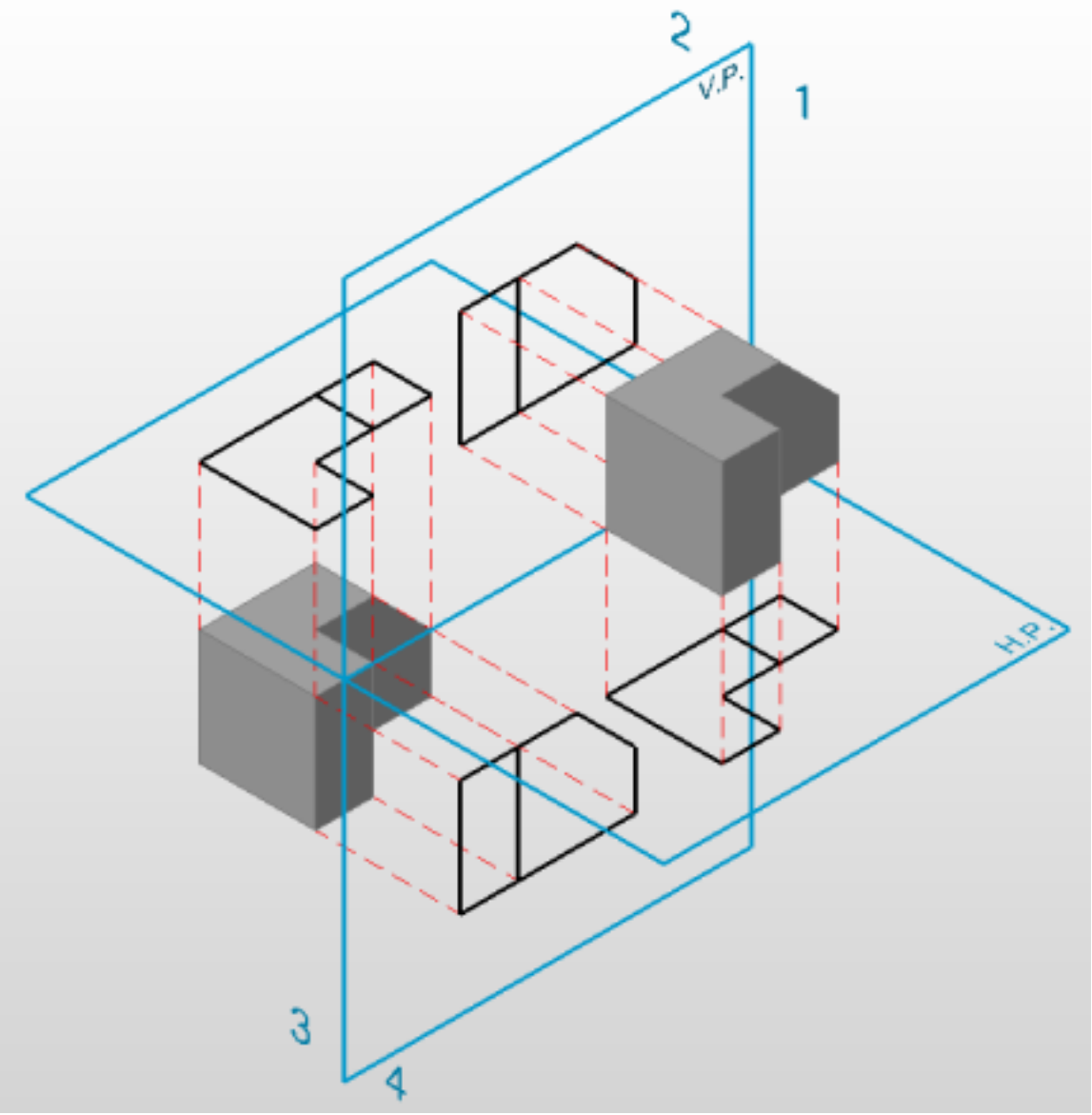


Figure 6.2505

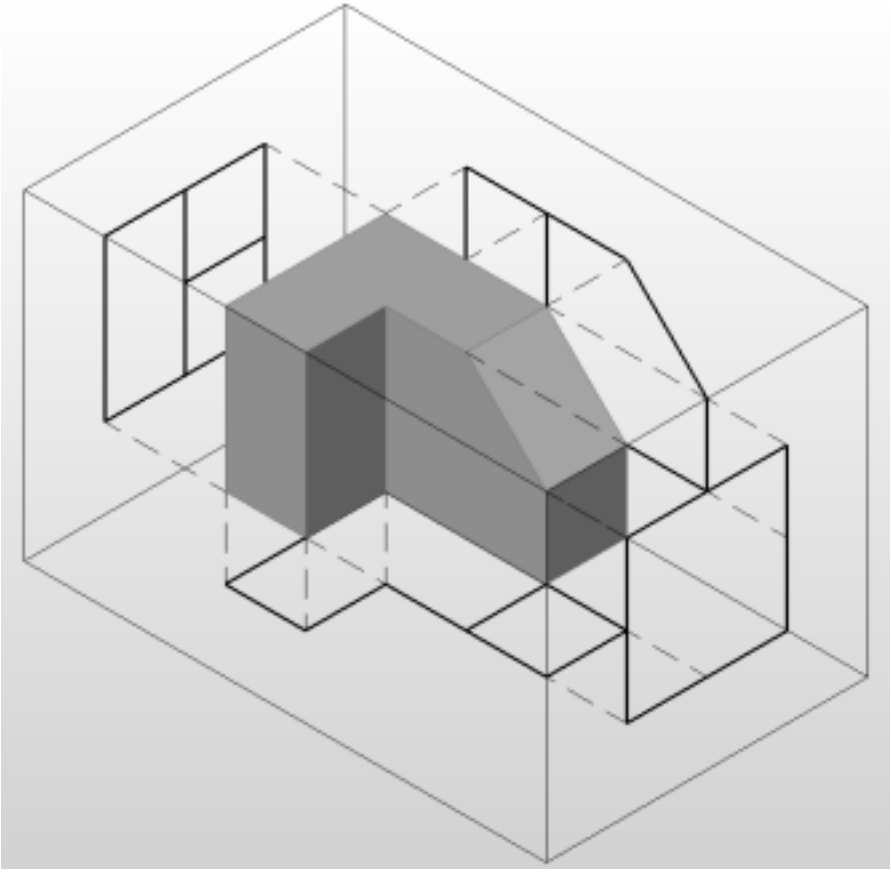


Figure 6.2506

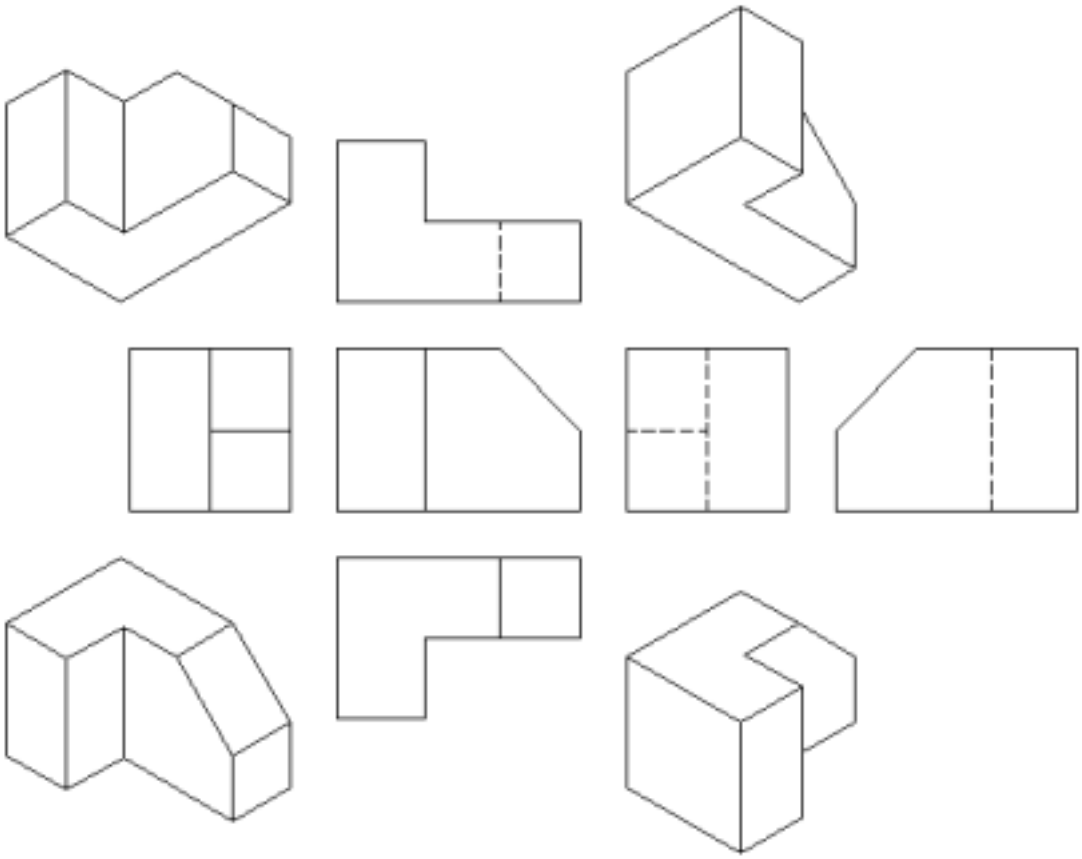


Figure 6.2507

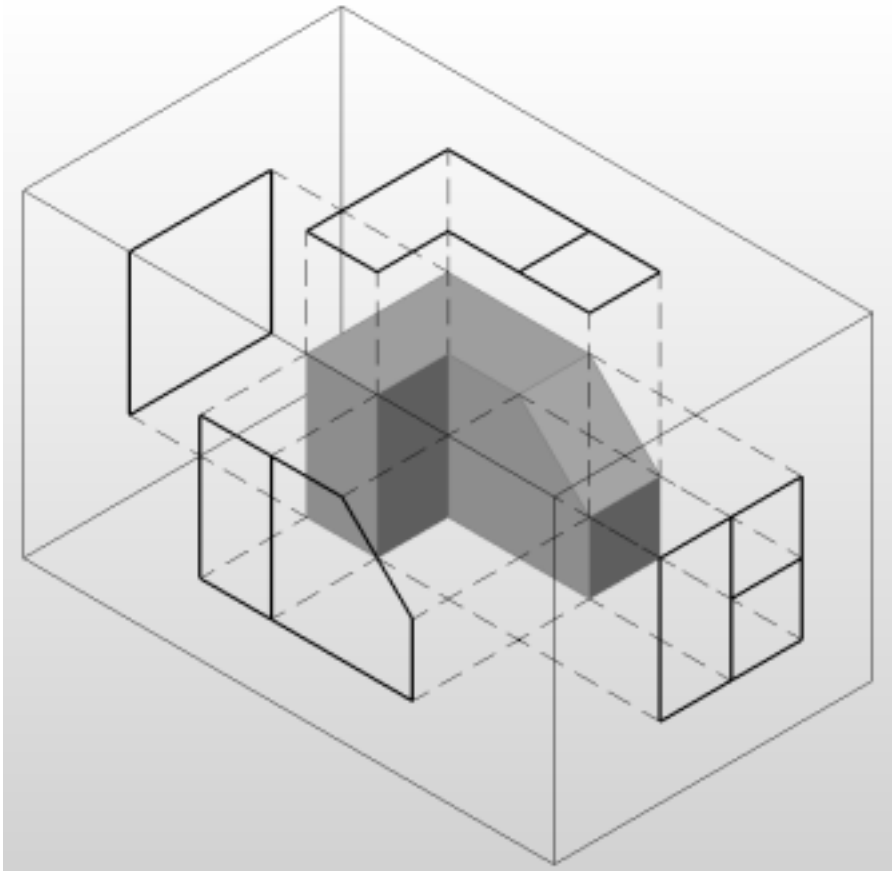


Figure 6.2508

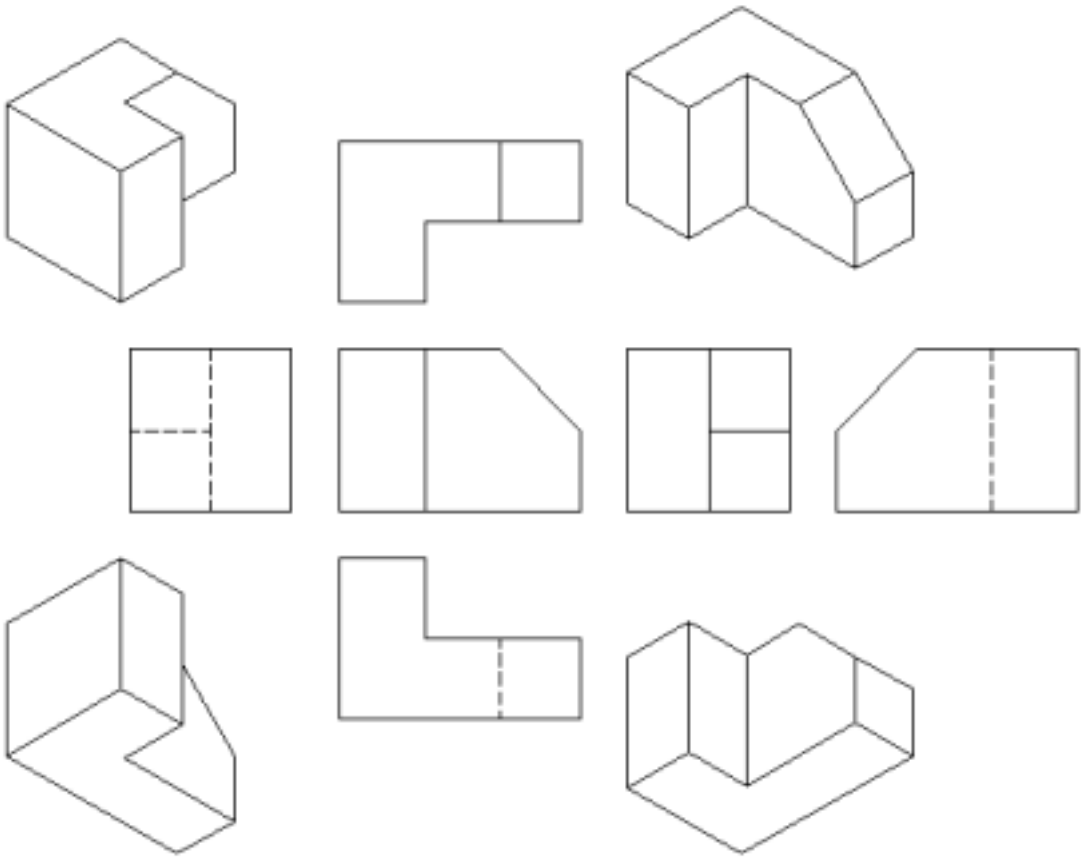


Figure 6.2509

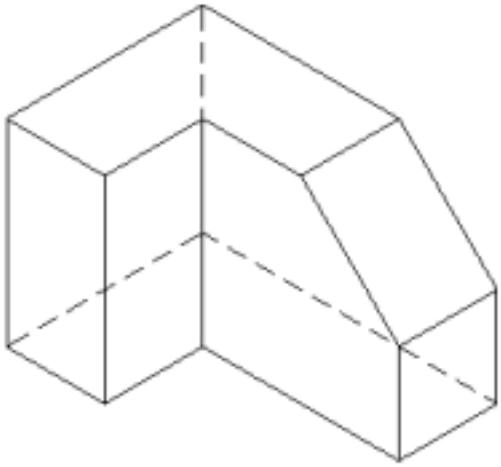


Figure 6.2510

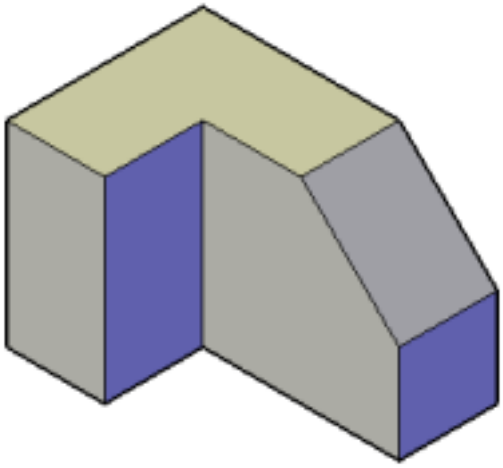
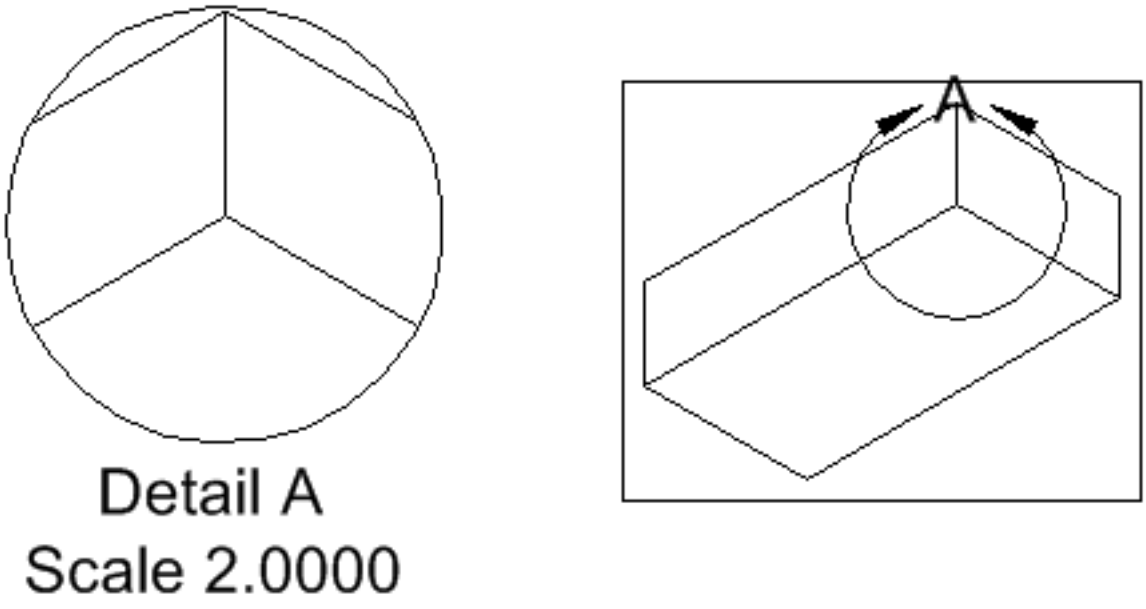


Figure 6.2511



Figure 6.2512



6.31.4 ViewDetail

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Draws view details of drawing views made withtheViewBase (6.31.3)command. **Note:** This command operates only paper space. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/eg.png})

Accessing the Command

 command bar: **viewdetail** menu bar: **View | Drawing Views | Detail View** toolbars:**Drawing Views |** **Detail Section Types |** ribbon: **Annotate | Drawing Views |** **: ViewDetail Prompts you in the command bar:** Select drawing view: **(Click inside a drawing view; the program highlights the selected view)** Specify detail center or select boundary type [Circular/Rectangular] <Circular>: **(Pick a point at the center of the detail view, or else choose an option)** Select radius of detail view: **(Pick a point or enter a number to set the radius of the detail view)** Select position for detail view [Scale] <Cancel>: **(Pick a point to position the detail view, or enter an option)** Select option [Scale/Hidden lines/Tangent lines/anChor/Annotation/Boundary/modEl edge] <Cancel>: **(Select an option, or else press Enterto end the command)**

Command Options

Option	Description
Select drawing view	Select the drawing view from which to made the detail: click inside a drawing view; the program highlights the selected view
Specify detail center	Specifies the center point of the detail view: pick a point inside the parent view

Select boundary type	Specifies the style of the view boundary: • **Circular** - draws a circular boundary; prompts you: **Select radius of detail view** - **(indicate the point with the cursor, or type a value)** • **Rectangular** - draws a rectangular boundary; prompts you: **Select radius of detail view**- **(indicate the point with the cursor, or type a value)** You can always use theBoundaryoption switch between circular and rectangular boundaries.
Select position for detail view	Specifies the location for the detail view: pick a point in the layout away from the parent view
Scale	Specifies the scale of the detail view, which by default is twice the parent viewport scale; prompts you: **Adjust view scale [Standard scales/ Custom/ Relative custom/Exit] <Standard scales>** - **(enter an option:)** • **Standard scales** - choose a standard scale in the list; thelist can be edited by the SCALELISTEDIT (6.27.7) command. • **Custom**- specify a custom scale factor; prompts you: **Enter scale <1.00>** - **(enter a scale factor.)** • **Relative custom**- calculates the scale factor relative to parent view; prompts you: **Enter scale <2.00>** - **(enter a number)**; the scale factor of the source view is multiplied by this number.
Hidden Lines	Control the visibility of hidden lines; prompts you: **Show hidden lines [Yes/No/from Parent]** - **(enter an option:)** • **Yes** - displays hidden lines in the section view • **No**- turns off hidden lines • **from Parent**- uses the same hidden lines setting as the parent view When the BM_Ortho_Hidden and BM_Isometric_Hidden layers are off or frozen, hidden lines will not display.
Tangent Lines	Controls the display of tangent edges, which appear in the transition from a flat face and a curved face, such as with fillets; prompts you: **Show tangent lines [Yes/No]** - **(enter an option:)** • **Yes**- displays tangent lines: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • **No**- does not display tangent lines (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so

anChor	Determines how the viewport changes relative to the section;prompts you: **Anchor view on paper space? [Yes/No]** - **(enter an option:)** • **Yes**- thecenter of the viewport is anchored so that the viewport grows and shrinks around its center point • **No-** the position of the geometry is fixed The AUTOVPFITTING (1.13.11)variable controls whether the size of the viewport is adjusted automatically to fit the current extents of the 3D geometry. By default AUTOVPFITTING = ON.
Annotation	Determines the annotations to use; prompts you: **Select annotation [Identifier/Label]** - **(enter an option:)** • **Identifier**- specifies the view detail identifier. prompts you: **Specify the identifier <A>** - **(enter a name for the view detail.)** • **Label**- toggles the display of the view detail label; prompts you: **Show view label [Yes/No]** - **(press Y or N)**
Boundary	Toggles the style the boundary between circular and rectangular; prompts you: **Detail boundary [Rectangular/Circular]** - **(choose an option:)** • **Circular** - changes the boundaryto a circle • **Rectangular** -changes the boundaryto a rectangle
Model Edge	Controls whether a connection line is drawn between the detail view and the detail boundary in the parent view; prompts you: **Model edge type [smooth with Border/smooth with Connection line]** - **(enter an option:)** • **smooth with Border** - does not draw a linem • **smooth with Connection line** - draws a connecting line from the detail view to the parent view

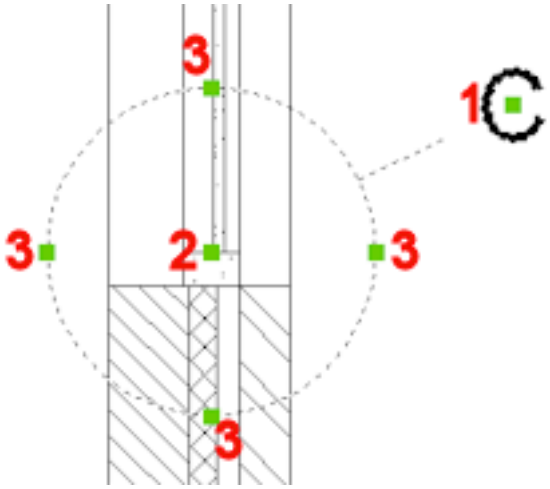
The GENERATEASSOCVIEWS (1.13.11)variable controls whether VIEWDETAIL produces 2D views which are persistently associated with the source 3D model.If GENERATEASSOCIVIEWS is ON, associative dimensions are updated automatically when the 3D model is modified.

Grips Editing

Detail views can be edited through grips: select the view and six grips display: (\vref {fig:C:/Dev/Task-2448/images/staticbricsy

1. Controls the position of the identifier
2. Controls the position of the section detail.
3. 4 grips control the size of the detail boundary.

Figure 6.2513



Related Commands

ViewBase (6.31.3) - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout. ViewExport (6.31.7) - exports the content of drawing views and sections obtained by VIEWBASE and ViewSection (6.31.10) to the Model Space of the drawing. ViewDetailStyle (6.31.5) - allows to specify the visual format of detail views and detail symbols. ViewEdit (6.31.6) - allows changing the scale and the hidden line visibility of drawing views. ViewExport (6.31.7) - Exports the content of drawing views obtained by ViewBase (6.31.3) and ViewSection (6.31.10) to the Model Space of the drawing or to a new drawing. ViewProj (6.31.8) - generates additional projected views from an existing drawing view. ViewSection (6.31.10) - creates a cross section view based on a standard drawing view generated by the ViewBase (6.31.3) command in a paper space layout. ViewSectionStyle (6.31.11) - allows to specify the visual format of section views and section lines. ViewUpdate (6.31.12) - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

Figure 6.2514

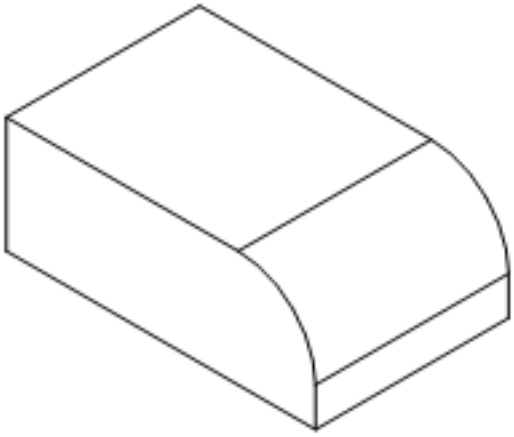


Figure 6.2515

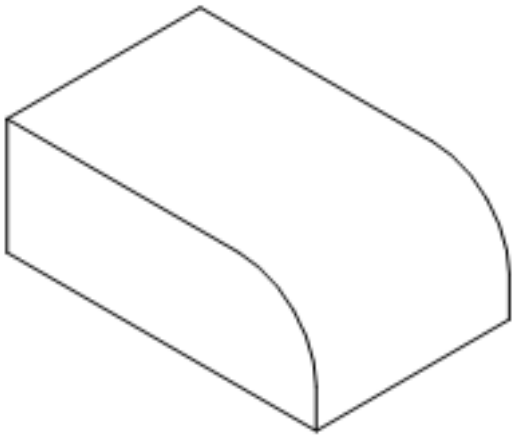
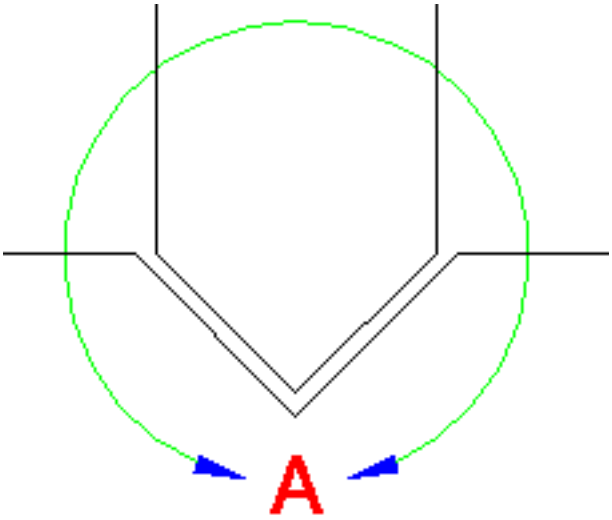


Figure 6.2516



Figure 6.2517



6.31.5 ViewDetailStyle

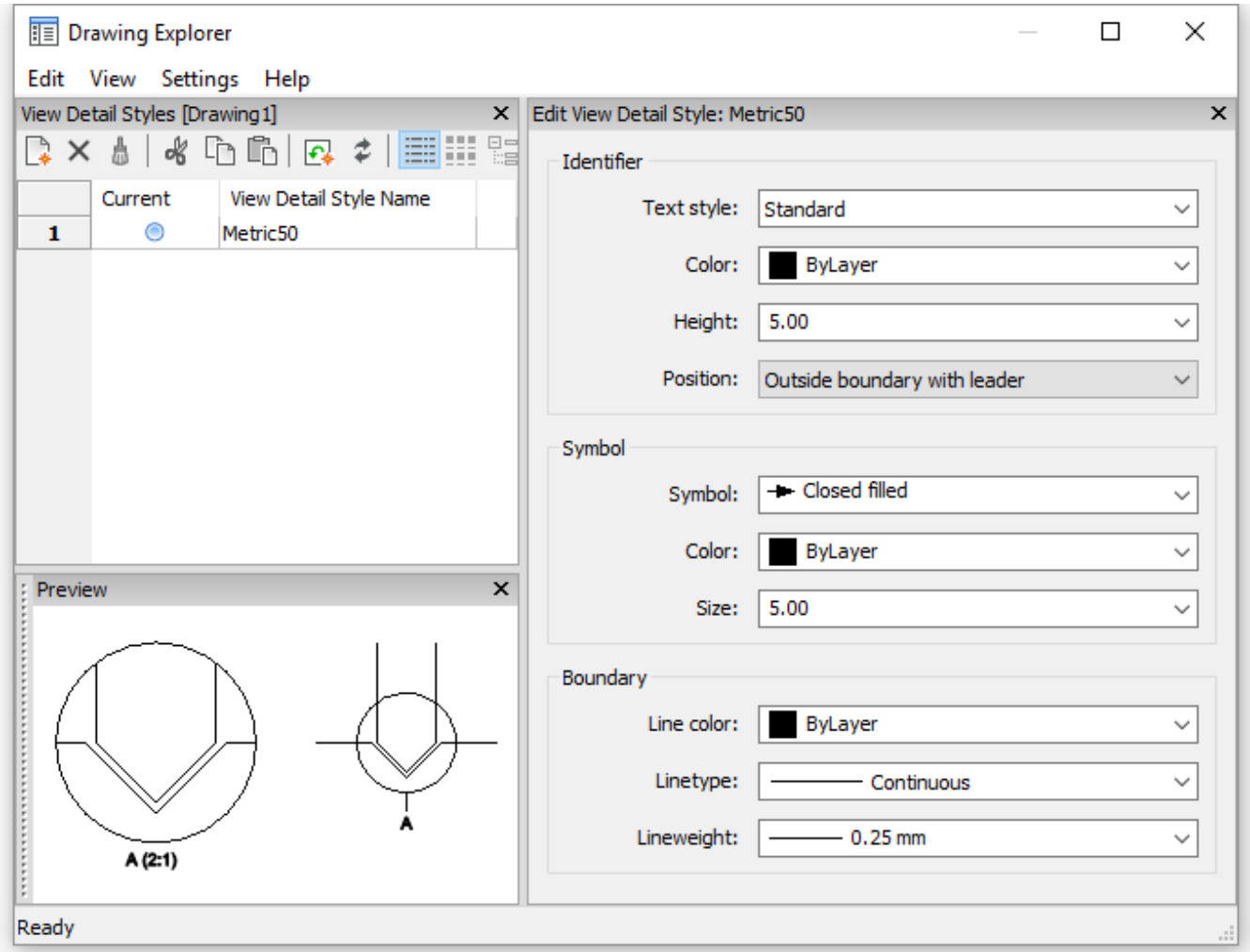
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Specifies the identifiers, symbols, and boundaries used by detail views made from drawing views, created by theViewBase (6.31.3)command, in the Drawing Explorer. (\vref {fig:C:/Dev/Task-2448/images/static eg.png})

- **Identifier** - red
- **Symbol** - blue
- **Boundary** - green

Accessing the Command

 command bar: **viewdetailstyle** menu bar: **View | Drawing Views | Detail Style** ribbon:Annotate | Drawing Views|  toolbar: **Drawing Views |**  **: ViewDetailStyle** Displays the View Detail Styles section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/DrawingExplorerVi

Figure 6.2518



Toolbar Option

Toolbar Option	Description
 New	Creates a new view detail style as a copy of the currently selected style.
Delete	Removes the selected style from the drawing.

Edit View Detail Style Options

Option	Description
Identifier	Defines the properties of the view detail identifier: **Text Style** - specifies the text style used by the identifier text: 1. Click the droplist 2. Choose the name of a text style To employ a different text style, use the Style command to create it. **Color** - specifies the color of the identifier: 1. Click the droplist 2. Choose a color swatch from the list, or else choose **Select Color** to get access to a lot more colors **Height** - specifies the height of the identifier: 1. Click the droplist 2. Choose one of the preset heights, or else type in a different value **Position** - determines where the identifier is located: • **Outside boundary** - places the identifier outside the detail's boundary; no symbol is used • **Outside boundary with leader** -outside the detail's boundary and draws a leader from the parent view to the detail;; no symbol is used • **On boundary**-on the detail's boundary • **On boundary with leader** -places the identifier on the detail's boundary and draws a leader from the parent view to the detail
Symbol	Defines the properties of the symbol, which typically is an arrow: • **Symbol** - specifies the look of the symbol; choose None for no symbol • **Color** - specifies the color of the symbol • **Size** - specifies the size of the symbol

Boundary	Defines the properties of the detail viewport boundary: • **Line color** - specifies the color of the boundary line • **Linetype** -specifies the line type of the boundary line • **Lineweight** -specifies the weight of the boundary line

Related Commands

ViewBase (6.31.3) - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout. ViewDetail (6.31.4) - creates a detail view of a portion of a standard generated drawing at a larger scale. ViewSection (6.31.10) - creates a cross section view based on a standard drawing view generated by the ViewBase (6.31.3) command in a paper space layout. ViewSectionStyle (6.31.11) - allows to specify the visual format of section views and section lines.

Figure 6.2519



6.31.6 ViewEdit

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Changing the scale and hidden line visibility of drawing views created by theViewBase (6.31.3)command. **Note:**This command operates only paper space.

Accessing the Command

 command bar: **viewedit** transparent:**'viewedit** menu bar: **Model | Drawing Views | Edit View** ribbon: **Annotate | Drawing Views| Edit View** toolbar: **Drawing Views |** Quad cursor: **Model | Edit View : ViewEdit Prompts you in the command bar:** Select drawing views/<All views>: **(Click inside one or more drafting viewports, then press Enter)** The next prompt depends on the selected view type; when different view types are selected, only the shared options are available:

- Standard views:

Select option [Scale/Hidden lines/Tangent lines/anChor/Geometry/sElect] <Cancel>: **(Select an option)**

- Section views:

Select option [Scale/Hidden lines/Tangent lines/ anChor/ Geometry/ Annotation/ Depth/ Projection] <Cancel>: **(Select an option)**

- Detail views:

Select option [Scale/Hidden lines/Tangent lines/anChor/Annotation/Boundary/modEl edge] <Cancel>: **(Select an option)**

Command Options

Option	Description
Select drawing views	Selects the drawings views to modify: click inside one or more drafting viewports, or press **Enter** to select all of them in the current layout: • **Standard views** - see the ViewBase (6.31.3)command for the meaning of options • **Section views** - see theViewSection (6.31.10) command • **Detail views** - see theViewDetail (6.31.4) command

Related Commands

ViewBase (6.31.3) - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout. ViewExport (6.31.7) - exports the content of drawing views and sections obtained by ViewBase (6.31.3) and VIEWSECTION (6.31.10) to the Model Space of the drawing. ViewDetail (6.31.4) - creates a detail view of a portion of a standard generated drawing at a larger scale. ViewDetailStyle (6.31.5) - allows to specify the visual format of detail views and detail symbols. ViewExport (6.31.7) - Exports the content of drawing views obtained by ViewBase (6.31.3) and ViewSection (6.31.10) to the Model Space of the drawing or to a new drawing. ViewProj (6.31.8) - generates additional projected views from an existing drawing view. ViewSection (6.31.10) - creates a cross section view based on a standard drawing view generated by the ViewBase (6.31.3) command in a paper space layout. ViewSectionStyle (6.31.11) - allows to specify the visual format of section views and section lines. ViewUpdate (6.31.12) - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

Figure 6.2520



6.31.7 ViewExport

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Exports drawing views generated by theViewBase (6.31.3) and ViewSection (6.31.10)commands to the model tab of the current drawing, via the Clipboard to a new drawing, or as a DWG/DXF file on disc. **Note:** This command operates only paper space.

Accessing the Command

command bar: **viewexport** transparent: **'viewexport** menu bar: **View| Drawing Views | Export Drawing Views to Model Space** toolbar: **View | Drawing Views |** ribbon: **Annotate | Drawing Views| : ViewExport Prompts you in the command bar:** Select drawing views/<All views>: **(Select one or more views, or press Enter to export all of them)** Choose destination [Clipboard/File/Model space] <Model space>: **(Select an option)**

Command Options

Option	Description
Select drawing views	Click inside the inside of one or more viewports; the program shows selected viewports by dashed lines.
All views	All views in the current layout will be exported.
Choose destination	Specifies the destination of the views: • **Clipboard** - exports the selected view(s) to the clipboard, which can then be pasted into another drawing • **File** - exports the selected views to a DWG or DXF file; displays the Save Exported Viewsdialog box • Model space - exports the view to the current drawing's model space; prompts you: 1. **Remove exported views? <No>/Yes** - **(enter an option:)** • **Yes**- the viewports of exported views are removed from the layout • **No**- viewports of exported views are kept in the layout 2. The program switches to model tab, and then prompts you: **Select position for exported views** - **(pick a point;)** the program places the views as blocks made of lines and other 2D entities

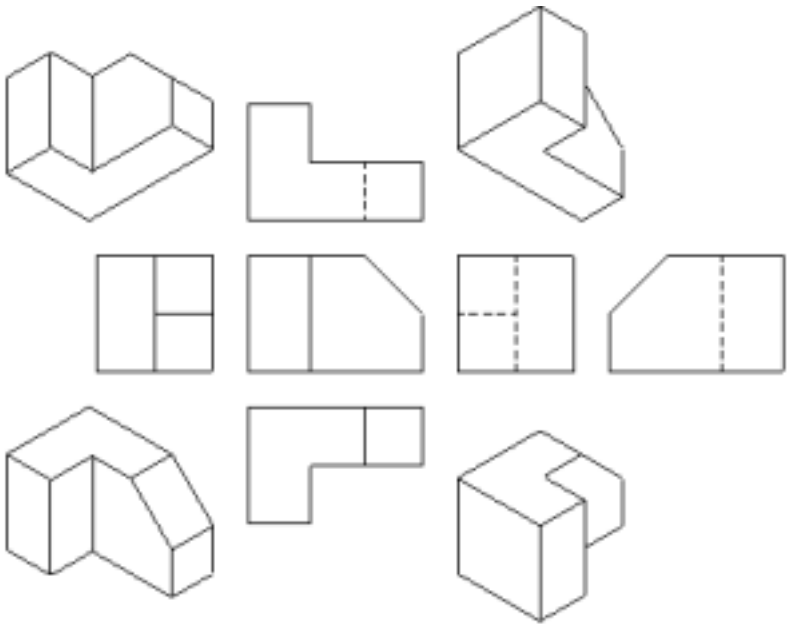
Related Commands

Flatshot (6.14.9) - creates a hidden line representation of 3D solids in model space. Solprof (6.27.62) - creates a hidden line representation of 3D solids in a layout viewport. ViewBase (6.31.3) - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout. ViewExport (6.31.7) - exports the content of drawing views and sections obtained by ViewBase. (6.31.3) ViewSection (6.31.10) to the Model Space of the drawing. ViewEdit (6.31.6) - allows changing the scale and the hidden line visibility of drawing views. ViewDetail (6.31.4) - creates a detail view of a portion of a standard generated drawing at a larger scale. ViewDetailStyle (6.31.5) - allows to specify the visual format of detail views and detail symbols. ViewProj (6.31.8) - generates additional projected views from an existing drawing view. ViewSection (6.31.10) - creates a cross section view based on a standard drawing view. ViewSectionStyle (6.31.11) - allows to specify the visual format of section views and

Figure 6.2521



Figure 6.2522



6.31.8 ViewProj

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Generates additional projected views from an existing drawing view previously created by the ViewBase (6.31.3) command (short for "view projection"). Up to five orthogonal and four isometric views can be generated from an existing drawing view: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou

Accessing the Command

command bar: **viewproj** transparent: **'viewproj** menu bar: **View| Drawing Views | Projected Views** ribbon: **Annotate | Drawing Views|** toolbar: **Drawing Views |** **: ViewProj Prompts you in the command bar:** Select drawing view: **(Click inside an existing drawing view)** Select position for projected view [Isometric geometry] <Cancel>: **(Position the projected views with respect to the existing drawing view, or press Enter to end the command)**

Command Options

Option	Description
Select drawing view	Selects the view from which to project the new view: click inside an existing drawing view
Select position of projected view	Positions the new view: move the cursor, and then click. Depending on where you move the cursor, you can placed one of five orthogonal views or four isometric views. The views are aligned automatically. Tap the **Ctrl**-key to toggle alignment on off; when off, you can place the view anywhere on the sheet.

Isometric geometry	Sets the style for isometric views; prompts you: **Create isometric views as [3D views / 2D views] <2D>** - **(enter an option:)** • **2D views** - draws isometric views as 2D projections of the 3D geometry, and turns on the BM_Isometric_Hidden layer so that hidden lines are shown as dashed lines • **3D views** - draws isometric views as 3D geometry, and sets the visual styleof the viewport (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so

Related Commands

ViewBase (6.31.3) - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout. ViewExport (6.31.7) - exports the content of drawing views and sections obtained by ViewBase (6.31.3) and ViewSection (6.31.10) to the Model Space of the drawing. ViewDetail (6.31.4) - creates a detail view of a portion of a standard generated drawing at a larger scale. ViewDetailStyle (6.31.5) - allows to specify the visual format of detail views and detail symbols. ViewEdit (6.31.6) - allows changing the scale and the hidden line visibility of drawing views. ViewExport (6.31.7) - Exports the content of drawing views obtained by ViewBase (6.31.3) and ViewSection (6.31.10) to the Model Space of the drawing or to a new drawing. ViewSection (6.31.10) - creates a cross section view based on a standard drawing view generated by the ViewBase (6.31.3) command in a paper space layout. ViewSectionStyle (6.31.11) - allows to specify the visual format of section views and section lines. ViewUpdate (6.31.12) - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

Figure 6.2523

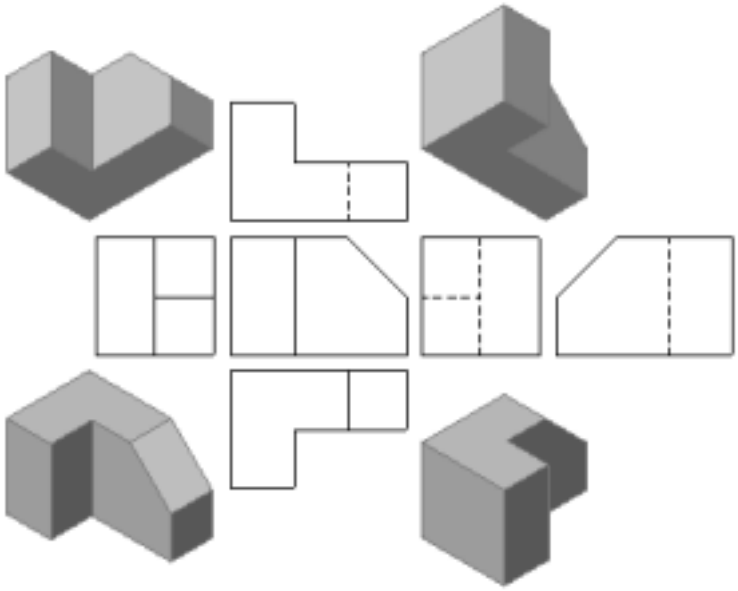


Figure 6.2524



6.31.9 ViewRes

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Sets the view resolution, and toggles fastzoom mode (short for "view resolution").

Accessing the Command

command bar: **viewres : ViewRes Prompts you in the command bar:** Do you want fast zooms? <Yes>: **(Type Y or N.)** Enter curve display quality (range 1-20000) <100>: **(Enter a number, or press Enter to accept the default.)** When ViewRes is set to a low value, zoomed-in circles can look like polygons: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image674.png}) Enter the Regen (6.26.15)command to make circles look round again: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sour

Figure 6.2525

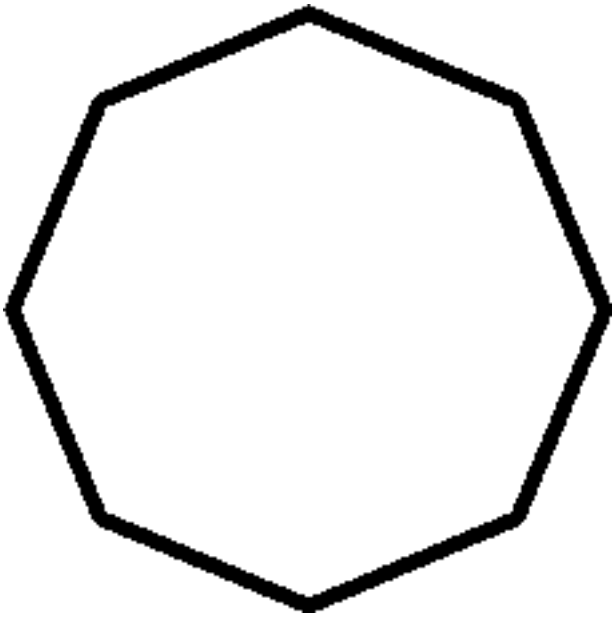
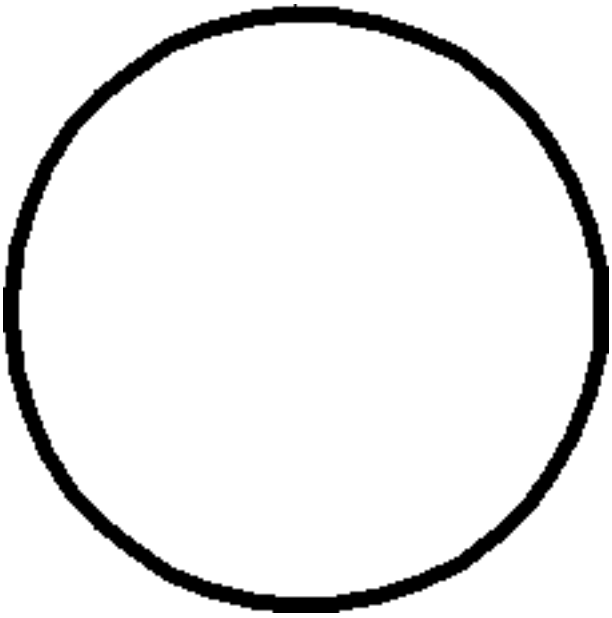


Figure 6.2526



Command Options

Option	Description
Do you want fast zooms?	Specifies how redraws are handled: • **Yes** - redraws and other commands that activate redraws refresh the drawing screen • **No** - redraws acts like regens: they re-read the drawing database and then refresh the screen
Enter curve display quality	Specifies the roundness of circles and other curved entities; enter a number between 1 and 20000: • **1** - very fast display, but circles can look like polygons • **100** - default value • **20000** - slower display, but circles almost always look round With today's fast computers and graphics boards, you can safely enter 20000.

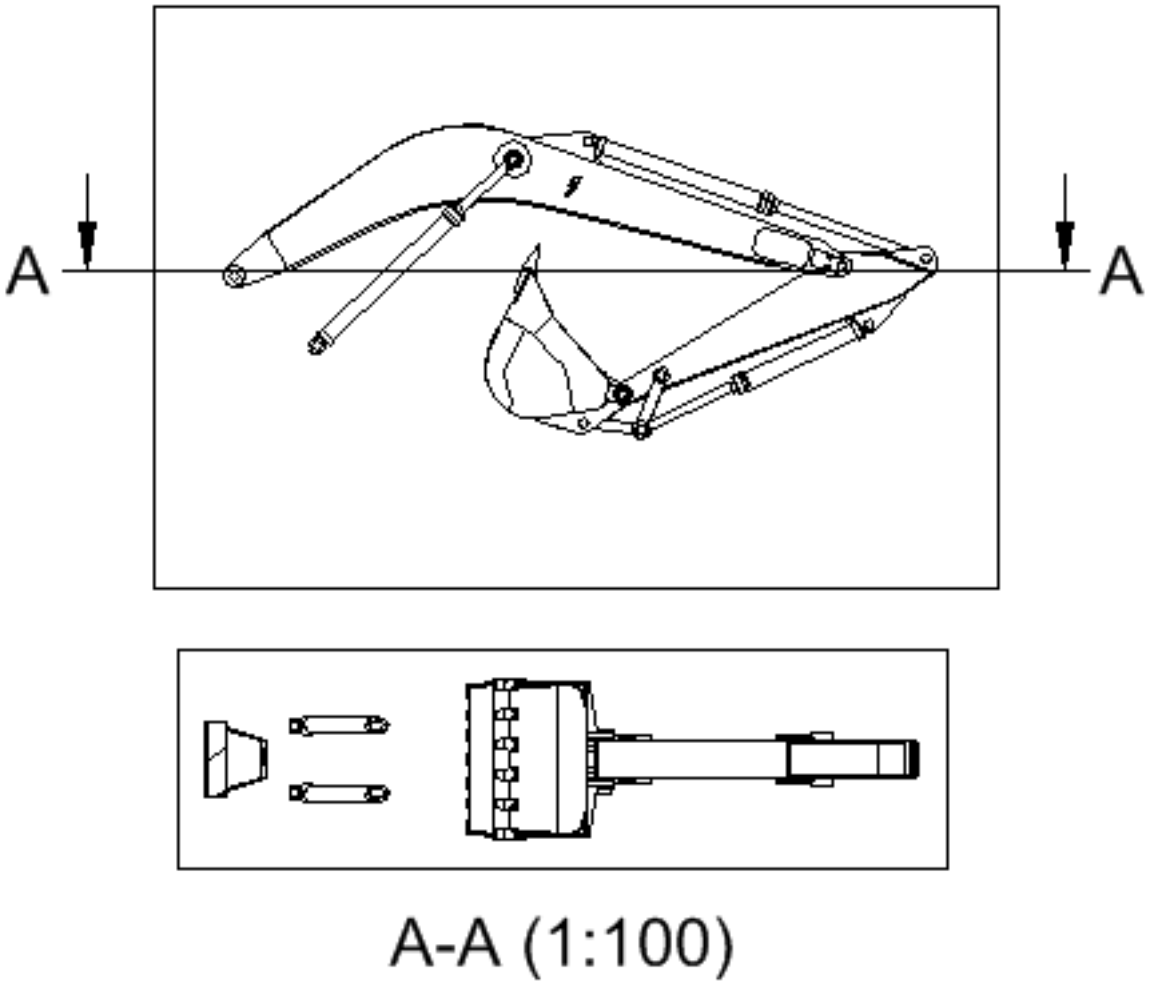
Related Command

Regen (6.26.15) - regenerates the drawing.

Figure 6.2527



Figure 6.2528



6.31.10 ViewSection

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Creates cross section views of drawing views that were generated by the ViewBase (6.31.3) command in a paper space layout. **Note:** This command operates only in paper space. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/viewsection-eg.png})

Accessing the Command

command bar: **viewsection** transparent: **'viewsection** menu bar: **Model | Direct Modeling | Drawing Views | Generate Section View Model | Direct Modeling | Drawing Views | Section Types** ribbon:**Annotate | Drawing Views |** toolbar:**Drawing Views | Section types |** : **ViewSection Prompts you in the command bar:** Select drawing view: **(Click inside a drawing view)** Specify start point of section line or select type [Full/Half/Offset/Aligned] <Full>: **(Pick a point, or enter an option)** Specify next point of section line [Done] <Done>: **(Pick another point)** As you move the cursor, the program draws the section view dynamically, parallel to the section line. Select position for section view: **(Pick a point to place the section view)** Select option [Scale/Hidden lines/Tangent lines/ anChor/ Geometry/ Annotation/ Depth/ Projection] <Cancel>: **(Enter an option, or else press Enter to finish the command)**

Command Options

Option	Description
Select drawing view	Selects the drawing view from which the section will be generated: click inside a drawing view,
Specify start point of section line	Specifies the one end of the section line; pick a point in the layout.
Specify next point of section line	Specifies the second point of the section line; prompts you: **Specify next point of section line [Done] <Done>** - **(enter an option.)** • Pick another point to create a jogged section line • Enter **Done** to end the section line drawing process The command repeats so that you can draw sections lines with jogs.
Select type	Control the shape of the section plane; prompts you: **Select type [Full/Half/Offset/Aligned] <Full>** - **(enteran option:)** 1. **Full**  - section line defines an infinite plane cutting through the entire model 2. **Half**  - section line defines a half-plane cutting part of the model; prompts you: 3. **Specify start point of section line or [Type] <Type>** - **(pick a point, such as #1 in the figure below.)** 4. **Specify next point of section line [Done] <Done>** - **(pick another point to define the limit of the half plane, such as at #2.)** 5. **Specify next point of section line [Done] <Done>** - **(pick the point that defines the view direction, such as #3.)** The position of this third point determines whether the part of the model that is not cut displays (left) or not (right). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so • **Offset**  - section line defines a series of cutting regions located on different offsets from each other; enter**Done**to complete the section line. • **Aligned**  - section line defines a polyline, where every segment defines a cutting region. The resulted section will have the length equal to the sum of the lengths of cutting regions; enter**Done**to complete the section line.

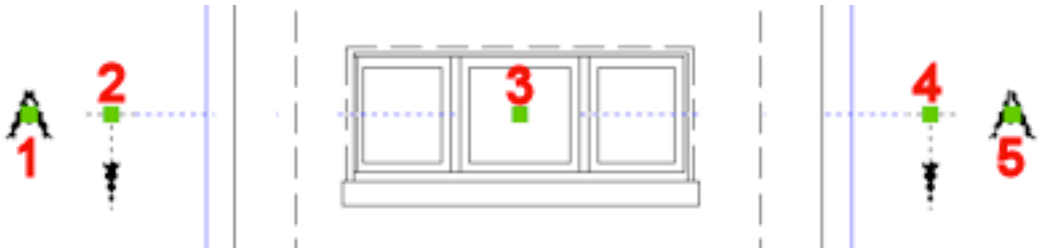
Select position for section view	Specifies the location of the section view: • As you move the cursor, the section view stays perpendicular to the section line • Tap the **Ctrl** key to turn off alignment so that you can place the section view anywhere on the sheet.
Scale	Sets the scale of the section view; prompts you: **Adjust view scale [Standard scales/Custom/from Parent]** - **(enteran option:)** • **Standard scales** - displays the list of scale factors defined by the ScaleListEdit (6.27.7) command; prompts you: **Enter the scale number <1>** - **(enter a number from the list displayed by the program.)** • **Custom** - prompts you to enter a scale in the command bar: **Enter scale <1.000>** - **(enter a scale factor.)** • **from Parent** -sets the scale of the section view equal to the scale of the parent viewport
Hidden lines	Allows to control the visibility of hidden lines; prompts you: **Show hidden lines [Yes/No/from Parent]** - **(enter an option:)** • **Yes** - displays hidden lines in the section view • **No**- turns off hidden lines • **from Parent**- uses the same hidden lines setting as the parent view When the BM_Ortho_Hidden and BM_Isometric_Hidden layers are off or frozen, hidden lines will not display.
Tangent lines	Controls the display of tangent edges, which appear in the transition from a flat face and a curved face, such as with fillets' prompts you: **Show tangent lines [Yes/No]** - **(enter an option:)** • **Yes**- displays tangent lines: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/Cm • **No**- does not display tangent lines (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/Cm
anChor	Determines how the viewport changes relative to the section;prompts you: **Anchor view on paper space? [Yes/No]** - **(enter an option:)** • **Yes**- thecenter of the viewport is anchored so that the viewport grows and shrinks around its center point • **No-** the position of the geometry is fixed The AUTOVPFITTING (1.13.11)variable controls whether the size of the viewport is adjusted automatically to fit the current extents of the 3D geometry. By default AUTOVPFITTING = ON.

Geometry	Selects the visual stylefor the section view; prompts you: **Choose geometry type [2D/3D]** - **(enter an option:)** • **2D**- section view uses the 2dWireframe visual style • **3D-** section view uses a rendered visual style, which is Conceptual by default;choose a different visual style for the viewport in the Properties (1.12.4) panel: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/Cm
Annotation	Determines the annotations to use; prompts you: **Select annotation [Identifier/Label]** - **(enter an option:)** • **Identifier**- specifies the view detail identifier. prompts you: **Specify the identifier <A>** - **(enter a name for the view detail)** • **Label**- toggles the display of the view detail label; prompts you: **Show view label [Yes/No]** - **(press Y or N.)**
Depth	Specifies the view depth of a section view; prompts you: **Choose depth mode [Full/Custom] -(enter an option:)** • **Full-** sets the depth to the extents of the model (maximum view depth) • **Custom-** limits the view depth; prompts you: **Enter depth distance** - **(Type a positive value, or move the cursor to define theview depth dynamically using the pointing device.)**
Projection	Determines how the section is projected; prompts you: **Select projection method [Normal/Orthogonal]** - **(enter an option:)** • **Normal**- draws the section as a union of projections of every cutting region (each segment of the section polyline) in its normal direction. • **Orthogonal**- draws the section in the normal direction of the first section region (first segment of the section polyline).

NOTE:

- The GENERATEASSOCVIEWS (1.13.11) system variable controls whether VIEWSECTION produces 2D views which are persistently associated with the source 3D model.
- If GENERATEASSOCIVIEWS (1.16.16) is ON, associative dimensions are updated automatically when the 3D model is modified.
- If the Sectionable property (6.8.29) of a mechanical component is OFF, the component appears non-sectioned in the section views of the **Full** section type.

Figure 6.2529



Grips Editing

Section lines can be edited through grips. Select either the section line, one of the identifiers or an arrow. 5 Grips display: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ViewSectionGripsEditing.png}) 1. Controls the position of the first identifier. 2. Defines the start point of the section line. 3. Allows to move the section line. 4. Defines the endpoint of the section line. 5. Controls the position of the second identifier.

Related Commands

Flatshot (6.14.9) - creates a hidden line representation of 3D solids in model space. Solprof (6.27.62) - creates a hidden line representation of 3D solids in a layout viewport. MvSetup (6.21.39) - Creates multiple paper space viewports; aligns, rotates and scales paper space viewports. In model space, the command creates a rectangle, representing a paper sheet with respect to a specified scale. ViewBase (6.31.3) - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout. ViewExport (6.31.7) - exports the content of drawing views and sections obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing. ViewDetail (6.31.4) - creates a detail view of a portion of a standard generated drawing at a larger scale. ViewDetailStyle (6.31.5) - allows to specify the visual format of detail views and detail symbols. ViewEdit (6.31.6) - allows changing the scale and the hidden line visibility of drawing views. ViewExport (6.31.7) - Exports the content of drawing views obtained by VIEWBASE (6.31.3) and VIEWSECTION (6.31.10) to the Model Space of the drawing or to a new drawing. ViewProj (6.31.8) - generates additional projected views from an existing drawing view. ViewSectionStyle (6.31.11) - allows to specify the visual format of section views and section lines. ViewUpdate (6.31.12) - updates a selection of drawing views or sections when VIEWUPDATEAUTO = 0.

Figure 6.2530

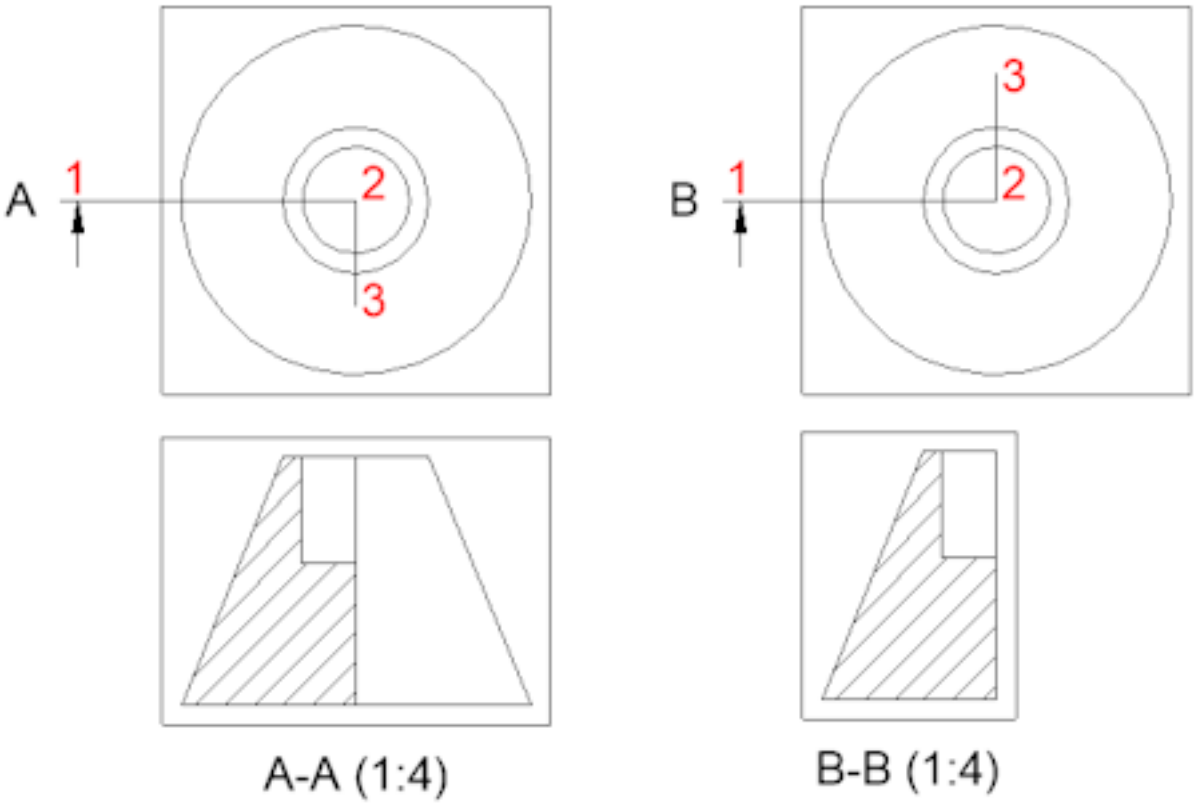


Figure 6.2531

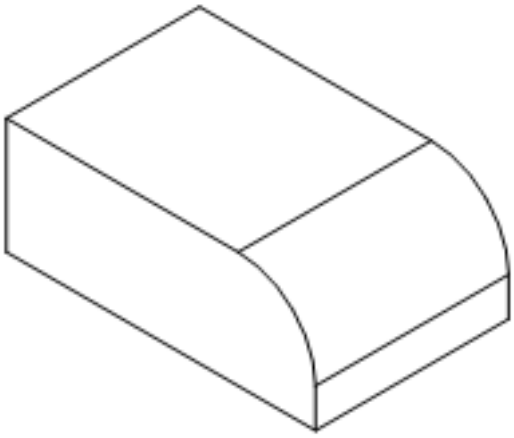


Figure 6.2532

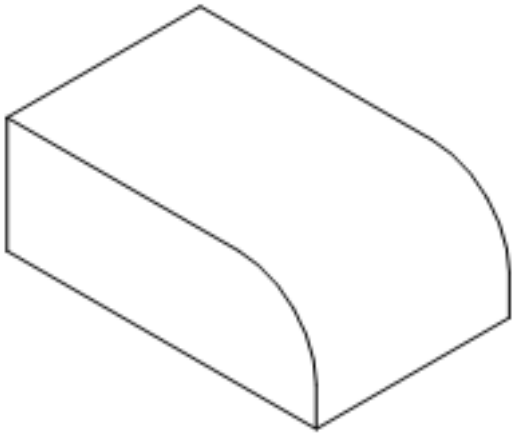


Figure 6.2533

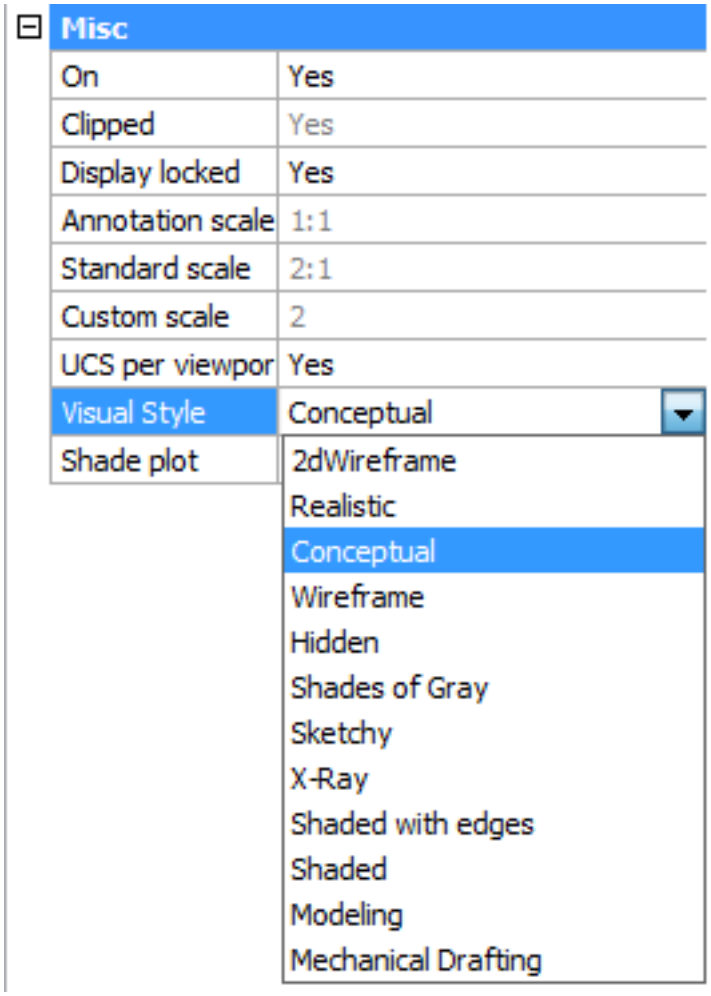
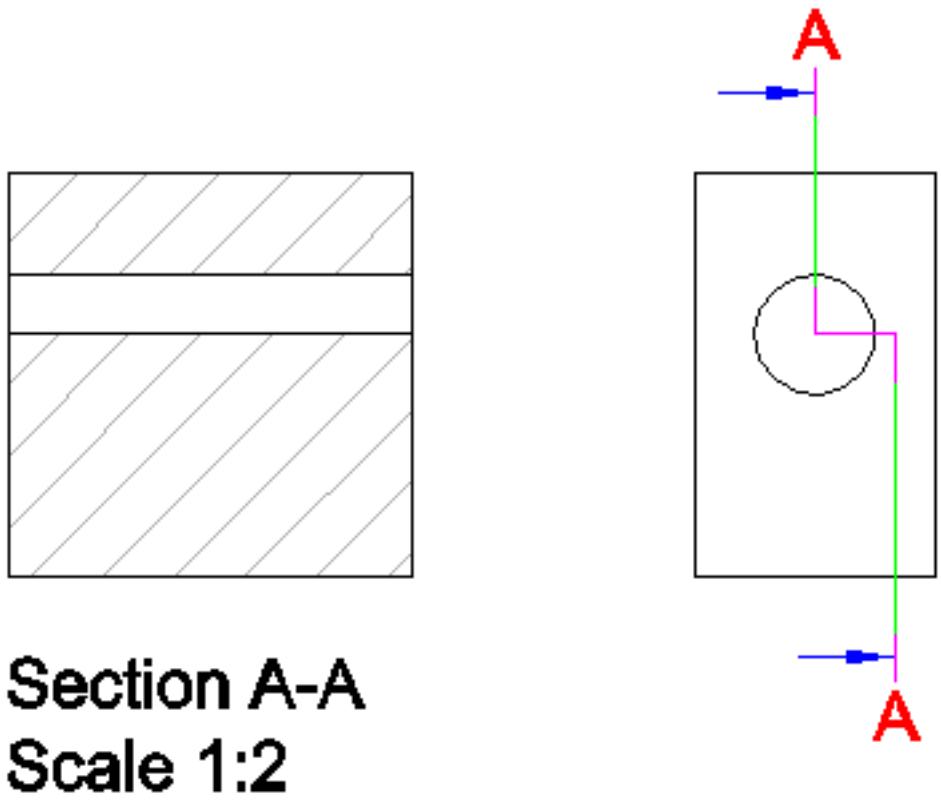


Figure 6.2534



Figure 6.2535



6.31.11 ViewSectionStyle

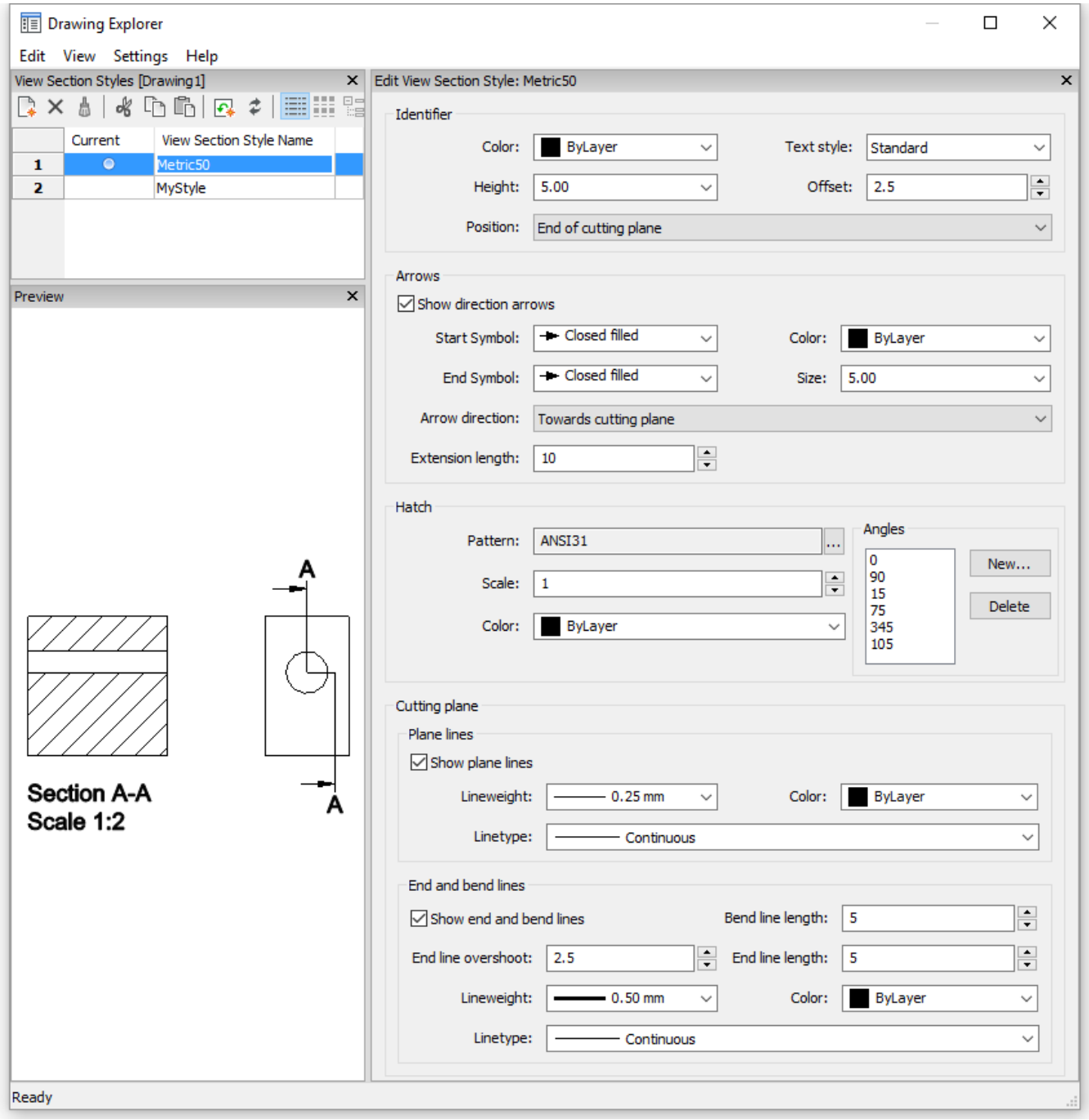
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Specifies the visual format of section views made from drawing views, created by theViewBase (6.31.3)command, in the Drawing Explorer. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/Veg.png})

- **Identifier**- red
- **Arrows**- blue
- **Hatch** - gray
- **Cutting plane**- green
- **End and bend lines -** pink

Accessing the Command

 command bar: **viewsectionstyle** transparent:'**viewsectionstyle** menu bar: **Drawing Views** | toolbar: **View| Drawing Views | Section Style** ribbon: **Annotate | Drawing Views|** **: ViewSectionStyle** Displays the View Section Styles section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V

Figure 6.2536



Toolbar Options

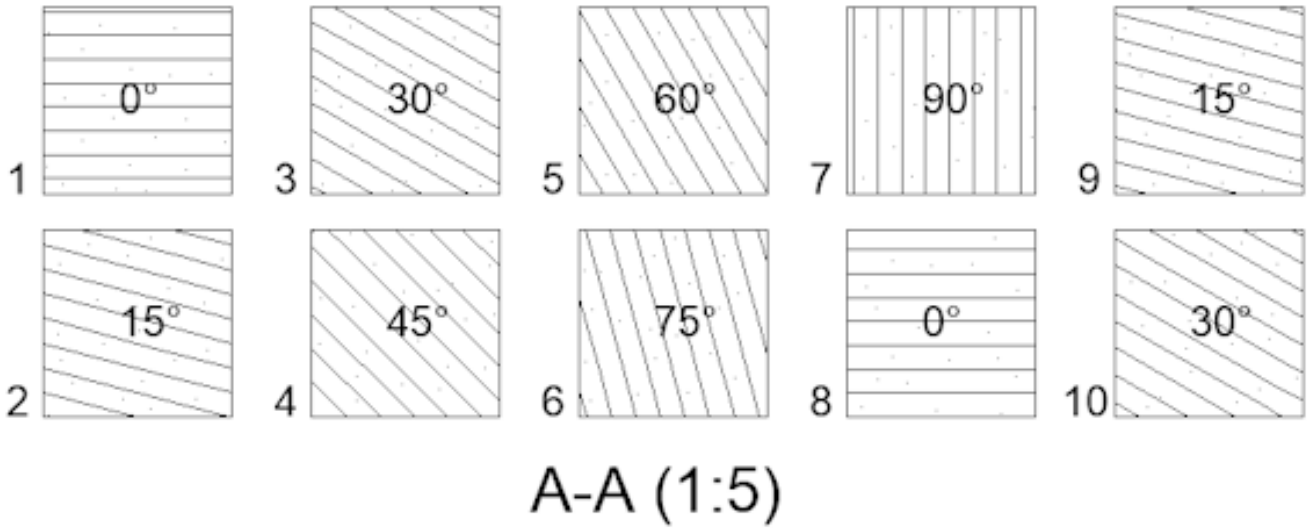
Toolbar Option	Description
 New	Creates a new view detail style as a copy of the currently selected style.
Delete	Removes the selected style from the drawing.

Edit View Section Style Options

Option	Description
Identifier	Sets the properties of the section identifier: **Color** - specifies the color of the identifier: 1. Click the droplist 2. Choose a color swatch from the list, or else choose **Select Color** to get access to a lot more colors **Text Style** - specifies the text style used by the identifier text: 1. Click the droplist 2. Choose the name of a text style To employ a different text style, use the Style (6.27.53) command to create it. **Height** - specifies the height of the identifier: 1. Click the droplist 2. Choose one of the preset heights, or else type in a different value **Offset** - specifies the distance from the end of the cut lines to the arrow; enter a distance. **Position** - determines where the identifiers are located: • **End of cutting plane**- places the identifiers at the ends of the cutting lines • **Above directionline**-outside the detail's boundary and draws a leader from the parent view to the detail;; no symbol is used • **On boundary**-on the detail's boundary • **On boundary with leader** -places the identifier on the detail's boundary and draws a leader from the parent view to the detail

Arrows	Sets the properties of the section arrows: • **Show direction arrows** - toggles the display of the arrows • **Start Symbol** - specifies the look of the first symbol • **Color** - specifies the color of the arrow • **End Symbol** - specifies the look of the first symbol • **Size** - specifies the size of the arrow • **Arrow direction** - points the arrow towards or away from the cutting line • **Extension length** - determines the length of the ”dimension” line on the arrows
Hatch	Sets the properties of the section hatching: • **Pattern** - choose a pattern name from the droplist • **Browse** - displays the Hatch Pattern Palette dialog box, from which you choose a pattern visually • **Angles** - sets the angle for the pattern, one angle for subsequent cuts; thefirst angle in the list is applied to the first solid, the second angle to the second, and so on. When there are more solids to cut than angles listed, then the program starts again from the first angle. See figure below. • **New** - adds angles to the list; displays the New Hatch Angle dialog box, in which you enter an angle • **Delete** - deletes the selected angle from the list • **Scale** - specifies the scale of the pattern • **Color** - specifies the color of the pattern (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/V
Cutting Plane | Plane Lines	Sets the properties of the cutting plane lines: • **Show plane lines**- toggles the display of the lines • **Lineweight** -specifies the weight of the lines • **Color** - specifies the color of the lines • **Linetype** -specifies the pattern of the lines
Cutting Plane | End and Bend Lines	Sets the properties of the end and bend lines: • **Show end and bend lines**- toggles the display of the lines • **Bend line length** - specifies the distance the bend line offsets from one cut line to another • **End line overshoot** - specifies the distance from the section to the end of the plane line • **End line length** - specifies the length of the end lines • **Lineweight** -specifies the weight of the lines • **Color** - specifies the color of the lines • **Linetype** -specifies the pattern of the lines

Figure 6.2537



Related Commands

ViewBase (6.31.3) - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout. ViewDetail (6.31.4) - creates a detail view of a portion of a standard generated drawing at a larger scale. ViewDetailStyle (6.31.5) - allows to specify the visual format of detail views and detail symbols. ViewSection (6.31.10) - creates a cross section view based on a standard drawing view generated by the ViewBase (6.31.3) command in a paper space layout.

Figure 6.2538



6.31.12 ViewUpdate

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Manually updates drawing views created theViewBase (6.31.3)and ViewSection (6.31.10)commands, when automatic updates (VIEWUPDATEAUTO) are turned off. **Note:** This command works only in paper space.

Accessing the Command

 command bar: **viewupdate** menu bar: **Model | Drawing Views | Update Drawing Views** ribbon: **Annotate | Drawing Views | Update Views** toolbar: **Drawing Views**  **: ViewUpdate Prompts you in the command bar:** Select drawing views/<All views>: **(Click inside a view, or press Enter to update all views)**

Command Options

Option	Description
Select drawing views	Click inside a view
All views	Updates all drawing views in the current layout

Related Commands

ViewBase (6.31.3) - generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout. ViewExport (6.31.7) - exports the content of drawing views and sections obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing. ViewDetail (6.31.4) - creates a detail view of a portion of a standard generated drawing at a larger scale. ViewDetailStyle (6.31.5) - allows to specify the visual format of detail views and detail symbols. ViewEdit (6.31.6) - allows changing the scale and the hidden line visibility of drawing views. ViewExport (6.31.7) - Exports the content of drawing views obtained by VIEWBASE (6.31.3) and VIEWSECTION (6.31.10) to the Model Space of the drawing or to a new drawing. ViewProj (6.31.8) - generates additional projected views from an existing drawing view. ViewSection (6.31.10) - creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout. ViewSectionStyle (6.31.11) - allows to specify the visual format of section views and section lines.

Figure 6.2539



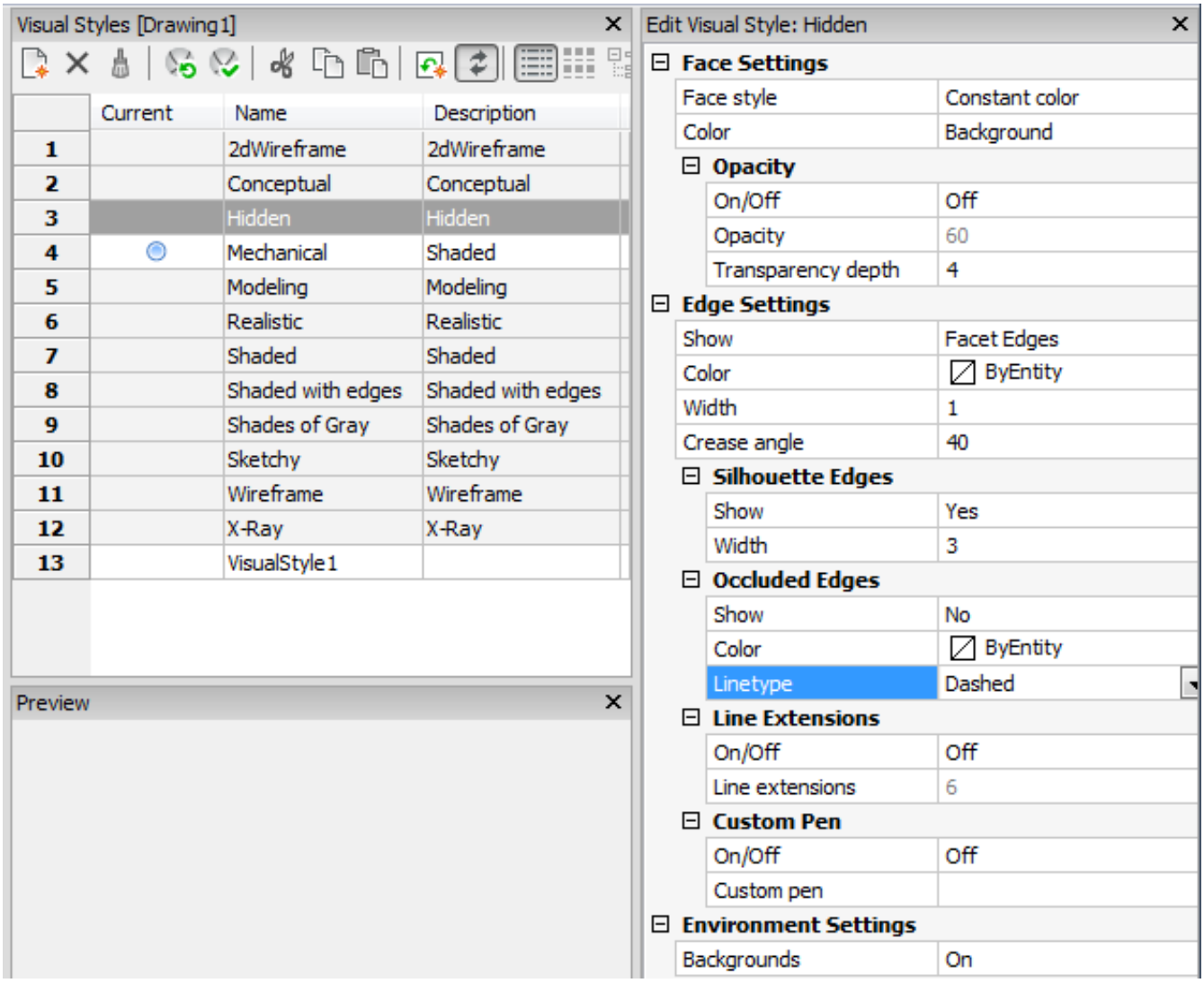
6.31.13 VisualStyles

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and modifies visual style definitions in the Drawing Explorer.

Accessing the Command


command bar: **visualstyles** menu bar: **Tools | Drawing Explorer | Visual Styles** ribbon: **Home | Settings | Drawing Explorer | Visual Styles** **: VisualStyles** Displays the Visual Styles section of the Drawing Explorer dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/visualstyles-dlg.png})

Figure 6.2540



Toolbar Options

Some options do not appear when certain settings are turned off.

Option	Description
===NO VALID URL FOR IMAGE=== New	Creates a new visual style.
 Delete	Deletes the selected visual style, except forpredefined visual styles such as 2dWireframe, 3D Hidden, 3dWireframe, Conceptual, Realistic, and so on.
 Apply visual style	Applies the selected visual style to the current viewport.
Edit visual style	Displays the settings of the selected visual styles.To edit a setting: click the settings field, then type a new value or select an option in the drop down list.

Edit Visual Styles Options

Option	Description

Face Settings	Specifies the look of faces of 3D models: **Face Style** - selects an overall color scheme for the visual style: • **Real** - makes the faces look real, like a rendered mode; displays materials ifapplied • **Gooch** - substitutes warm and cool colors for light and dark • **Constant Color** - applies a single color to each face • **None** - no style applied to faces, like wireframe mode **Lighting Quality**- specifies the quality of curved faces: • **Faceted** - lowest quality but fastest rendering • **Smooth** - good quality • **Smoothest** - best quality at cost of slower rendering **Color**- • **Normal** - face colors are displayed normally • **Monochrome** -face colors are displayed in a monochrome color, as specified by theMonochrome Color option • **Tint** -face colors have their hue and saturation changed • **Desaturate** -face colors have their saturation reduced by 30%. • **Background** - faces take on the background color **Material Display**- determines whether materials are displayed; materials are applied with the MaterialAssign (6.21.9) command: • **Materials and texture** - both materials (the look of the surface, such as brick) and the texture maps (bumpiness) are rendered, at the cost of rendering speed • **Materials** - materials are displayed on the faces of 3D models • **Off** - materials are not displayed, which leads to quicker renderings When Material Display is not off, then highlights andopacity are ignored, as materials provides their own values for them.
Opacity	Specifies the level of transparency of faces: **On/off** - toggles opacity on and off **Opacity** -specifies the level of transparency (Not available when materials are on): • **0** - Opaque • **1** - Transparent • **2** thru **99** - Translucent **Transparency depth** - specifies how far transparency reaches in terms of the number of overlapping entities

Lighting	Determines whether shadows are displayed. **Shadow display**- choose one: • **Off** — objects cast no shadows, the preferred option asshadow-casting slows down the program'sdisplay speed, so leave it turned off unless you are saving images to file or plottingthem • **Ground shadows** — objects cast shadows on the shadow plane (described later), butnot on each other • **Full shadows** — objects cast shadows on the ground and on each other
Edge Settings	Specifies the style of edge to display: **Show** - **(choose one:)** • **None** - neither facets nor isolines nor edges are displayed; this setting turns off many other settings • **Isolines** - isolines and edges are displayed; isoline are the curved lines that simulate the surfaces of curved faces • **Facet Edges** - facets and edges are displayed on entities **Color** -specifies the color for all edges **Width** - specifies the width of edge lines **Number of lines** -specifies the number of isolines drawn on curved surfaces, from 0 to 2047 **Always on top** - determines if all isolines are displayed, or only those that are "on top" (hidden isolines being hidden)
Silhouette Edges	Determines how fat outlines are around models: **Show** -toggles the display of silhouette edges **Width** -width of the silhouette edges in the range of 1 to 25 pixels;applies to all entities in the viewport equally
Occluded Edges	Specifies how to handle occluded (hidden) edges and facets;this setting allows you to display hidden lines in a different color and linetype from visible lines: **Show** -toggles the visibility of occluded edges and facets: • **On** - hidden lines are shown"through" the model • **Off** - hidden lines are not shown **Color** -specifies the color for visible obscured edges and facets; **Linetype** -specifies the line type for visible obscured edges and facets; the program does not use the usual linetype patterns, but a separate set: • Solid • Dashed • Dotted • Sparse dot • Short dash • Medium dash • Medium long dash • Long dash • Double short dash • Double medium dash • Double long dash All of these settings do not apply to isolines.

Line Extensions	Extends edges beyond their boundaries; also known as ”overhang”: **On/Off** - turn line extensions on or off **Line extensions** - specify the distance the lines extend beyond boundaries, such as face edges
Custom Pen	Determines the use of a custom line drawing style: **On/Off** - turns it on and off **Custom pen** - specifies the style of line drawing
Background Settings	Toggles the display of a background in the viewport: **Backgrounds**- turn on or off Backgrounds consist of either a solid color, a gradient of two or three colors, or a raster image, and are set by the Background (6.6.1) command.

The ANTIALIASSCREEN (1.13.11) system variable controls the amount of anti-aliasing (edge smoothing) applied during on-screen rendered mode display. The default value is 1, maximum value is 5. High anti-alias values incur a high calculation cost. **Mechanical Drafting** and **Mechanical Drafting Section** visual styles are used by the BmGenDraft (6.31.3) and BmGenSection (6.31.10) commands

Related Commands

AniPath (6.4.8) - records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file. Render (6.26.22) - generates a photorealistic rendering using materials and lights. Shademode (6.27.29) - sets the shading style for 3D drawings. -VisualStyles (6.31.14) - manages visual style definitions at the command line VsCurrent (6.31.21) - sets the visual style at the command line. Explorer (6.13.14) - Opens the Drawing Explorer dialog box.

Figure 6.2541



6.31.14 -VisualStyles

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Manages the definitions of visual styles at the command line.

Accessing the Command

command bar: **-visualstyles : -VisualStyles Prompts you in the command bar:** Visual styles: set_Current/Saveas/Rename/De **(Enter an option)**

Command Options

Option	Description
Set Current	Prompts you for a visual style to be set current: **Enter visual style Wireframe/ Hidden/ Realistic/ Conceptual/ Shaded/ shaded_with_Edges/ shades_of_Grey/ SKetchy/ X-ray/<Other>** - **(enter an option:)** • Type the capitalized letter of one of the default visual styles. • Press **Enter** to set a user-defined visual style; prompts you: **Enter a visual style name or [?]** - **(enter the name of a visual style, or else press?to see a list of the visual styles in the drawing.)**
Saveas	Saves the current visual style with a new name; prompts you: **Enter new visual style name or [?]** - **(enter the name of a visual style, or else press?to see a list of the visual styles in the drawing.)**
Rename	Renames a visual style; prompts you: 1. **Enter visual style to rename or [?]** - enter the name of a visual style, or else press**?**to see a list of the visual styles in the drawing; prompts you: 2. **Enter new visual style name** - **(type a new name.)**
Delete	Deletes a visual style; prompts you: **Enter visual style to delete or [?]** - **(enter the name of a visual style, or else press?to see a list of the visual styles in the drawing.)** You cannot delete default visual styles or the current visual style.
?	Lists the visual styles in the drawing; press the **F2** function key to open the **Prompt History** window: `2dWireframe Conceptual Hidden Mechanical Modeling`

Related Commands

AniPath (6.4.8) - records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file. Render (6.26.22) - generates a photorealistic rendering using materials and lights. Shademode (6.27.29) - sets the shading style for 3D drawings. VisualStyles (6.31.13) - creates and controls visual style definitions in the Drawing Explorer. VsCurrent (6.31.21) - sets the visual style at the command line.

Figure 6.2542



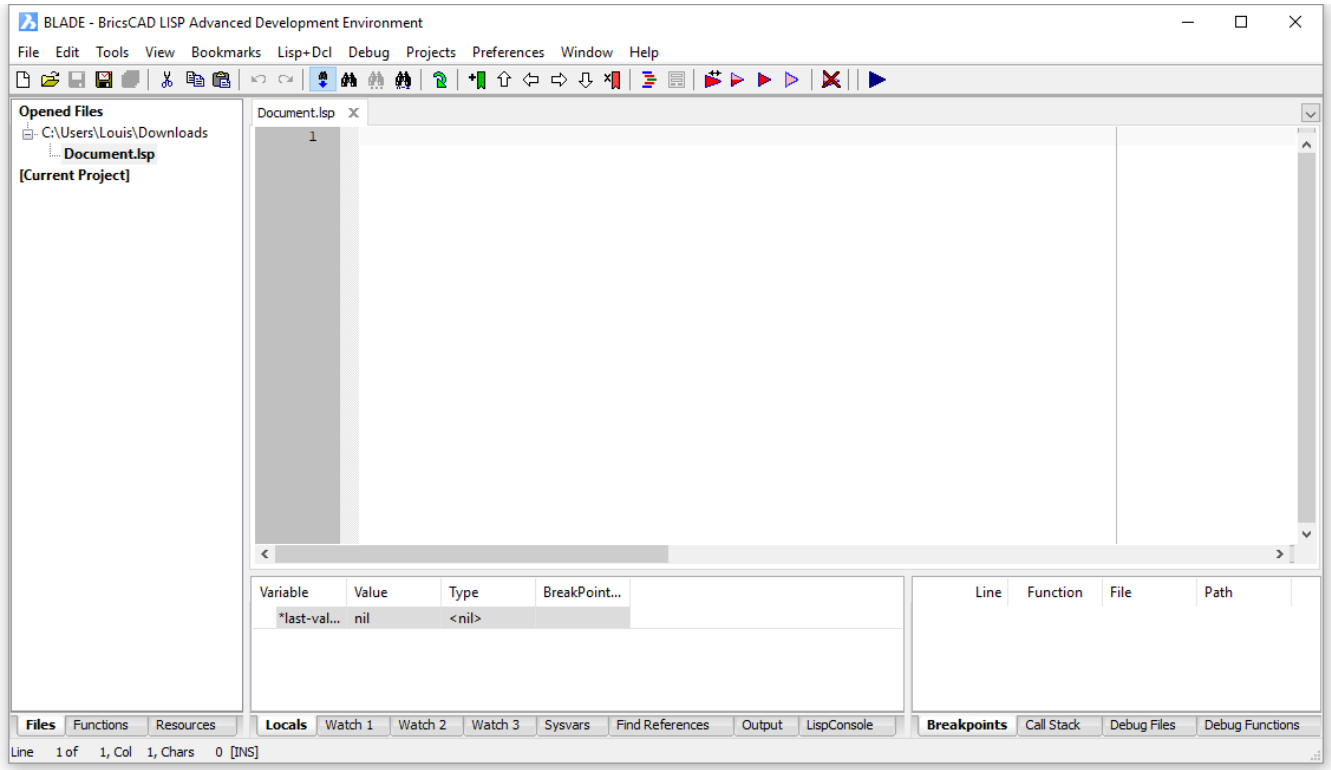
6.31.15 VLide

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the LISP Development Environment for editing and debugging LISP applications.

Accessing the Command

command bar: **vlide : VLide** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/Cm

Figure 6.2543



Command Options

No command options

Related Commands

Blade (6.6.9) - Opens the advanced environment for editing and debugging LISP applications. AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. VbaLoad (6.32.2) loads VBA projects. VbaMan (6.32.4) - manages VBA projects. VbaNew (6.32.5) - starts a new VBA project. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaUnload (6.32.9) - unloads VBA projects. VbaSecurity (6.32.8) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.

Figure 6.2544



6.31.16 VmlOut

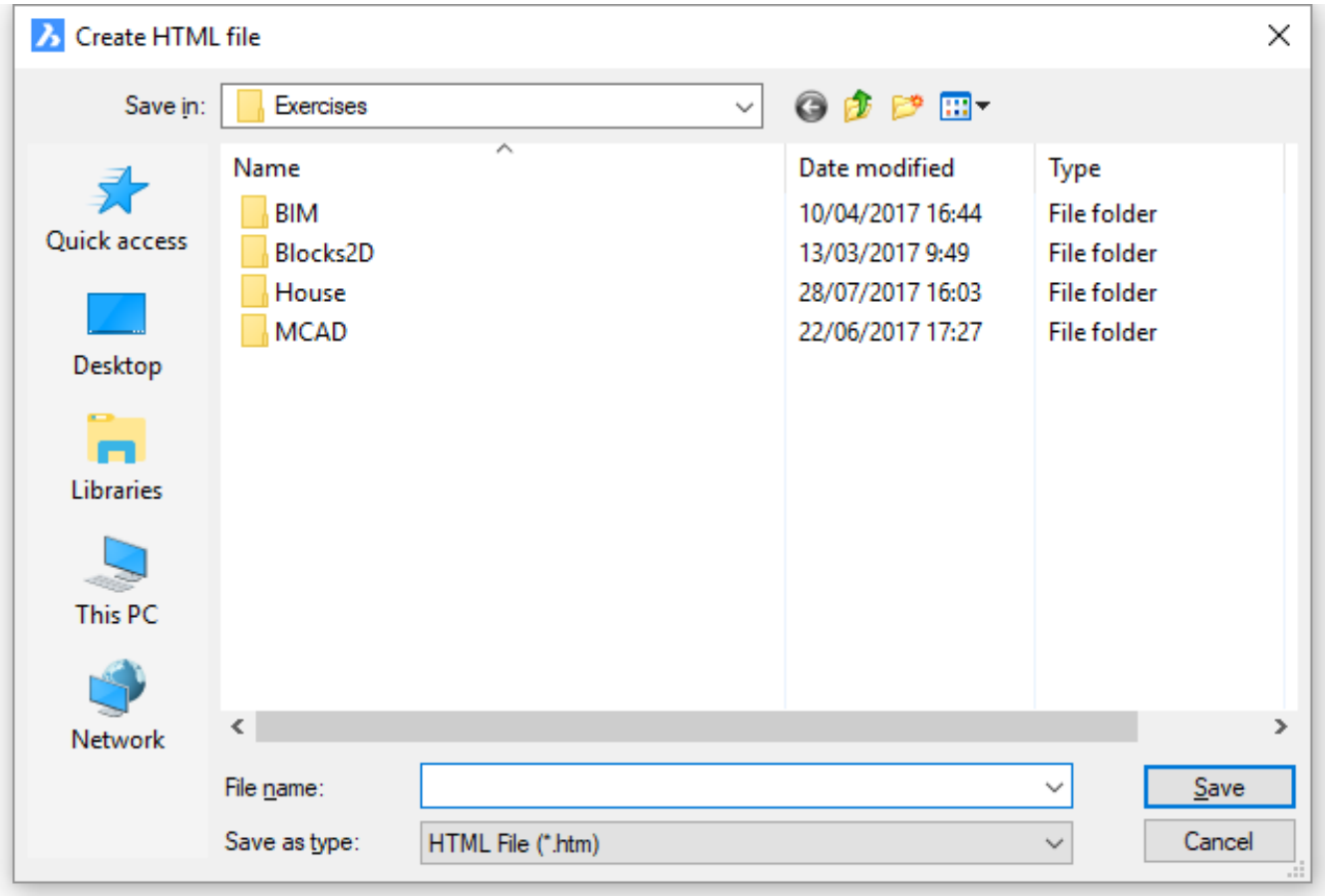
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Exports drawings in VML format, embedded in an HTML file (short for "vector markup language").

Accessing the Command

command bar: **vmlout** : **VmlOut** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18

1. Enter a file name.
2. Click **Save**.The VML data is saved in an HTML file.
3. To view the image, open the file in a Web browser; you may need to install a VML plug-in.

Figure 6.2545



Command Options

Option	Description
File name	Specifies the name of the file.
Save as type	Saves the VML data in an HTML file, short for "hyper text markup language".
Save	Saves the file. The output is in HTML format and looks something like this:

```
<html xmlns:v="urn:schemas-microsoft-com:vml"> <head>
```

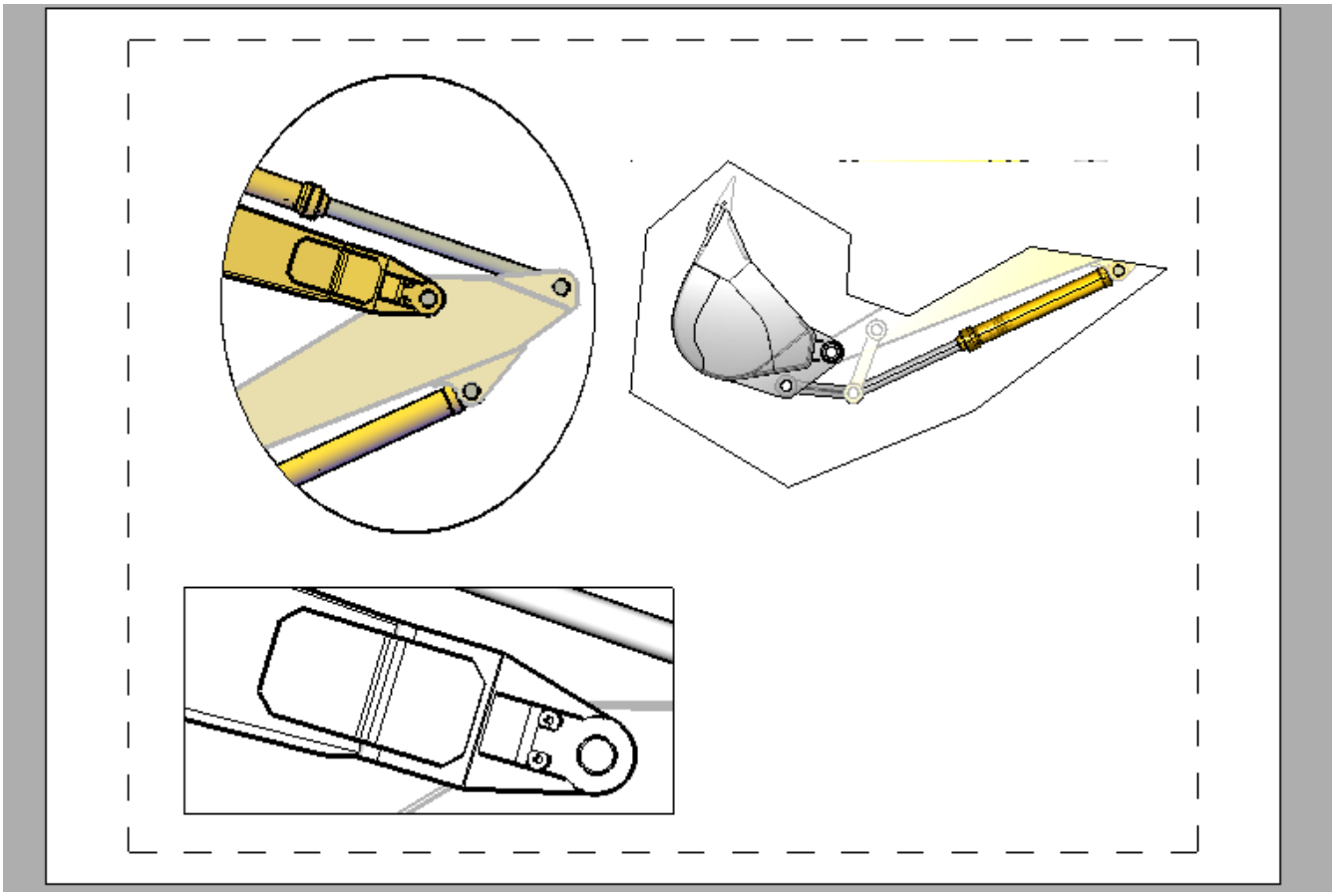
Related Commands

BmpOut (6.6.14) - exports drawings in BMP format. Export (6.13.15) - exports drawings in BMP, DWF, DXF, DWT, PDF, WMF, EMF, SLD, and older DWG formats. Import (6.17.10) - imports DXF files. MSlide (6.21.31) - exports drawings in WMF, EMF, and SLD formats. VSlide (6.31.22) - displays files in WMF, EMF, and SLD formats. WmfOut (6.33.12) - exports drawings in WMF, EMF, and SLD formats.

Figure 6.2546



Figure 6.2547



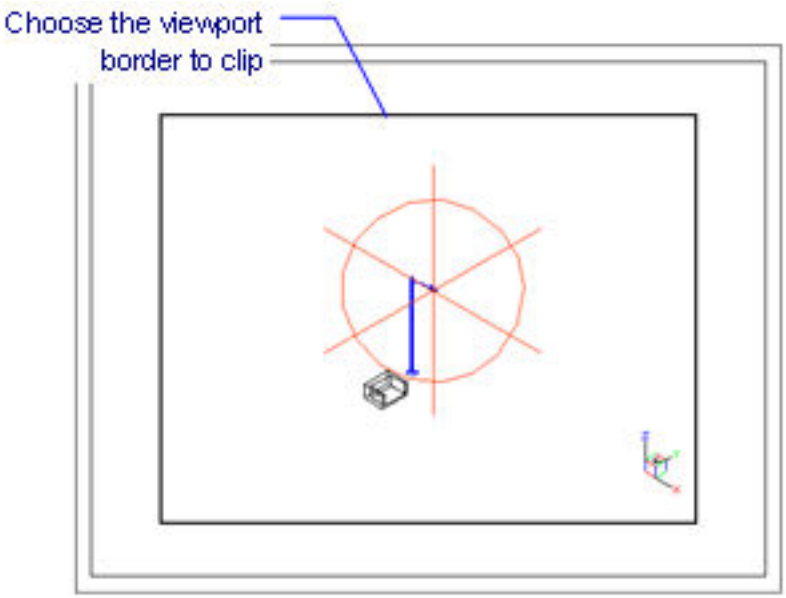
6.31.17 VpClip

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Makes viewports from entities or as polygons (short for "viewport clipping"); this command is useful for showing less of a drawing by unwanted clipping portions. **Note:** This command operates only in the paper space of a layout tab, and works only with existing viewports; use theMView (6.21.37) command to make additional viewports (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/vpclip-eg-1.png})

Accessing the Command

command bar: **vpclip : VpClip Prompts you in the command bar:** Select viewport to clip: **(Choose a viewport border)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image367.jpg}) Select clipping entity or [Polygonal]: **(Choose a closed entity, or press P)**

Figure 6.2548



Command Options

Option	Description
Select viewport to clip	Chooses the viewport to clip: in a layout tab: • Click on the border of a single viewport. • Clipped viewports can be reclipped. • Use the Delete option to remove clipping.
Select clipping entity	Converts a closed entity, such as a circle or closed polyline. into a viewport boundary.The entity must have be drawn in paper space. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Polygonal	Draws a polygonal viewport border made of straight and arc segments; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **Specify start point** - **(pick the first point of the polyline.)** 2. **Set next point or [draw Arcs/Distance]** - **(pick another point, or choose an option.)** 3. **Set next point or [draw Arcs/Distance/Follow/Undo]** - **(pick another, point or choose an option.)** 4. **Set next point or [draw Arcs/Close/Distance/Follow/Undo]** - **(enter an option:)** • Pick another point • Choose an option • Press **Enter** to close the polygon automatically Polygonal viewports must have at least three vertices.

Polygonal | draw Arcs	Draws an arc segment in the polygonal viewport; prompts you: **Set end of arc or [draw Lines/ Angle/ CEnter/ CLose/ Direction/ Radius/ Second point/Undo]** - **(Pick a point or enter an option;)** see theArc (6.4.12) command for the meaning of the options
Polygonal | Distance	Specifies the distance and angle for the next polygonal segment; prompts you: 1. **Distance for segment** - **(enter a distance, or show the distance by picking a point.)** 2. **Angle of segment** - **(enter an angle, or show the angle by picking a point.)**
Polygonal | Follow	Draws the next segment at the same angle as the previous polygonal segment; prompts you: **Length of line** - **(specifies the length of the segment.)**
Polygonal | Undo	Undoes the last polygonal segment.
Polygonal | Close	Closes the polygon, and then displays the model space drawing.
Polygonal | Delete	Deletes the clipping boundary, and returns the original rectangular viewport.(This option appears only when you select a clipped viewport.)

Related Commands

XClip (6.34.2) - clips externally-referenced drawings. ImageClip (6.17.7) - clips images. MView (6.21.37) - creates viewports in paper space. VPorts (6.31.20) - creates viewports in model space. MSpace (6.21.34) - switches to model space. PSpace (6.24.54) - switches to paper space.

Figure 6.2549

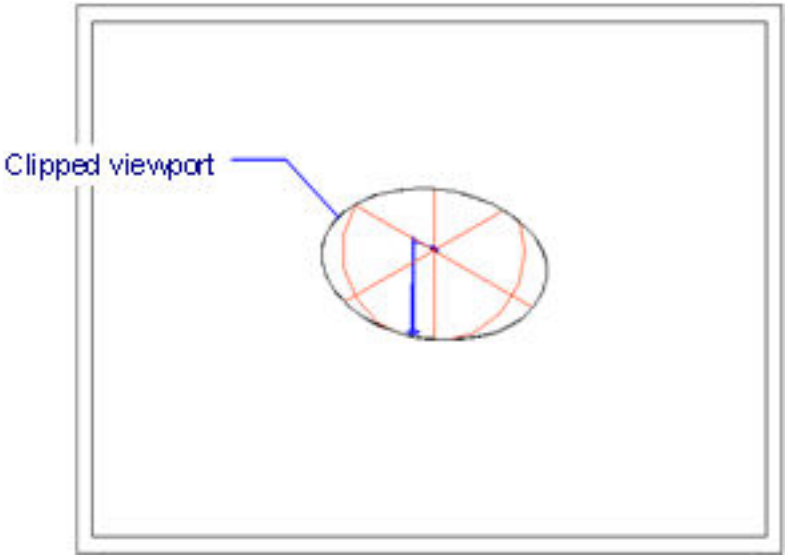


Figure 6.2550

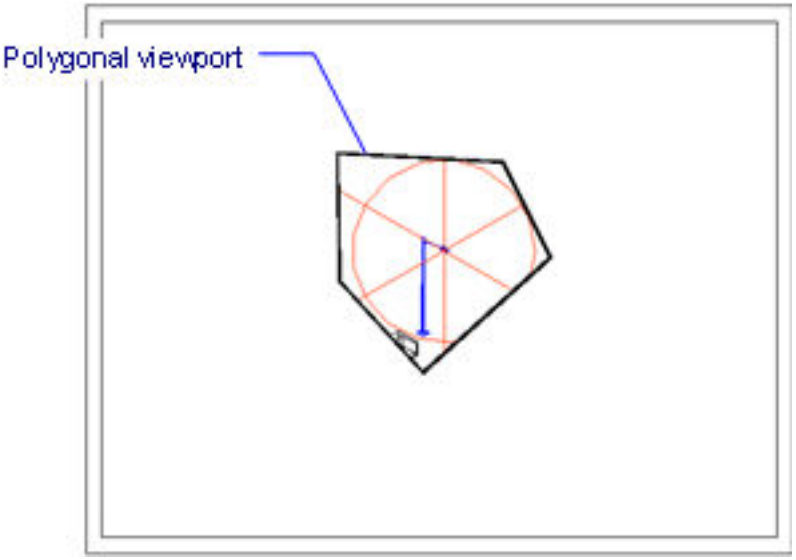


Figure 6.2551



6.31.18 VpLayer

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Changes the properties of layers in the current paper space viewport, at the command prompt (short for "view port layer"). This allows every viewport to show a different set of layers. **Note:**It is easier and faster, to use the Layer command to set viewport layer properties. This command does not operate in model space.

Accessing the Command

command bar: **vplayer** alias: **vl : VpLayer Prompts you in the command bar:** [List frozen layers/Color/LineType/LineWeight/TR layers/Thaw layers/Reset layers/New frozen layers/Default visibility setting]: **(Enter an option, or press Enter to leave the layer properties unchanged)** This command operates only in layout mode.In model space, this command responds, "This command can only be used in a Layout Tab ('Paper Space')."

Command Options

Option	Description
List frozen layers	Reports whether layers are frozen in the selected viewport; prompts you: **Select a viewpoint** - **(click a viewport border; responds with the following report:) The following layers are frozen in viewport 2: LayerName** When no layers are frozen, reports: **No layers are frozen in viewport.**
Color	Overrides the color of specific layers in selected viewports; prompts you: 1. **New truecolor: Red,Green,Blue** - **(type the R-G-B code for the new layer color, such as 123,213,12 for the levels of red, green, and blue.)** 2. **Select layer name(s) to override color** - **(type the layer name(s), separated by commas.)** 3. **Specify viewport(s) [All/Select/V18/eXcept current] <Current>** - **(choose an option:)** • **All** - apply color to all viewports in the current layout. • **Select** - prompts you to select the viewports: **Select a viewport**- **(click a viewport.) Entities in set: 1 Select a viewport**- **(click another viewport, or press Enterto end.)** • **Current**- applies to the color to current viewport • **Except current** - apply the color to all viewports, except the current one

Line Type

Overrides the linetype of specific layers in selected viewports; prompts you:

1. **Loaded linetype name or [? to list] <Continuous>** - **(choose an option:)**
   - **? to list** - displays a list of available linetypes in the command bar.
2. **Select layer name(s) to override linetype** - **(type the layer names separated by commas.)**
3. **Specify viewport(s) [All/Select/Current/eXcept current] <Current>** - **(choose an option:)**

- **All** - applies linetype all viewports in the current layout
- **Select** - prompts you to select the viewports: **Select a viewport**- **(click a viewport.) Entities in set: 1 Select a viewport**- **(click another viewport, or press Enterto end.)**
- **Current**- applies to the linetype to current viewport
- **Except current** - apply the linetype to all viewports, except the current one

Line Weight

Overrides the line weight of specific layers in selected viewports; prompts you

1. **Enter lineweight (0.00 mm - 2.11 mm)** - **(type a lineweight.)**
2. **Select layer name(s) to override lineweight** - **(type the layer names separated by commas.)**
3. **Specify viewport(s) [All/Select/Current/eXcept current] <Current>** - **(choose an option:)**

- **All** - applies the line weight to all viewports in the current layout
- **Select** - prompts you to select the viewports: **Select a viewport**- **(click a viewport.) Entities in set: 1 Select a viewport**- **(click another viewport, or press Enterto end.)**
- **Current**- applies to the line weight to current viewport
- **Except current** - apply the line weight to all viewports, except the current one

Transparency	Overrides the transparency of specific layers in selected viewports; prompts you 1. **Enter transparency (0 - 90**) - **(type a value.)** 2. **Select layer name(s) to override transparency** - **(type the layer names separated by commas.)** 3. **Specify viewport(s) [All/Select/Current/eXcept current] <Current>** - **(choose an option:)** • **All** - applies the transparency to all viewports in the current layout • **Select** - prompts you to select the viewports: **Select a viewport**- **(click a viewport.) Entities in set: 1 Select a viewport**- **(click another viewport, or press Enterto end.)** • **Current**- applies to the transparency to current viewport • **Except current** - apply the transparency to all viewports, except the current one
Freeze layers	Freezes specific layers in the current viewport; prompts you: **Select the layer(s) to freeze** - **(enter the name(s) of layers to freeze, separating multiple layer names with commas.)**
Thaw layers	Thawsspecific layers in the current viewport; prompts you: **Select the layer(s) to reset** - **(enter the names of layer(s) to thaw, separating multiple layer names with commas.)**
Reset layers	Resets frozen and thawed layers to their original settings; prompts you: **Select the layer(s) to reset** - **(enter the names of layers to reset, separating multiple layer names with commas.)**
New frozen layers	Creates new layers that are initially frozen (invisible) when a new viewport is created; prompts you: **New frozen layer name(s)** - **(enter the names of layers to create and freeze, separating multiple layer names with commas.)**
Default visibility settings	Changes the default freeze/thaw setting for layers; prompts you: 1. **Layer name(s) -(enter one or more layer names; separate multiple layer names with commas.)** 2. **Setting Frozen/<Thawed>** - **(enter an option:)** • **Frozen** - changes the layer's default property to frozen • **Thawed** - changes the layer's default property to thawed

Related Commands

-Layer (6.20.3) - creates layers and changes their properties in the command bar. Layer (6.20.2) - creates layers, changes their properties, and removes layers through the Drawing Explorer. LayersPanelOpen (6.20.6) - opens the Layers panel, which allows to edit the layers in the current drawing, apply layer states and layer filters.

Figure 6.2552



6.31.19 VPoint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Changes the 3D viewpoint. **Note:** It is faster, and easier, to use the LookFrom (6.20.39) widget to change the 3D viewpoint.

Accessing the Command

command bar: **vpoint** aliases: **viewpoint -vp -viewpoint -vpoint : VPoint Prompts you in the command bar:** Set view point or [Rotate/PLan view/PErspective] <0, 0, 1>: **(Enter x,y,z coordinates, an option, or else press Enter for a dialog box)**

Command Options

Option	Description
Set view point	Specifies the viewpoint: enter x,y or x,y,z coordinates such as -1,-1,1 for the front-left isometric viewpoint
Rotate	Changes the viewpoint by specifying angles; prompts you: 1. **Angle in XY plane from X axis <45>** - **(specify an angle counter-clockwise from the x ais, or pick two points.)** 2. **Angle from the XY plane <60>** - **(specify an angle up or down from the x,y plane, or pick two points.)**
Plan view	Displays the plan view of the current UCS.See the Plan (6.24.25) command.
Perspective	Set the Perspective property for the viewport; prompts you: **Perspective [perspective ON/perspective OFf] <Perspective off>** - **(choose an option:)** • ON -turns on perspective viewing mode, so that objects further away look smaller • **OFf** - turns off perspective mode, returning to orthogonal viewing mode

Dialog Box Options

See theDdVpoint (6.10.22) command.

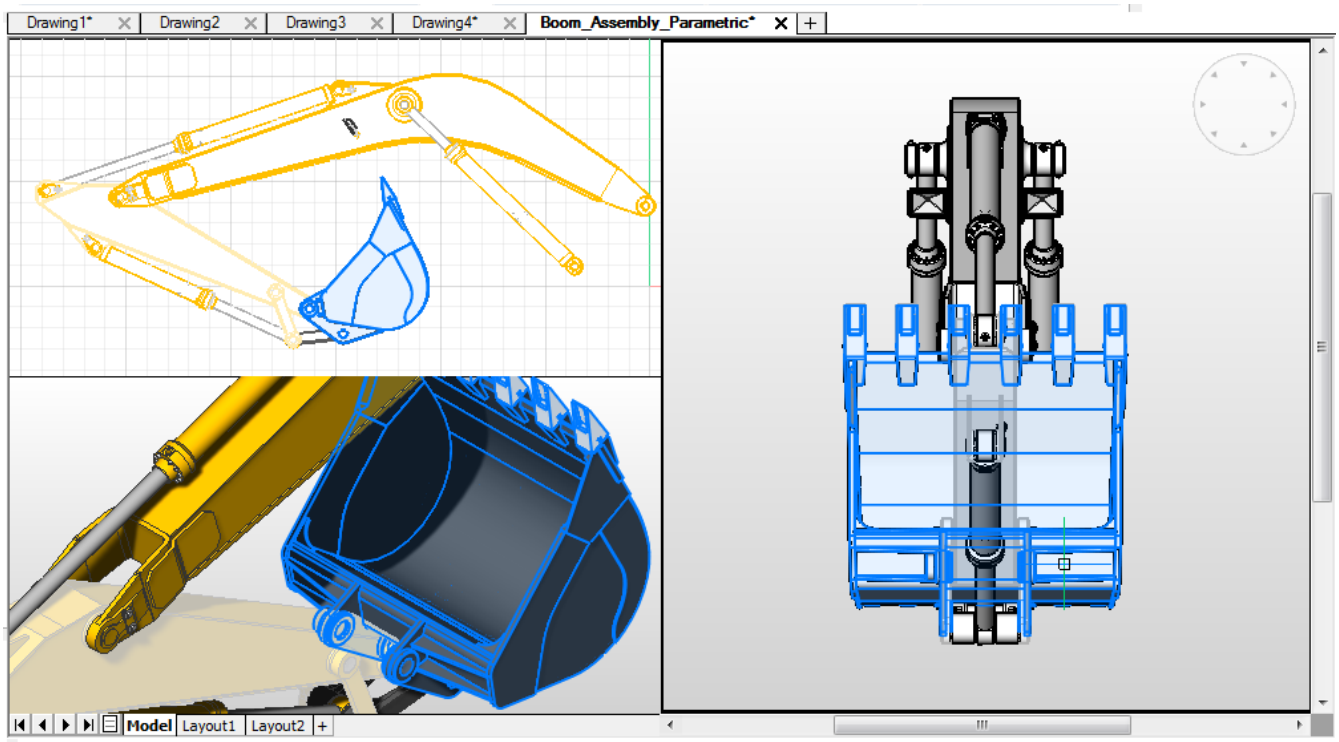
Related Commands

DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. DView (6.10.45) - changes the 3D viewpoint interactively, and turns on perspective mode. RtRot (6.26.38) - rotates 3D drawings in real-time

Figure 6.2553



Figure 6.2554



6.31.20 VPorts

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates one or more viewports in model space (short for "viewports").Viewports allow you to see more than view of the same drawing. **Note:** To create viewports in paper space, use theMView (6.21.37) command. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/vports-eg-1.png})

Accessing the Command

command bar: **vports** transparent:**'-vports** aliases:**viewports vport vw -vports** menu bar: **View | Viewports** ribbon:**Home | Settings Layouts | Viewports View | View | Viewports** toolbar: **View | Layout | : VPorts Prompts you in the command bar:** Viewports [? to list/Save/Restore/Delete/Join/SIngle/create 2 viewports/create 3 viewports/create 4 viewports] <Create 3 viewports>: **(Enter an option or press Enter to create 3 viewports.)** The current (active) viewport is shown by a heavy rectangle. Double-click a viewport to make it the active one. You can repeat the command to create additional viewports inside viewports to a maximum of 64. Properties like grid, snap, UCS icon, and visual style are set independently in each viewport.

Command Options

Option	Description

? to list

Lists the names and x,y coordinates of saved viewports; atypical display looks like this:

```
Viewport Name   LL Corner      UR Corner --------------
```

- **LL Corner** - lower-left corner
- **UR Corner** - upper-right corner.

Press **F2** to view the list in the prompt history window.

Save

Saves the current viewport arrangement by name; prompts you: **Viewport configuration to save** - **(specifies a name by which to save the viewport configuration)**; whenthe name already exists, prompts you: **A viewport configuration with that name exists, Overwrite?(Yes/No)?** - **(enter an option)**:

- **Yes** - replaces the viewport configuration name with the new one
- **No** - repeats the prompt, 'Viewport configuration to save'; enter a different name

Restore

Restores a named viewport configuration; prompts you: **Viewport configuration to restore** - **(enter the name of a save viewport configuration to restore.)**

Delete

Erases a named viewport configuration from the drawing; prompts you: **Viewport configuration to delete** - **(specify the name of a viewport configuration.)** You cannot delete more than one viewport configuration at a time.

SIngle

Creates a single viewport of the current viewport, removing all other viewports. (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/
Use this option to return the drawing to its original, single-viewport state.

Join

Joins two or more viewports into a single viewport; prompts you:

1. **Select dominant viewport <Current>** - **(choose the viewport that will be retained, or press Enter to select the current viewport)**
2. **Select viewport to join** - **(choose the viewport that will unified with the first viewport.)**

If the two viewports were to join to form a non-rectangular shape, such as an L or a T shape, BricsCAD complains, "The selected viewports do not form a rectangle." The 'Select viewport to join' prompt repeats for you to try again.

2	Spits the current viewport into two rectangular viewports; prompts you: **Two viewports [Horizontal/Vertical] <Vertical>** - **(type H or V.)** • **Horizontal** - creates two horizontal viewports, one above the other: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Vertical** - creates two vertical viewports, one beside the other: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
3	Spits the current viewport into three rectangular viewports; prompts you: **Three viewports [Horizontal/Vertical/Above/Below/Left/Right] <Right>** - **(enter an option.)** • **Horizontal** - creates three horizontal viewports: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Vertical** - creates three vertical viewports: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Above** - creates a single wide viewport above two side-by-side viewports: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Below** - creates two side-by-side viewports above one wide viewport: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Left** - creates one tall viewport to the left of two stacked viewports: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Right** - creates one tall viewport to the right of two stacked viewports: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
4	Spits the current viewport into four rectangular viewports: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i This style is sometimes called ”engineering” viewports, as it allows you to see the top, front, side, and isometric views at the same time -- once you adjust the viewpoint in each viewport, which can be done with the MvSetup (6.21.39) command.

Procedures

- Creating viewports in model space (1.16.13)
- Drawing in multiple viewports (1.16.13)

- Joining adjacent viewports (1.16.13)

Related Commands

VpClip (6.31.17) - clips viewports. MView (6.21.37) - creates viewports in paper space. MvSetup (6.21.39)- creates four engineering viewports MSpace (6.21.34) - switches to model space. PSpace (6.24.54) - switches to paper space. RedrawAll (6.26.10) - redraws all viewports. RegenAll (6.26.16) - regenerates the drawing in all viewports. Zoom (6.35.7) - utilizes the XP option to specify the zoom factor in paper space relative to model space.

Figure 6.2555



Figure 6.2556



Figure 6.2557

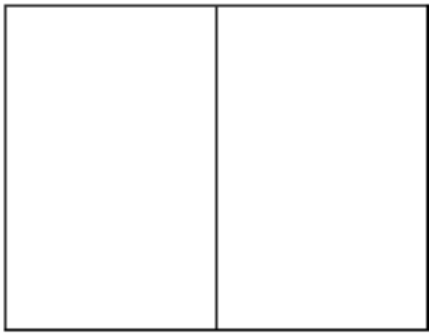


Figure 6.2558



Figure 6.2559

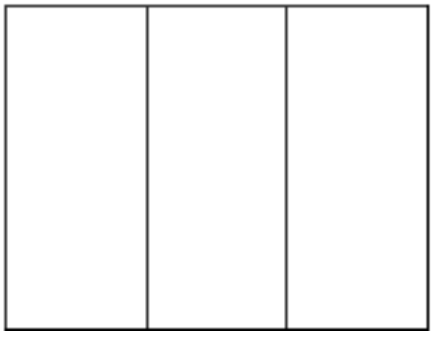


Figure 6.2560



Figure 6.2561



Figure 6.2562

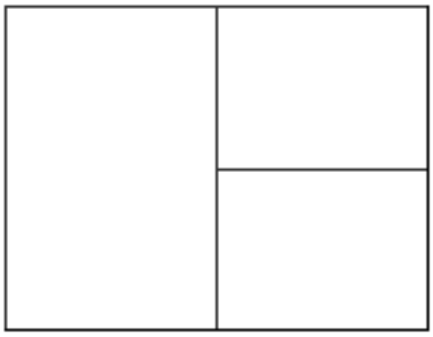


Figure 6.2563

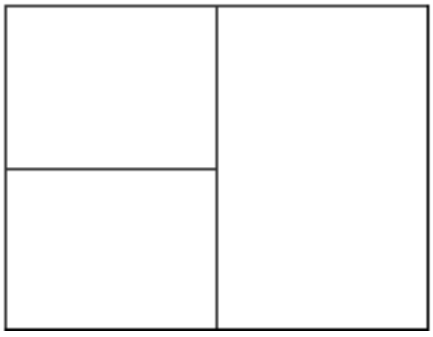


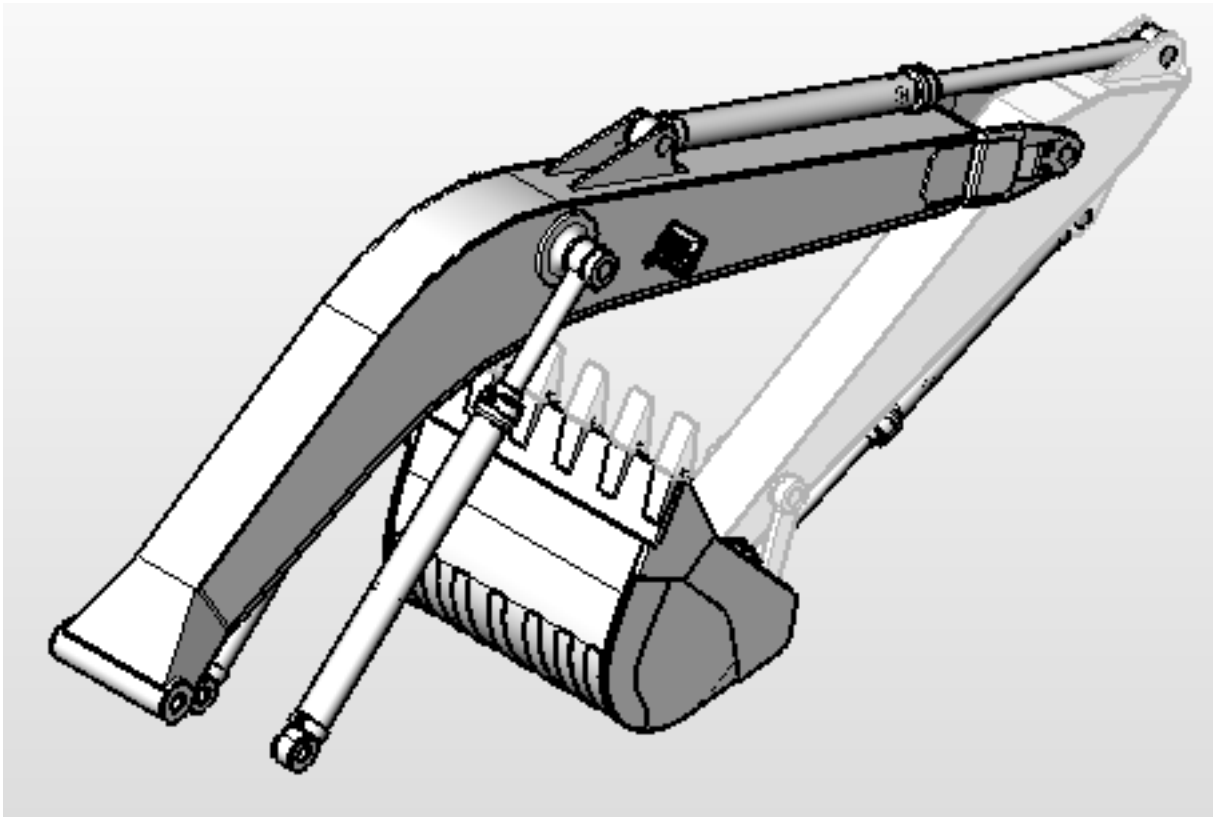
Figure 6.2564



Figure 6.2565



Figure 6.2566



6.31.21 VsCurrent

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the visual style for the current viewport, in the command line. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/vscurrent-eg.png})

Accessing the Command

command bar: **vscurrent** : **vscurrent Prompts you in the command bar:** Enter visual style [2dwireframe/Wireframe/Hidden/Rea with Edges/shades of Grey/SKetchy/X-ray/Other/cUrrent] <Modeling>: **(Enter an option)**

Command Options

Option	Description
2dwireframe	Specifies the visual style for 2D drawings.
cUrrent	Specifies the current visual style; press **Enter** to keep it.
Wireframe, Hidden, Realistic, Conceptual, Shaded, shaded with Edges, shades of Grey, SKetchy, **and** X-ray	Specifies the visual style to apply to the current viewport for 3D models: • Enter the name of a style • Press **Enter** to keep the default style The styles named in this listare the ones included with the program.

Other	Choose a user-defined visual style that was created with the VisualStyles command; prompts you: **Enter a visual style name or [?]** - **(enter an option:)** • Type the name of a visual style • Press **?** to see a list all available visual styles; perss**F2** to open the Prompt History window: `2dWireframe Conceptual Hidden Modeling Realistic`

Related Commands

-VisualStyles (6.31.14) - manages visual style definitions at the command line. VisualStyles (6.31.13) - creates and controls visual style definitions in the Drawing Explorer.

Figure 6.2567



6.31.22 VSlide

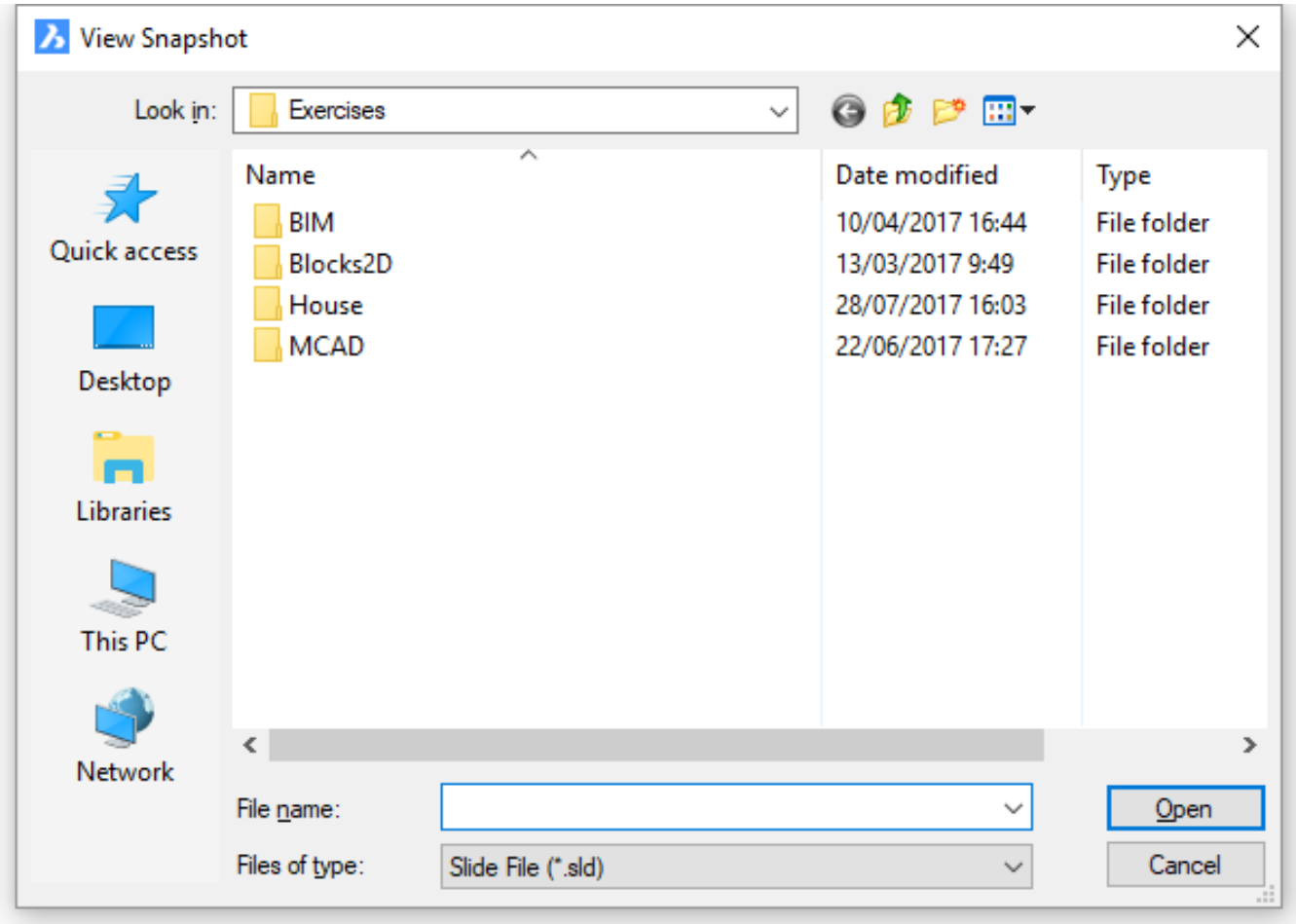
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays images saved as SLD and WMF files (short for "view slide"). **Note:** WMF files are not supported by the Linux platform.

Accessing the Command

 command bar: **vslide** aliases: **vs vsnapshot** menu bar: **Tools | View Snapshot** ribbon:**Tools | Action Recorder | View Snapshot** toolbar: **Tools |**  **: VSlide** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsys

1. Choose a file.
2. Click **Open**.
3. To remove the slide from the viewport, enter the Redraw (6.26.9) command.

Figure 6.2568



Command Options

Option	Description
File name	Specifies the name of the slide file.
Files of type	Selects the file type of the slide file: • Slide File (*.sld) - proprietary graphics format created by Autodesk for saving screen grabs of viewports • Slide Library (*.slb) - collection of slide files used to display slideshows of CAD designs • Enhanced Metafile (*.emf) - enhanced WMF format • Windows Metafile (*.wmf) - hybrid raster-vector format developed by Microsoft
Open	Displays the slide in the current viewport. Enter the Redraw (6.26.9) command to remove the slide.

Related Commands

MSlide (6.21.31) - saves drawings in SLD, EMF, and WMF formats. Redraw (6.26.9) - removes the slide from the viewport.

6.32 VBA

6.32.1 Vbalde

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) Command only available on the Windows platform Opens the Microsoft Visual Basic editing window for writing and debugging VBA code (short for ”visual basic for applications integrated development environment”).

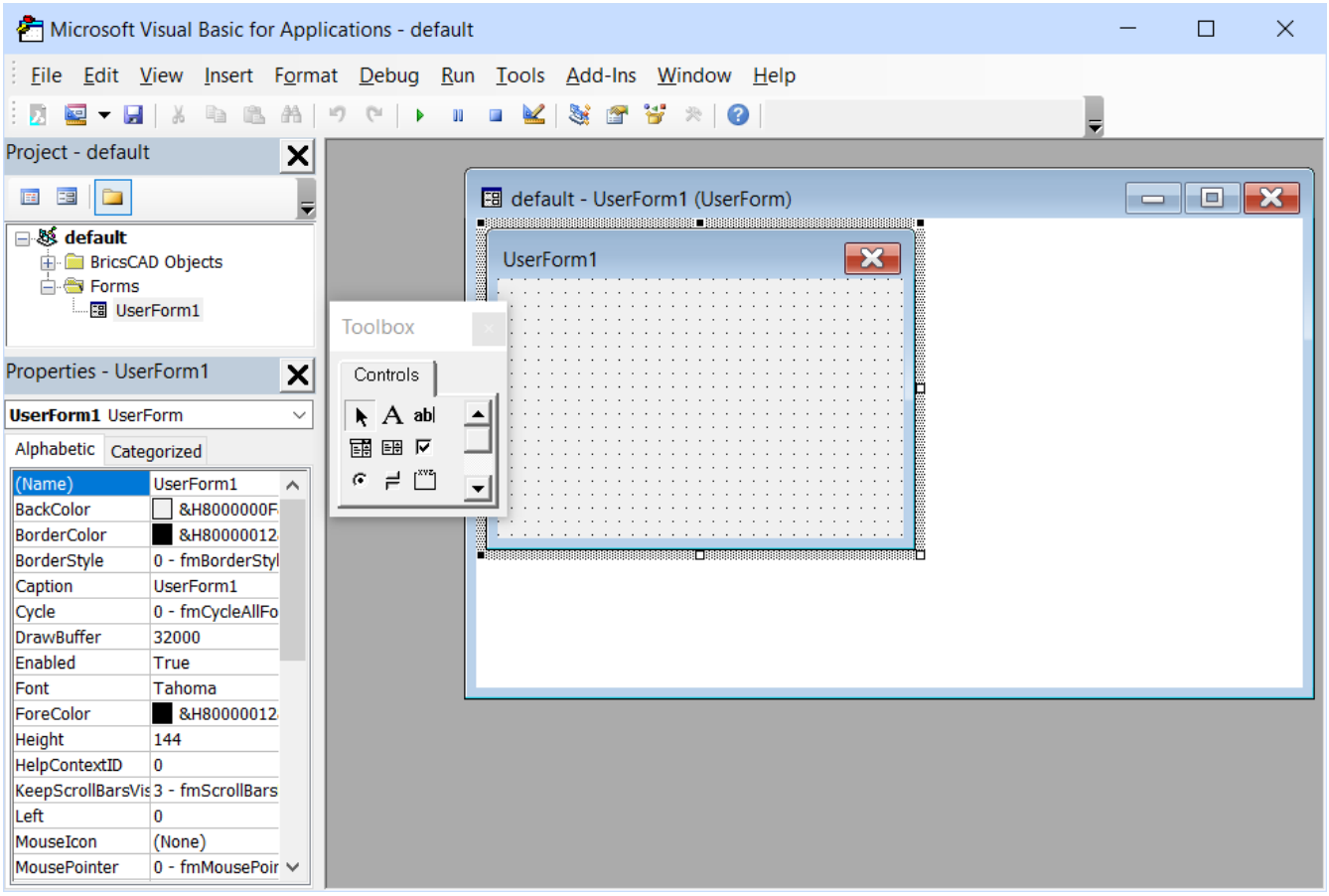
Figure 6.2569



Accessing the Command

 command bar: **vbaide** shortcut: **Alt+F11** alias: **vba** menu bar: **Tools | VBA | Visual Basic for Applications** ribbon: **Tools | Applications | Visual Basic for Applications** toolbar: **Tools |**  **: Vbalde** Displays a window: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/vbaide-dlg.png})

Figure 6.2570



Command Options

See the help files provided by Microsoft for Visual Basic.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. VbaMan (6.32.4) - manages VBA projects. VbaNew (6.32.5) - starts a new VBA project. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaSecurity (6.32.8) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.2571



6.32.2 VbaLoad

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Loads VBA projects through a dialog box (short for ”visual basic for applications load”).

Accessing the Command

command bar: **vbaload** menu bar: **Tools | VBA | Load Project : VbaLoad** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/vbaload-dlg.png})

1. Choose a DVB or VBI file.
2. Click **Open**. BricsCAD warns about the potential hazard of macros: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscor
3. Click **Enable Macros**; see the Security (**??**) command for details.
4. To run the macros in the project, use the VbaRun (6.32.6) command.

Figure 6.2572

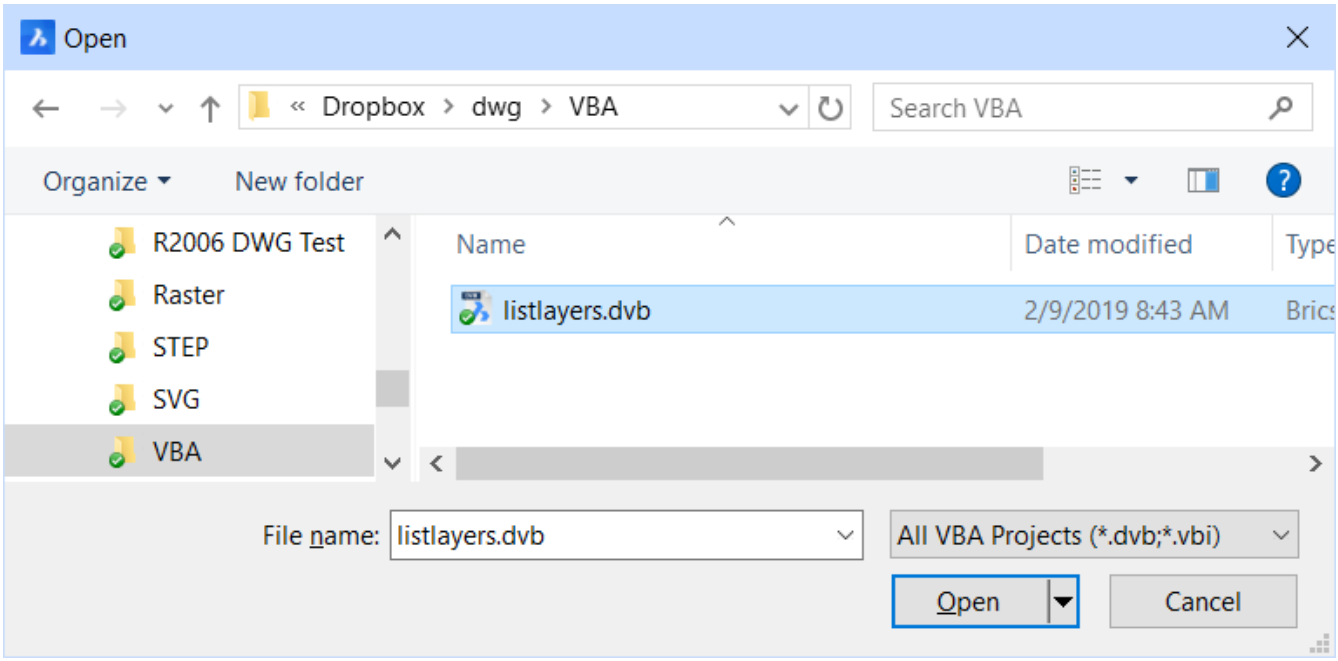
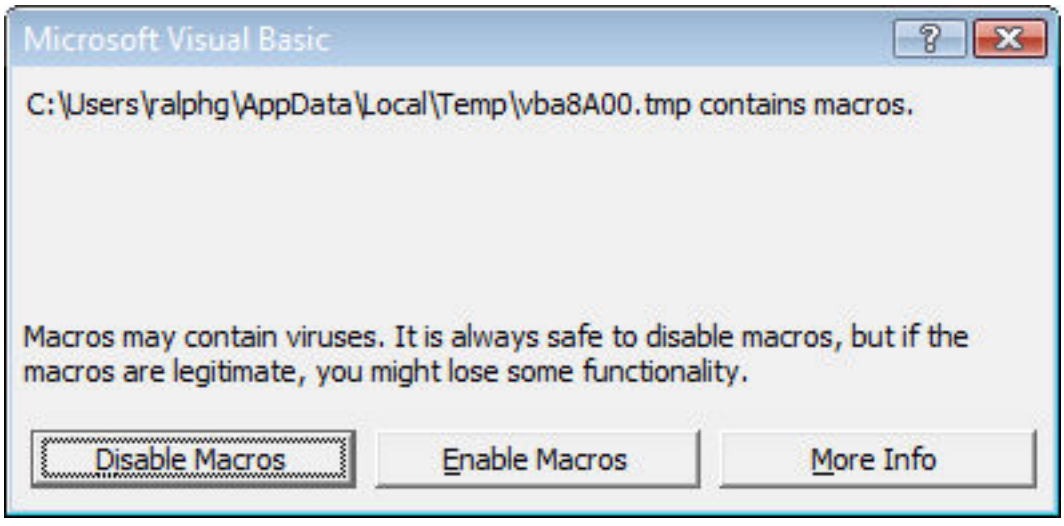


Figure 6.2573



Command Options

Option	Description
File name	Specifies the name of the DVB file.DVB files hold VBA projects.
Open	Opens the DVB file.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. -VbaLoad (6.32.3) - loads VBA projects. VbaMan (6.32.4) - manages VBA projects. VbaNew (6.32.5) - starts a new VBA project. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaSecurity (6.32.8) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.2574



6.32.3 -VbaLoad

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Loads VBA projects in the command bar (short for "visual basic for applications load").

Accessing the Command

command bar: **-vbaload : -VbaLoad Prompts you in the command bar:** VBA Project Name: **(Enter the path and name of a DVB or VBI file)** When you specify the name of an already-loaded DVB or VBI file, BricsCAD reports, "Project is already loaded." This command will not load VBA projects that are embedded in drawings, because opening a drawing loads its associated project automatically.

Command Options

Option	Description
VBA Project Name	Names the DVB or VBI file.DVB and VBI are VBA project files, which contain one or more VBA macros.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. VbaLoad (6.32.2) loads VBA projects. VbaMan (6.32.4) - manages VBA projects. VbaNew (6.32.5) - starts a new VBA project. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaSecurity (6.32.8) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.2575



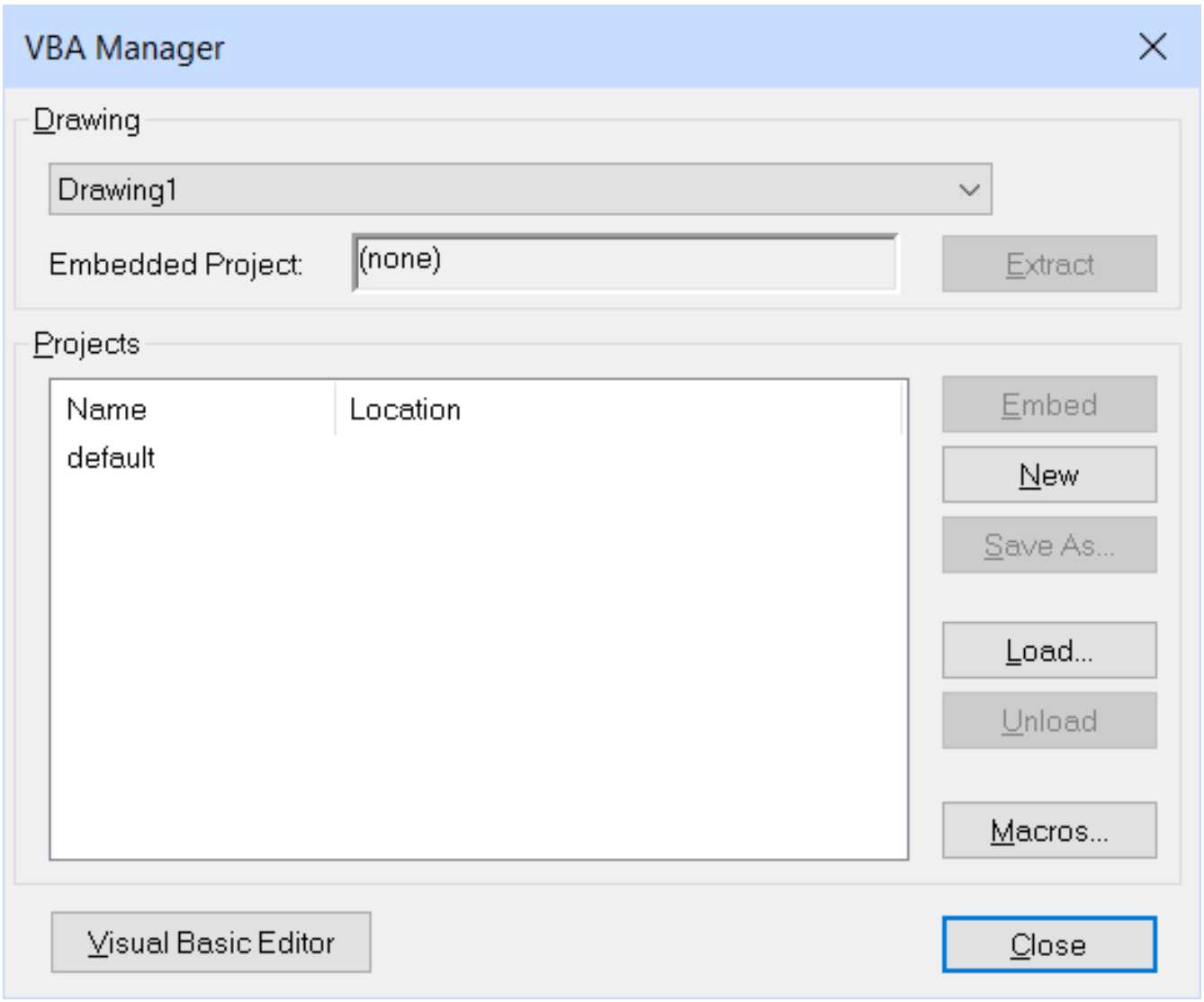
6.32.4 VbaMan

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Manages VBA projects (short for ”visual basic for applications manager”).

Accessing the Command

command bar: **vbaman** shortcut:**Shift+F6** menu bar: **Tools | VBA | Project Manager : VbaMan** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/vbaman-dlg.png}) Choose options, and then click Close.

Figure 6.2576



Command Options

Option	Description
Drawing	Lists the names of drawings currently open in BricsCAD: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Choose a drawing name from the droplist; the embedded project is listed in the Embedded Project text box, if any.
Embedded Project Name	Lists the name of the embedded project, if any. Lists the names of projects loaded into BricsCAD at this time.
Location	Lists the drive and folder names of DVB and VBI files.
Extract	Removes the project from the current drawing; displays dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Yes** - displays the Save As dialog box: 1. Name the project. 2. Click **Save**. • **No** - removes the project from the drawing without saving it as a DVB file. • **Cancel** - cancels the command; the project remains in the drawing.
Embed	Embeds a project in the current drawing. If the project is a new one, and has not yet been saved, this option displays the Save As dialog box. Each drawing can contain just one embedded project; after one project is embedded, the Embed button becomes unavailable. To change embedded projects, use the Extract option to remove the current project.
New	Creates a new VBA project; gives the generic name VbaProject . Use the Save As option to rename and save the DVB or VBI project file.
Save As	Saves the selected project by another name. Displays the Save As dialog box: 1. Enter a file name. 2. Click **Save**.
Load	Loads DVB and VBI project files; displays the Open dialog box: 1. Choose a DVB or VBI file. 2. Click **Open**. See theVbaLoad (6.32.2) command.
Unload	Unloads the selected project without a warning.
Macros	Displays the Run BricsCAD VBA Macro dialog box: 1. Choose the macro to run. 2. Click **Run**. See theVbaRun (6.32.6) command.
Visual Basic Editor	Opens the VBA integrated development environment. See theVbaIde (6.32.1) command.
Close	Closes the dialog box.

Figure 6.2577

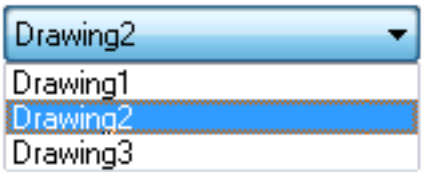
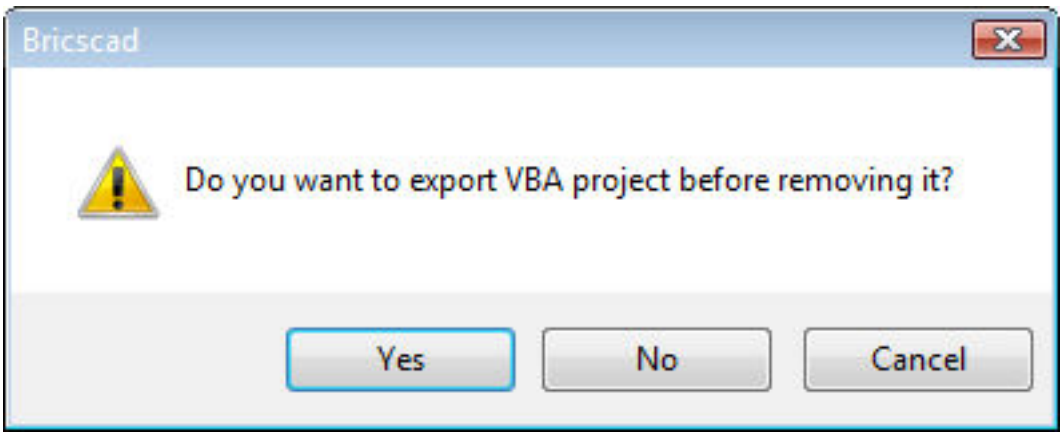


Figure 6.2578



Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. VbaNew (6.32.5) - starts a new VBA project. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaSecurity (6.32.8) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.2579



6.32.5 VbaNew

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Starts a new VBA project(short for "visual basic for applications new").

Accessing the Command

command bar: **vbanew** menu bar: **Tools | VBA | New Project : VbaNew**

Command Options

No options.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. vbaide (6.32.1) - opens the VBA editing window. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaMan (6.32.4) - manages VBA projects. VbaSecurity (6.32.8) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.2580



6.32.6 VbaRun

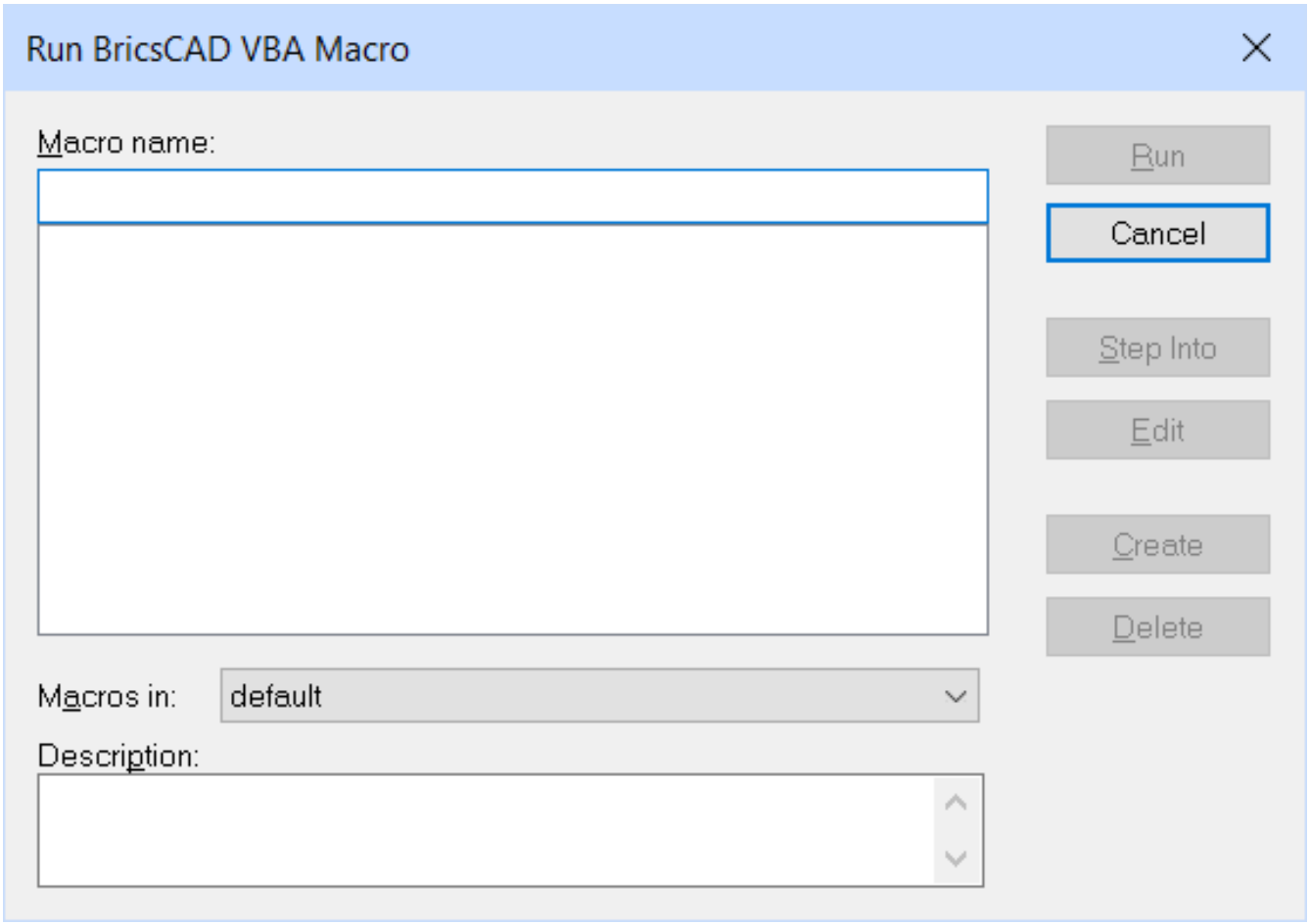
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Runs, creates, edits, and deletes VBA macros through a dialog box (short for ”visual basic for applications run”).

Accessing the Command

command bar: **vbarun** shortcut:**Alt+F8** menu bar: **Tools | VBA | Macros : VbaRun** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/vbarun-dlg.png})

1. Choose the name of a macro.
2. Click **Run**.

Figure 6.2581



Command Options

Option	Description
Macro name	Lists the names of macros found in the project file. Choose one to run.
Macros in	Lists the names of projects loaded into the current drawing. Choose one from the droplist.
Description	Describes the macro. You can edit this text.
Run	Runs the selected macro.
Cancel	Closes the dialog box.
Step Into	Opens the integrated development environment window. The macro is in debug mode, which runs one line of code at a time. See theVbaIde (6.32.1) command.
Edit	Opens the macro in the integrated development environment window. The macro is in editing mode, which allows you to change the code. See theVbaIde (6.32.1) command.
Create	Creates new macros; follow these steps: 1. In the **Macro Name** field, enter a name for the new macro 2. Click **Create**. 3. Notice the integrated development environment window. Enter VBA code. See theVbaIde (6.32.1) command.

Delete	Deletes the macro; prompts you: **Are you sure you want to delete the macro -(enter Y or N.)** • **Yes** - erases the macro • **No** - does not erase the macro

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. VbaLoad (6.32.2) and -VbaLoad (6.32.3) - loads VBA projects. VbaMan (6.32.4) - manages VBA projects. VbaNew (6.32.5) - starts a new VBA project. VbaRun (6.32.6) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaSecurity (6.32.8) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.2582



6.32.7 -VbaRun

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Runs VBA macros from the command bar (short for "visual basic for applications run"). **Note:**This command is meant for use by macros and programs.

Accessing the Command

command bar: **-vbarun : -VbaRun Prompts you in the command bar:** Macro name: **(Enter the name of a VBA macro)**

- If the macro is part of a project and a module, enter its name in this format: **project.module.macro**
- If the DVB file is not loaded into BricsCAD, enter the name of the macro in this format: **project.dvb!project.module.macro**

Command Options

Option	Description
Macro name	Names the VBA macro to be run.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. VbaLoad (6.32.2) and -VbaLoad (6.32.3) - loads VBA projects. VbaMan (6.32.4) - manages VBA projects. VbaNew (6.32.5) - starts a new VBA project. VbaRun (6.32.6) - runs VBA macros. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaSecurity (6.32.8) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.2583



6.32.8 VbaSecurity

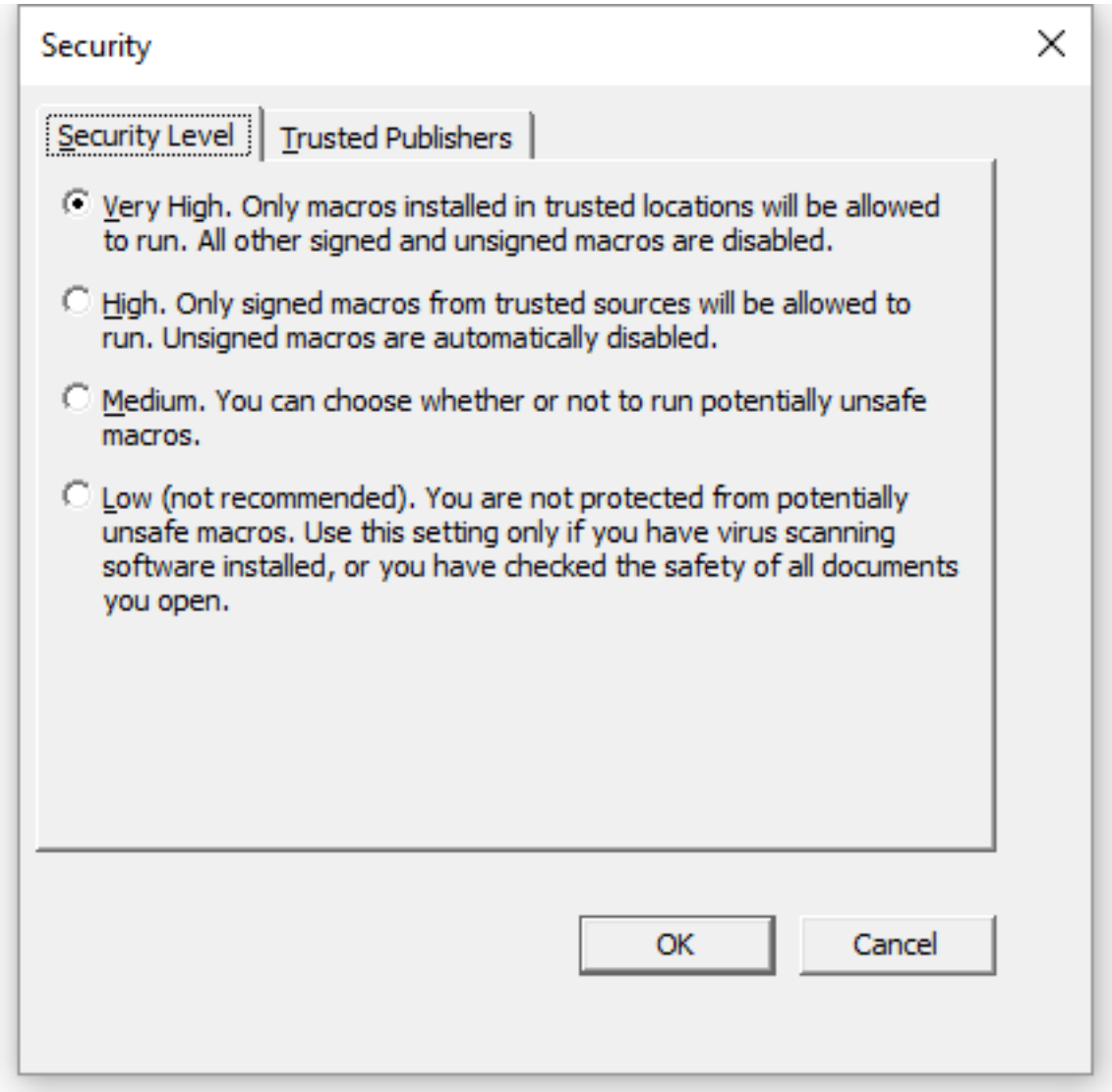
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Sets the security level to Low, Medium, or High to determine whether VBA macros are allowed to run automatically. **Note:**This command helps protect your computer, because Microsoft's design of VBA applications allows them to run destructive code, which can damage files on your computer.

Accessing the Command

command bar: **vbasecurity** alias: **security** menu bar: **Tools | VBA | Security : VbaSecurity** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/vbasecurity-dlg.png})

1. Choose a level of security
2. Click **OK**.

Figure 6.2584



Command Options

Option	Description
Very High	Only VBA macros installed in trusted locations will be allowed to run. All other signed or unsigned macros are disabled.
High	Only signed macros from trusted sources will be allowed to run. Unsigned macros are automatically disabled.
Medium	BricsCAD displays a dialog box when a VBA macro runs. The dialog box allows you to choose whether or not to run the macro.
Low	All VBA macros are run.

Trusted Sources Option	Description
Trusted Publishers	Lists the names of VBA macros that can be trusted. These are the only macros that will run when security is set to High. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/V/vbtrustedpublishers-dlg.png})
Remove	Removes the selected VBA macro from the list.
Trust access to Visual Basic Project	Allows access to the VBA Project.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaMan (6.32.4) - manages VBA projects. VbaNew (6.32.5) - starts a new VBA project. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaUnload (6.32.9) - unloads VBA projects.

Figure 6.2585

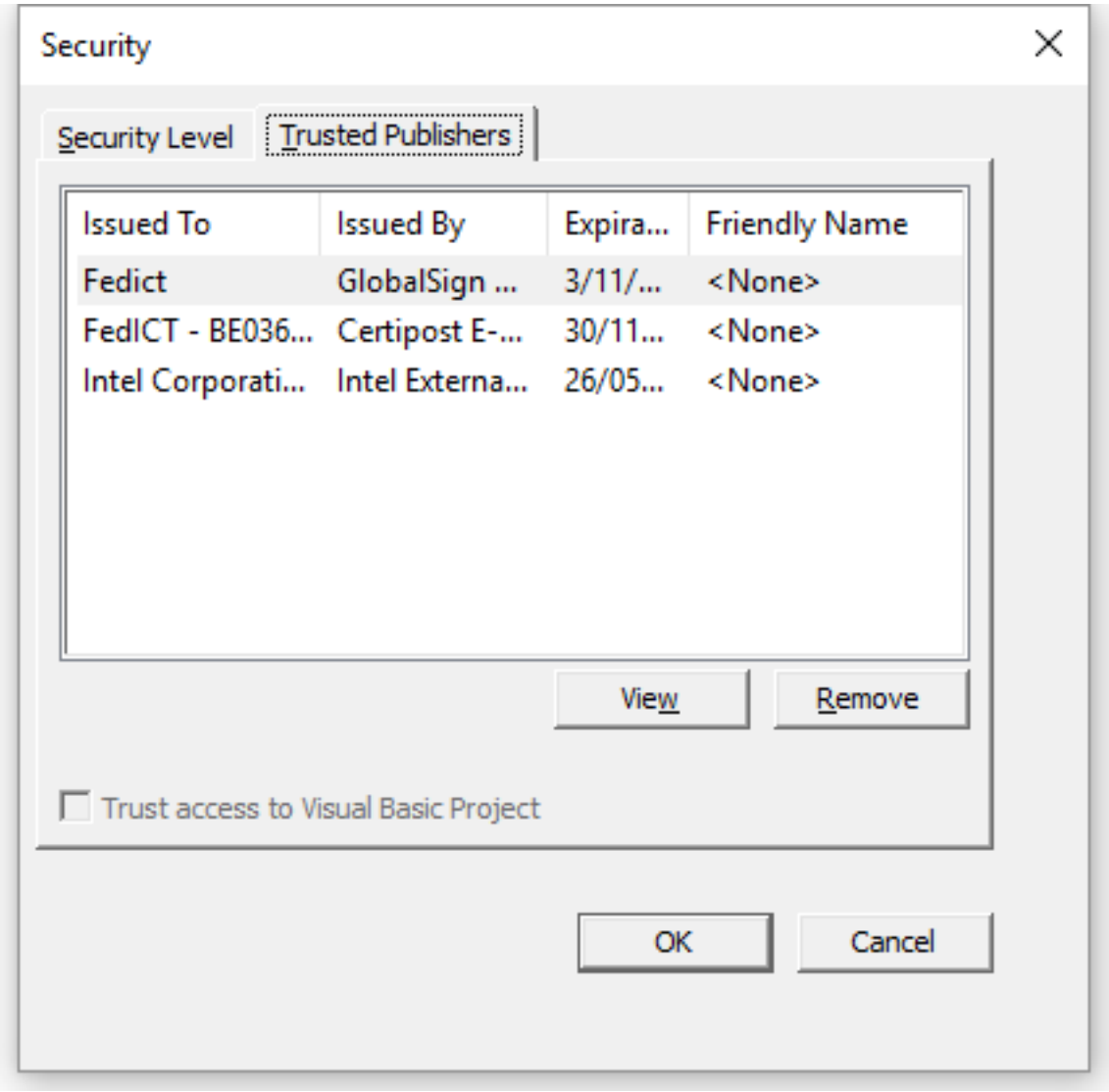


Figure 6.2586



6.32.9 VbaUnload

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Unloads VBA projects (short for "visual basic for applications unload"). **Note:** This command is not available in the 64-bit version of the program.

Accessing the Command

command bar: **vbaunload : VbaUnload Prompts you in the command bar:** VBA Project Name: **(Enter the name of a DVB or VBI project, or press Enter)**

Command Options

Option	Description
VBA Project Name	Names the DVB or VBI file to unload.To unload the current project, press Enter at the 'VBA Project Name" prompt. DVA and VBI files are VBA project files, that contain one or more VBA macros.

Related Commands

AddInMan (6.4.4) - loads and unloads VBA functions through a dialog box. AppLoad (6.4.14) - selects external routines to load when BricsCAD starts. Netload (6.22.3) - loads a .NET application. VbaIde (6.32.1) - opens the VBA integrated development environment. VbaMan (6.32.4) - manages VBA projects. VbaNew (6.32.5) - starts a new VBA project. VbaRun (6.32.6) and -VbaRun (6.32.7) - runs VBA macros. VbaSecurity (6.32.8) - depending on the selected security level (Low, Medium or High) VBA macros are allowed to run automatically or not.

6.33 W

6.33.1 WBlock

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Saves blocks and other drawing entities to DWG/DXF files, through a dialog box (short for ”write block”).

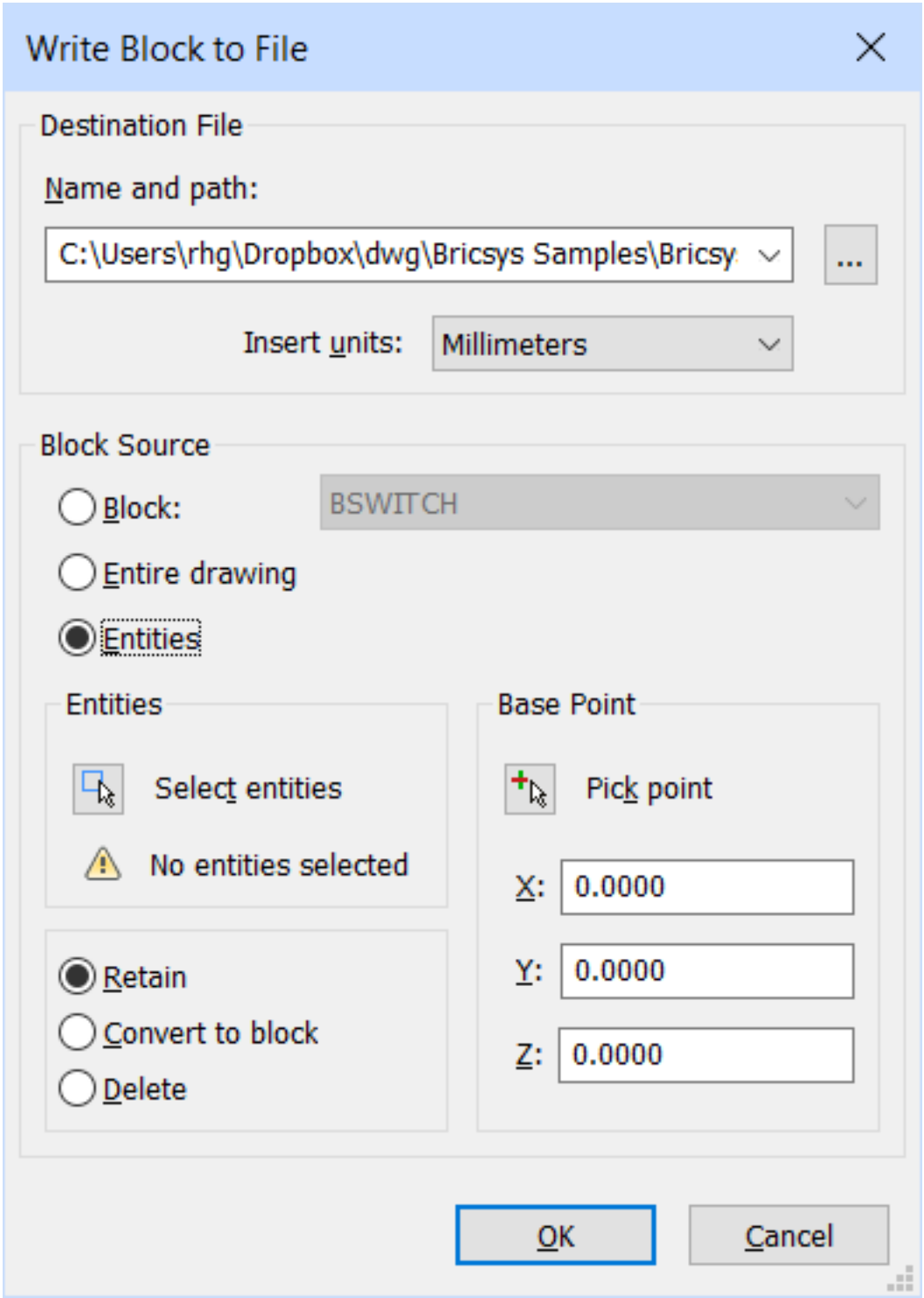
Figure 6.2587



Accessing the Command


command bar: **wblock** alias: **w** menu bar: **Tools | Write Block** ribbon: **Insert | Blocks | Create Block | Save Block** toolbar: **Tools** |  : **WBlock** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V1 dlg.png})

Figure 6.2588



Dialog Box Options

Option	Description
Destination File	Specifies what the file should be named, and where it should be saved **Name and path** - specifies the default path and file name (new-block.dwg) **Browse** - changes the path and file name; displays the Save Block dialog box: • **Save in** - select a folder and/or drive in which to save the file • **File name** - type the name of the file, or accept the default • **Save as type** - choose from DWG or DXF format **Insert Units** - specifies the units to be used by the exported entities when they are later inserted into other drawings: 1. Click the **Insert Units** droplist 2. Choose a measurement unit Units are saved in theINSUNITS variable.
Block Source	Selects the part of the drawing to save to a new file: **Block** - saves a block from the drawing; this option is unavailable when no blocks exist in the drawing (all other options become unavailable): 1. Choose the **Block** radio button 2. Click the droplist 3. Choose the name of a block from the the list **Entire drawing** - saves the entire drawing to the file; this is like using the SaveAs (6.27.3) command. although some longtime CAD users report that using WBlock can strip out errors in the drawing (all other options become unavailable). **Entities** - saves entities that you select to the file

Entities	When the Block Source is set to Entities, then the following options become available: **Select**  - selects entities in the drawing; the dialog box disappears and you are prompted at the command line: 1. **Select entities for block - (choose one or more entities using any selection method.) Entities in set: 1** 2. **Select entities for block - (choose more entities, or else press Enter to return to the dialog box.)** Specifies what to do with the selected entities after you click OK to close the dialog box: • **Retain** - keeps the selected entities as they are; this is like copying the selection to a new drawing • **Convert to block** - converts the selected entities to a block in the current drawing • **Delete** - deletes the entities you had selectedin the drawing; this is like moving the selection to a new drawing
Base Point	Specifies the origin point of the saved entities,in the new file: **Select**  - selects entities in the drawing; the dialog box disappears and you are prompted • **Insertion point for new block or [Annotative] - (select the base point in the drawing or enter x,y,z coordinates; ignore the Annotative option, as it does not work.)** **X**, **Y**, and **Z** - enter coordinates of the base point
OK	Saves the entities -- the block, the entire drawing, or selected entities -- to the file on disk. You can use the Open (6.23.8) command to open the file in a new window, or use the Insert (6.17.13) command to insert the file as a block into any drawing.

Procedures

- To save a block to a separate file (1.22.5)

Related Commands

-WBlock (6.33.2) - writes blocks and other drawing parts to files on disk through the command bar. -Block (6.6.12)- creates blocks at the command line. Block (6.6.11)- creates blocks through a dialog box. Blockify (6.6.13) - Automatically detects equally shaped solids in a model and replaces them by block references.Searches the drawing for an identical set of a 2D entities and replaces them by block references. Export (6.13.15) - exports the drawing in a variety of raster and vector formats. ExportLayout (6.13.16) - exports all visible objects from the current layout to the model space of a new drawing. Insert (6.17.13) - inserts blocks and drawings through a dialog box. -Insert (6.17.14) - inserts blocks and drawings through the command bar. MInsert (6.21.17) - inserts blocks as a rectangular array. SaveAs (6.27.3) - saves drawings in DWG, DWF, and DXF formats for a variety of versions.

Figure 6.2589



6.33.2 -WBlock

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Saves blocks and other drawing entities to DWG/DXF files, (short for "write block").

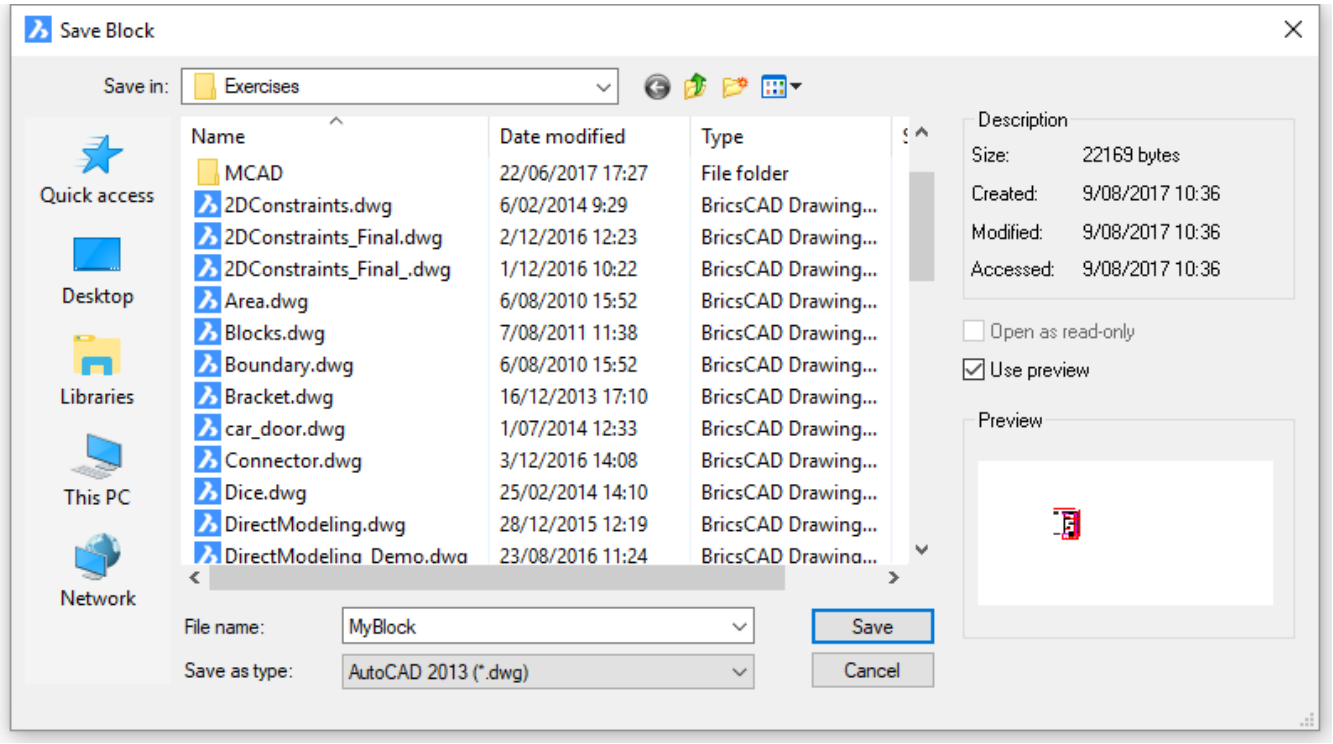
Accessing the Command

command bar: **-wblock** alias: **-w** : **-WBlock** A File dialog displays: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help

1. (Optional) Select a folder from the Save In droplist
2. Enter a name in the File name field or accept the default
3. Click Save. The program prompts you in the command bar:

Block to save as WBlock or [& to include multiple blocks/* for whole drawing] <Select entities>: **(Type the name of the block you want to write to a file, or enter an option)**

Figure 6.2590



Command Options

Option	Description
Block to save as WBlock	Type the name of the block you want to write to a file.
Select entities	Selects individual entities to write to the file on disk: 1. **Insertion point for new block or [Annotative]** - **(pick the origin point of the drawing, or enter x,y,z coordinates; ignore the Annotative option, as it does not work.)** 2. **Select entities for block**- **(select one or more entities.) Entities in set: 1** 3. **Select entities for block**- **(select more entities, or else press Enter to create the drawing file)** The selected entities are deleted from the current drawing. Use the Oops (6.23.7) command to restore the entities.
& to include multiple blocks	Saves multiple blocks from this drawing to the file on disc. Type & or M: **Blocks to save** - type the names of the blocks, each separated by commas. The block definitions are saved to the new drawing.
for whole drawing	Saves the whole drawing, equivalent to SaveAs (6.27.3).Type * or **A**.All entities in the current drawing are saved to the new drawing.

Related Commands

WBlock (6.33.1) - writes blocks and other drawing parts to files on disk through a dialog box. Blockify (6.6.13) -Automatically detects equally shaped solids in a model and replaces them by block references.Searches the drawing for an identical set of a 2D entities and replaces them by block references. Export (6.13.15) - exports the drawing in a variety of raster and vector formats. ExportLayout (6.13.16) - exports all visible objects from the current layout to the model space of a new drawing. Insert (6.17.13) - inserts blocks and drawings through a dialog box. -Insert (6.17.14) - inserts blocks and drawings through the command bar. MInsert (6.21.17) - inserts blocks as a rectangular array. SaveAs (6.27.3) - saves drawings in DWG, DWF, and DXF formats for a variety of versions.

Figure 6.2591



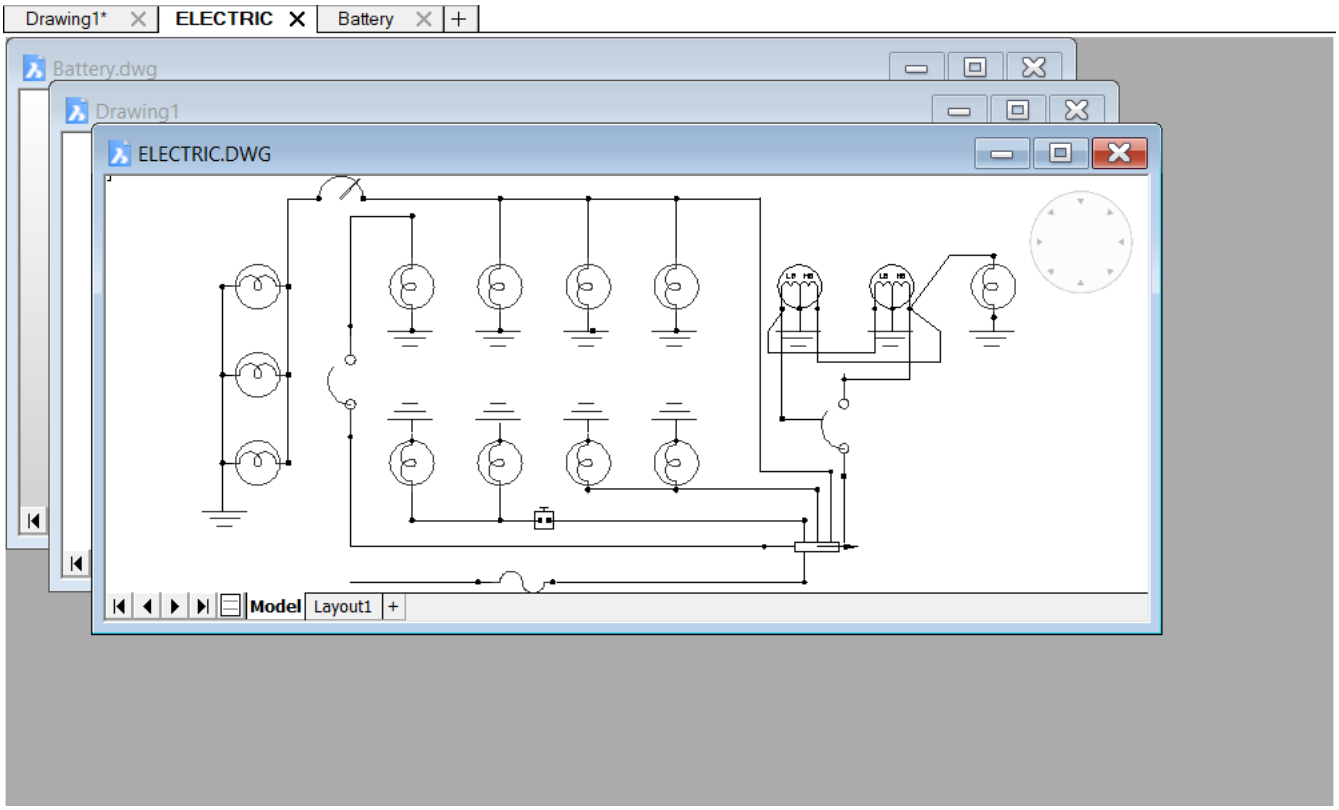
6.33.3 WCascade

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Cascades drawing windows in an overlapping pattern (short for ”window cascade”).

Accessing the Command

 command bar: **wcascade** menu bar: **Window | Cascade** ribbon:**View | Windows | Cascade** : **WCascade** Displays no prompt in the command bar.Windows are cascaded (overlapping), the the current drawing the topmost one. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/W/wcascade-eg.png})

Figure 6.2592



Command Options

No options.

Related Commands

SysWindows (6.27.61) - arranges windows. WClose (6.33.4) - closes the current window. WCloseAll (6.33.5) - closes all windows. WhTile (6.33.9) - tiles windows horizontally. WiArrange (6.33.10) - arranges iconized windows. WvTile (6.33.19) - tiles windows vertically.

Figure 6.2593



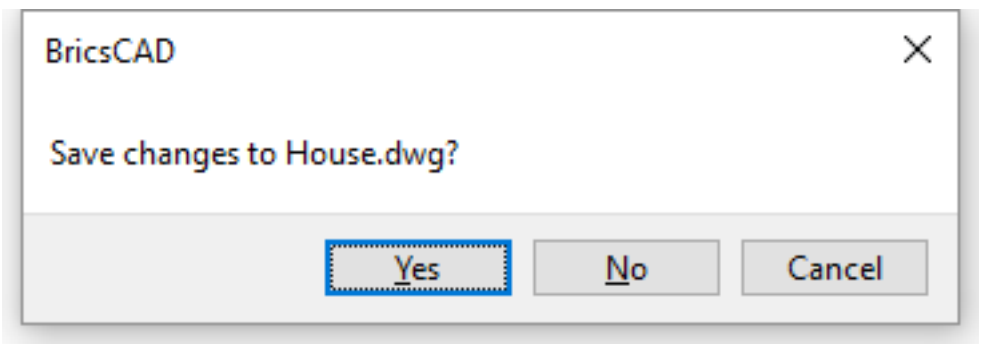
6.33.4 WClose

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Closes the current drawing window.

Accessing the Command

command bar: **wclose** shortcut: **Ctrl+F4** alias: **close** menu bar: **File | Close Window | Close** : **WClose** When the drawing is not saved, then the program displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enU
Click a button.

Figure 6.2594



Command Options

Option	Description
Yes	Saves the drawing.Displays the Save Drawing As dialog box; enter a file name, and then click **Save** . See theSaveAs (6.27.3) command.
No	Does not save the drawing.Closes the drawing without saving.
Cancel	Cancels the command.Returns to the drawing; does not save or close it.

Related Commands

WCascade (6.33.3) - cascades the windows. Close (6.9.25) - closes the current drawing. WCloseAll (6.33.5) - closes all windows. WhTile (6.33.9) - tiles windows horizontally. WiArrange (6.33.10) - arranges iconized windows. WvTile (6.33.19) - tiles windows vertically. SysWindows (6.27.61) - arranges windows.

Figure 6.2595



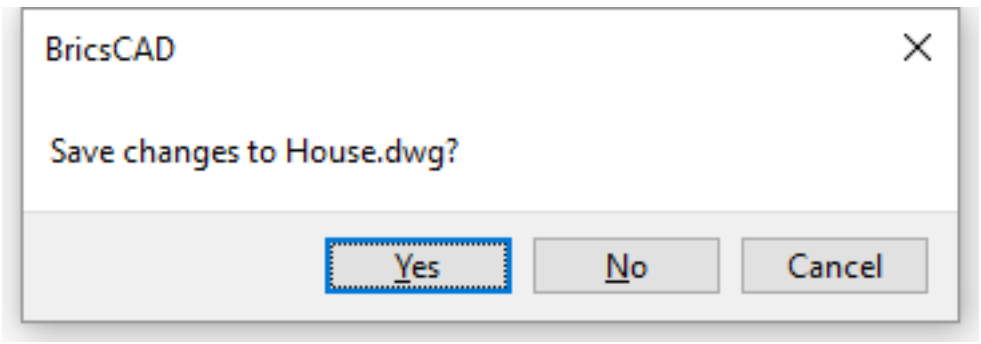
6.33.5 WCloseAll

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Closes all drawing windows at once (short for "windows close all").

Accessing the Command

command bar: **wcloseall** alias: **closeall** menu bar: **Window | Close All** : **wcloseall** If any drawing has not been saved, displays a dialog box for each one: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/s Click a button. This dialog box repeats for every unsaved drawing.

Figure 6.2596



Command Options

Option	Description
Yes	Saves the drawing.Displays the Save Drawing As dialog box; enter a file name, and then click **Save**. See SaveAs (6.27.3) command.
No	Does not save the drawing.Closes the drawing without saving.
Cancel	Cancels the command.Returns to the drawing; does not save or close it.

Related Commands

WCascade (6.33.3) - cascades the windows. WClose (6.33.4) - closes the current windows. WhTile (6.33.9) - tiles windows horizontally. WiArrange (6.33.10) - arranges iconized windows. WvTile (6.33.19) - tiles windows vertically. SysWindows (6.27.61) - arranges windows.

Figure 6.2597



6.33.6 Weblight

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates web lights, which illuminates scenes with a 3D representation of the light's intensity distribution. **Note:** Web lights are defined by IES files, which are provided by lighting manufacturers.



Accessing the Command

 command bar: **weblight** menu bar: **View | Rendering | Lights | New web light** toolbar: **Lights |**  : **Weblight**
Prompts you in the command bar:

- If the LIGHTINGUNITS variable is 0 (No lighting units), the following prompt displays:

Command not allowed when LIGHTINGUNITS = 0

- If the LIGHTINGUNITS variable is 1 (American lighting units) or 2 (International lighting units), the following prompts display:

Specify source position <0,0,0>: **(Specify the position of the light)** Specify target position <0,0,-10>: **(Specify the target point of the light)** Enter an option to change [Name/Intensity factor/ Status/ Photometry/ weB/ shadoW/ filterColor/eXit] <eXit>:**(Enter an option)**

Command Options

Option	Description
Specify source position	Specifies the location of the light in 3D space: • Pick a point in the drawing to position of the light • Enter x,y,z coordinates • Press **Enter** to accept the default location
Specify target position	Specifies the location of the target, the point at which the light is focused: • Pick a point in the drawing to position of the target • Enter x,y,z coordinates • Press **Enter** to accept the default location
Name	Specifies a name for the light; prompts you: **Enter light name <Weblight1>** - **(enter a name or press Enter to accept the default name.)**

Intensity Factor

Specifies the intensity of the light; prompts you: **Enter intensity (0.00 - max float) <1.0>** - **(enter an option:)**

- Enter an intensity values between 0.00 and the maximum value supported by your system (max float = maximum value of a floating point number)
- Press **Enter** to accept the default value

Status

Toggles the use of this light; prompts you: **Enter status [ON/OFf] <ON>** - **(enter an option:)**

- **ON** - light is included when calculating renderings
- **OFf** - light is excluded

Photometry

Specifies the photometric properties of the light; prompts you: **Enter a photometric option to change [Intensity/Color/eXit] <Intensity>** - **(enter an option:)**

- **Enter intensity (Cd) or enter an option [Flux/Illuminance] <1500.0>**: - **(enter the intensity expressed in candela or an option:) Enter Flux (Lm)**- **(enter the luminous flux expressed in lumens.) Enter Illuminance (Lx or Fc) or enter an option [Distance] <9424.8>**- **(enter the illuminance expressed in lux or foot-candles.) Enter Distance** - **(Enter a distance expressed in drawing units.)**

- **Enter color name or enter an option [?/Kelvin] <D65>**- **(enter a color name.) ?** - lists the names of available colors, such as:

```
D65        Fluorescent        Coolwhite        White
```

Enter Kelvin temperature <3600.0> - **(enter a color temperature expressed in degrees Kelvin.)**

Web

Specifies the light distribution map, called a ”web” as an uneven light distribution can look vaguely like a spider’s web: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/W/web.jpg}) Prompts you: **Enter a Web options to change: File/X/Y/Z/<Exit>** - **(enter an option:)**

- **Enter Web file** - **(enter the name of an IES file, which is a text file that describes the intensity of a light source at many points on a spherical grid (image credit Vertheim).)**
- **Enter Web X rotation in degrees <0.0000>** - **(define the X rotation for the web.)**
- **Enter Web Y rotation in degrees <0.0000>** - **(define the Y rotation for the web.)**
- **Enter Web Z rotation in degrees <0.0000>** - **(define the Z rotation for the web.)**

Shadow	Specifies the look of shadows cast by this light; prompts you: **Enter shadow settings [Off/Sharp/soFt mapped/soft sAmpled] <Sharp>** - **(enter an option:)** • **Off** - disables the calculation of shadows for the light. • **Sharp** - displays shadows with sharp edges. Use this option to increase performance. • **Soft mapped** - displays realistic shadows with soft edges. Prompts you: **Enter map size [64/128/256/512/1024/2048/4096] <256>:** • **Soft sampled** : Displays realistic shadows with softer shadows based on extended light sources; prompts you: **Enter an option to change Shape/sAmples/Visible/<eXit>** - **(enter an option:) Enter shape [Disk/Rect] <Disk>** - **(type D or R.) Enter radius <0.0>** - **(enter the radius of the disk.) Enter Width <0.0>** - **(enter width of the rectangle.) Enter Length <0.0>** - **(enter length of the rectangle.)** • **Enter Shadow Sample <16>** - **(specify the sampling size of the shadow; larger numbers are more accurate but take longer to render.)** • **Enter Shape Visibility [Yes/No] <No>** - **(type Y or N to cast a shadow that represents shapes (more accurate) or is rectangular (faster to render).)**
Filter Color	Specifies the color of the light; prompts you: **Enter true color (R,G,B) or enter an option [Index color/Hsl]:<255,255,255>** - **(enter an option:)** • **Enter true color** - the RGB color model is an additive color model in which red, green, and blue light are added together in various ways to reproduce a broad array of colors. The name of the model comes from the initials of the three additive primary colors, red, green, and blue. The component values are stored as integer numbers in the range 0 to 255, the range that a single 8-bit byte can offer (by encoding 256 distinct values). • **Index color** - specifies an AutoCAD Color Index color. • **Hsl** - specifies the color using three parameters Hue, Saturation and Lightness
Exit	Concludes the command and creates the light.

Related Commands

Distantlight (6.10.32) - creates a distant light. Light (6.20.28) - creates a new light in the drawing. Lightlist (6.20.29) - displays the Drawing Explorer - Lights dialog window. Pointlight (6.24.42) - creates a point light. Render (6.26.22) - calculates a photorealistic image of the 3D model. Spotlight (6.27.42) - creates a spot light. Sunproperties (6.27.57) - displays the Sun Properties of the Active Viewport editor in the Drawing Explorer - Lights dialog.

Figure 6.2598

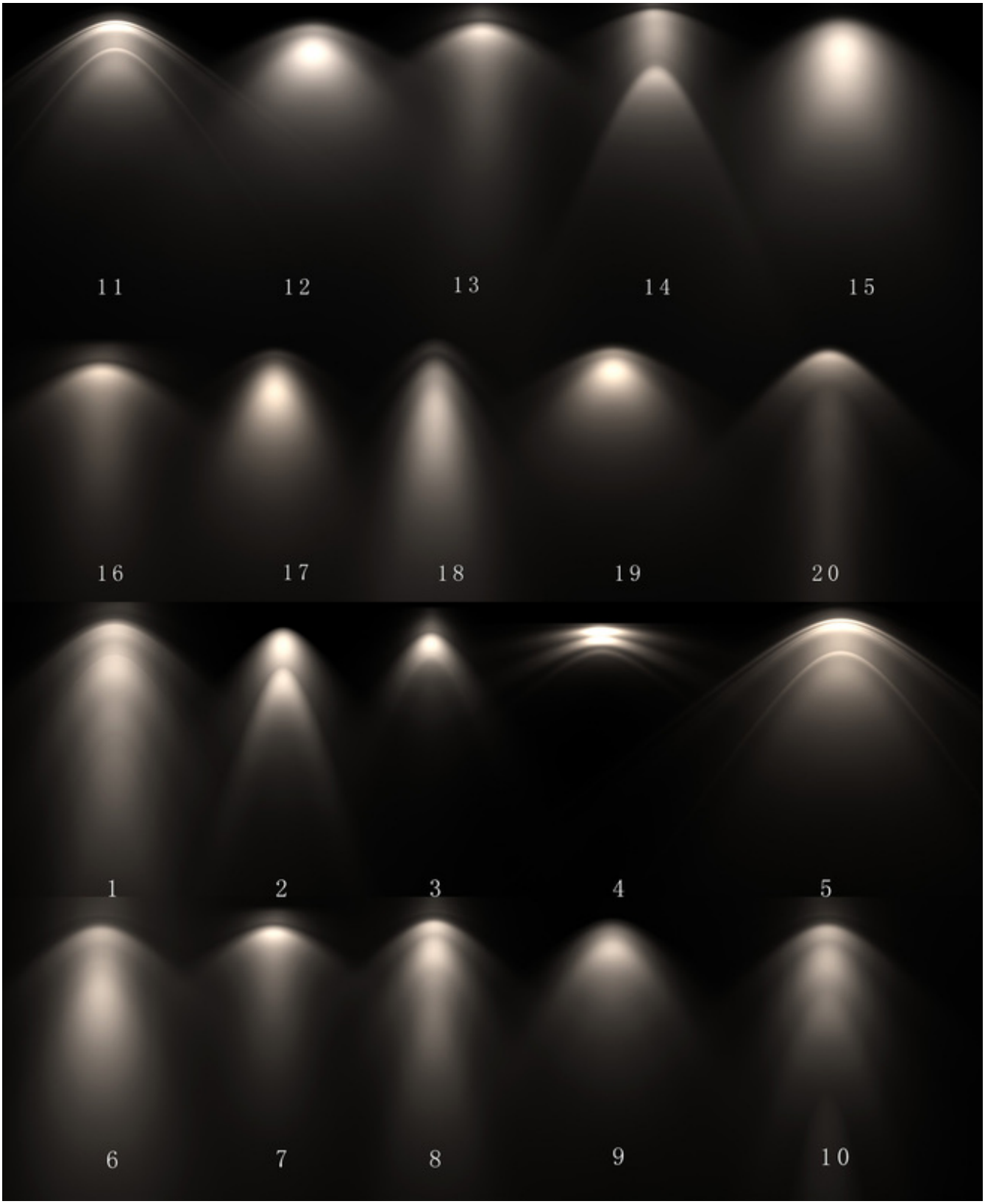
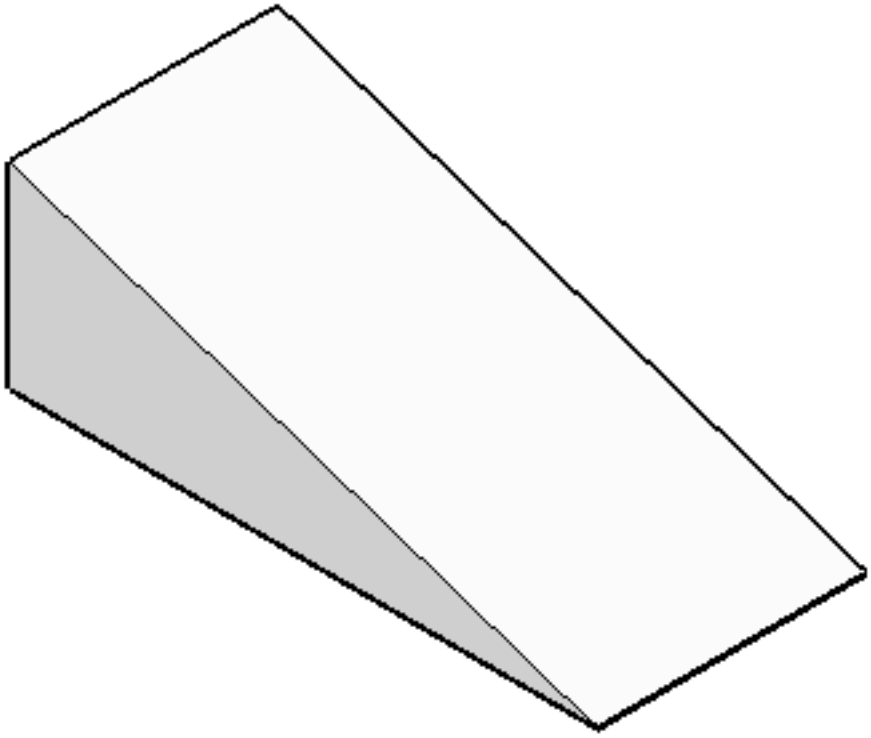


Figure 6.2599



Figure 6.2600



6.33.7 Wedge

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws wedges:

- Classic: wedge is drawn from a 3D polyface mesh; see the Ai_Wedge (6.5.16) command
- Shape, Pro and Platinum: wedge is drawn as a 3D solid

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/W/wedge-eg.png})

Accessing the Command

 command bar: **wedge** alias: **we** menu bar: **Model| 3D Solids | Wedge** ribbon: **Solid | Create | Primitive | Wedge** toolbar: **3D Solids |**  : **Wedge Prompts you in the command bar:** Set corner of wedge or [Center] <4.25,0.00,1.84>: **(Pick a point, or enter coordinates)** Set opposite corner or [Cube/Length of side]: **(Pick another point, or else enter an option)** Height of wedge [2Point]: **(Specify the height, or else enter an option.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image29.jpg}) If Dynamic Dimensions (1.15.9) are active, you can enter the dimensions in the dynamic dimensions entry fields. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/WedgeDynDim.png})

Figure 6.2601

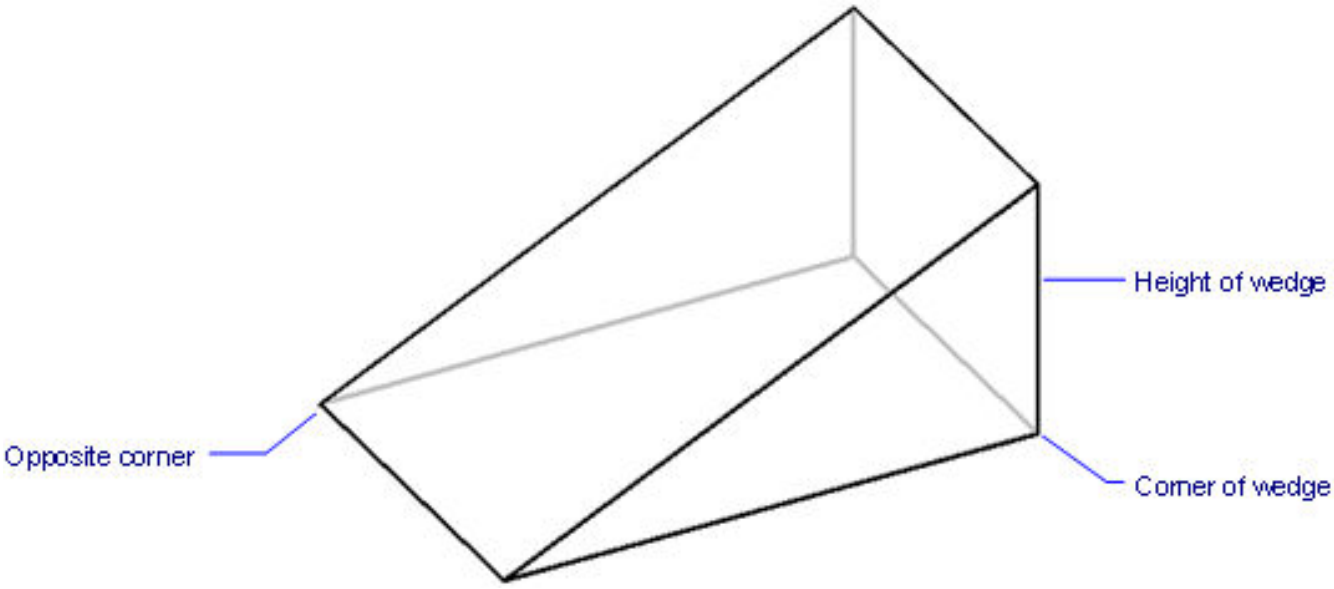
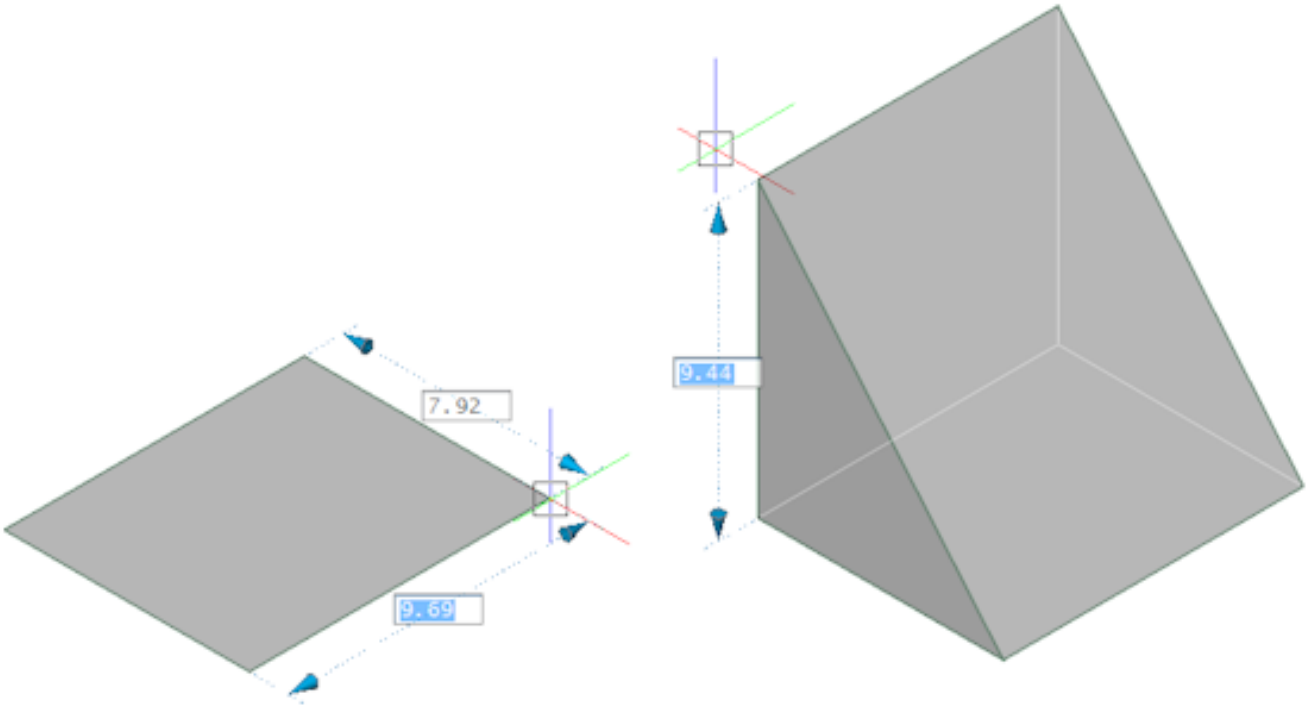


Figure 6.2602



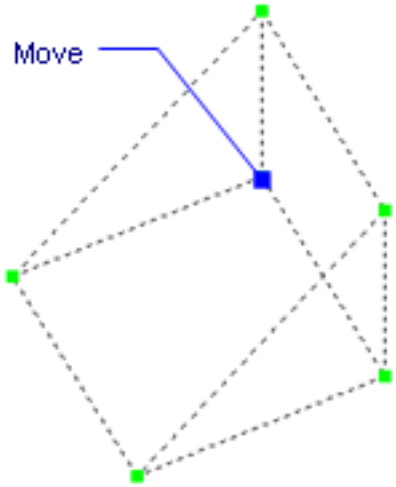
Command Options

Option	Description
Set corner of wedge	Specifies one corner of the wedge's base: • Pick a point • Enter x,y or x,y,z coordinates • Press **Enter** to accept the default coordinates • Or select the Center option Prompts you: **Set opposite corner** - **(pick another point, or enter other coordinates.)**
Center	Draws the wedge based on the center point of the entire wedge and a corner of the base; prompts you: 1. **Center of wedge** - **(specify the center of the entire wedge (not the base).)** 2. **Corner of wedge** - **(specify the location of one of the base's corners..)**
Height of wedge	Specifies the height of the wedge; the tall end of the wedge is drawn from the first corner point that you specified: • Pick a point • Enter a height; enter a negative height to draw the wedge upside down • Press **Enter** to accept the default height • Or select the**2Point** option
Cube	Draws a wedge with equal sizes for its length, width, and height; prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i **Length of sides of cubic wedge** - **(enter an option:)** • Enter a number for the distance that applies equally for the length, width, and height • Press **Enter** to accept the default
Length	Draws the wedge by specifying distances for each of its length, width, and height; don't enter coordinates! Prompts you: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i 1. **Length of side** - **(specify the length, which will be drawn in the x direction; enter a negative number to draw the wedge in the negative x direction.)** 2. **Width across wedge** - **(specify the width of the base, which will be drawn in the y direction.)** 3. **Height of wedge** - **(specify the height, which will be drawn in the z direction.)** In all cases, you can enter a negative number to drawn the wedge in the opposite direction, such as a negative height to draw it upside down.

Grips Editing

Wedges can be edited directly through grips:

Figure 6.2603



1. Select the wedge. Notice that it has several grips.
2. Drag a grip to move the wedge. (All grips perform the same function: move.)

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image757.png})

Related Commands

Box (6.6.17) - creates 3D boxes as solid models. Cone (6.9.63) - creates 3D cones as solid models. Cylinder (6.9.60) - creates 3D cylinders as solid models. Dish (6.10.30) - creates the lower half of 3D balls as solid models. Dome (6.10.34) - creates the upper half of 3D balls as solid models. Extrude (6.13.21) - extrudes 2D entities into 3D solid models. Interfere (6.17.18) - creates 3D volumes of intersecting 3D solid models. Intersect (6.17.19) - removes volumes not in common between two solid models. MassProp (6.21.4) - reports the mass properties of 3D solid models and 2D regions. Pyramid (6.24.58) - creates a pyramid as a 3D solid (Pro and Platinum). Creates a pyramid as a polygon mesh (Classic) Revolve (6.26.30) - revolves 2D entities into 3D solid models. Slice (6.27.36) - cuts 3D solids into two parts. SolidEdit (6.27.39) - edits the bodies, faces, and edges of 3D models. Sphere (6.27.64) - creates 3D balls as solid models. Subtract (6.27.56) - removes one solid model from a second. Torus (6.29.22) - creates 3D donuts as solid models. Union (6.30.6) - joins two or more solid models into a single entity.

Figure 6.2604

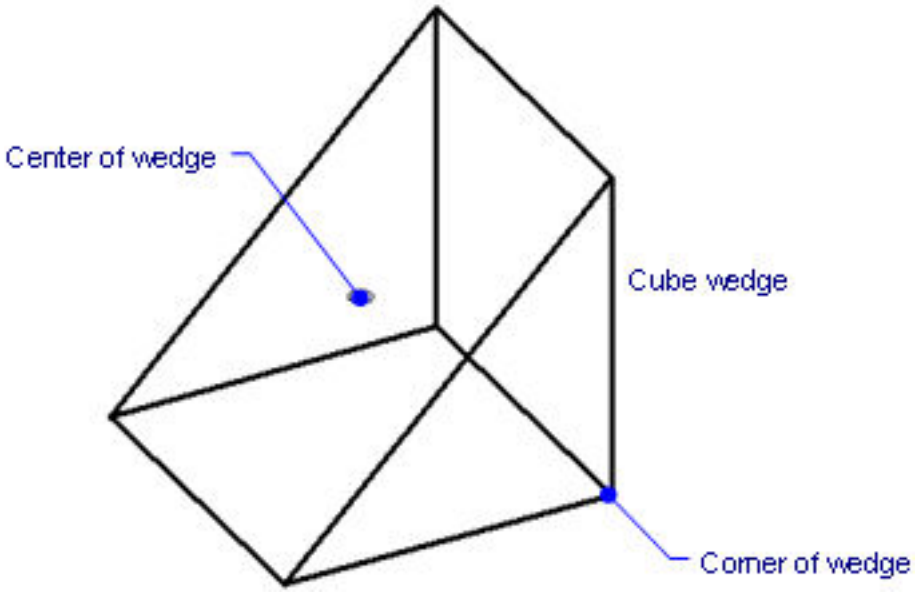


Figure 6.2605

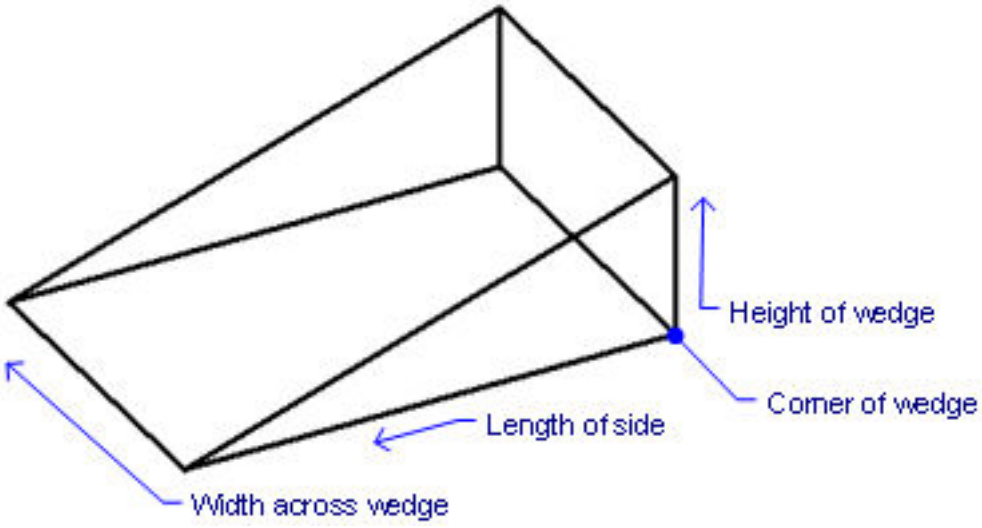


Figure 6.2606



6.33.8 WhoHas

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Reports the ownership of a drawing file that is in use by someone else.

Accessing the Command

command bar: **whohas** : **WhoHas** Displays a file dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS dlg.png})

1. Choose a drawing to check on its ownership.
2. Click **Open**. The program displays the following dialog box.
3. Review its message, then clik **OK**.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/W/whohas-1-dlg.png}) **Note:**The information in the dialog box is stored in a temporary DWL (drawing lock) file, which is deleted when the drawing is closed.

Figure 6.2607

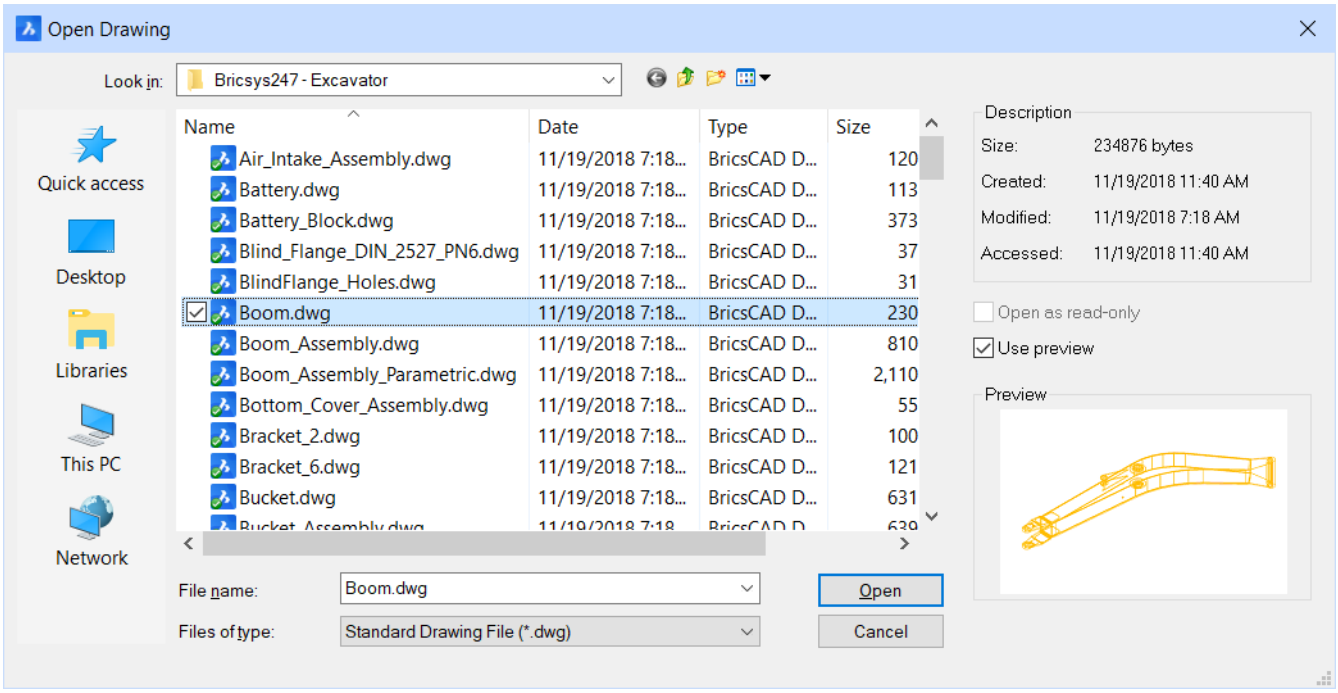
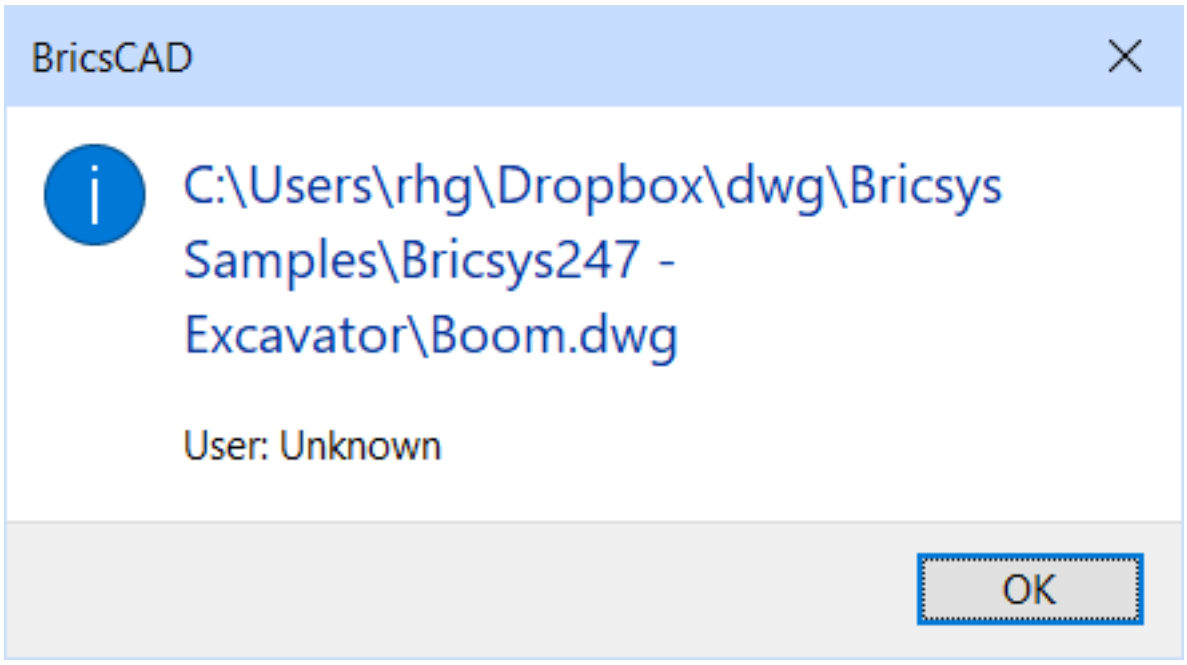


Figure 6.2608



Command Options

No Options.

Related Commands

DwgProps (6.10.42) - opens the Drawing Properties dialog box, showing the general information and user defined properties stored with a drawing. Open (6.23.8) - opens drawing (DWG), template (DWT), and interchange (DXF) files. SecurityOptions (6.27.15) - sets a password to protect a drawing

Figure 6.2609



6.33.9 WhTile

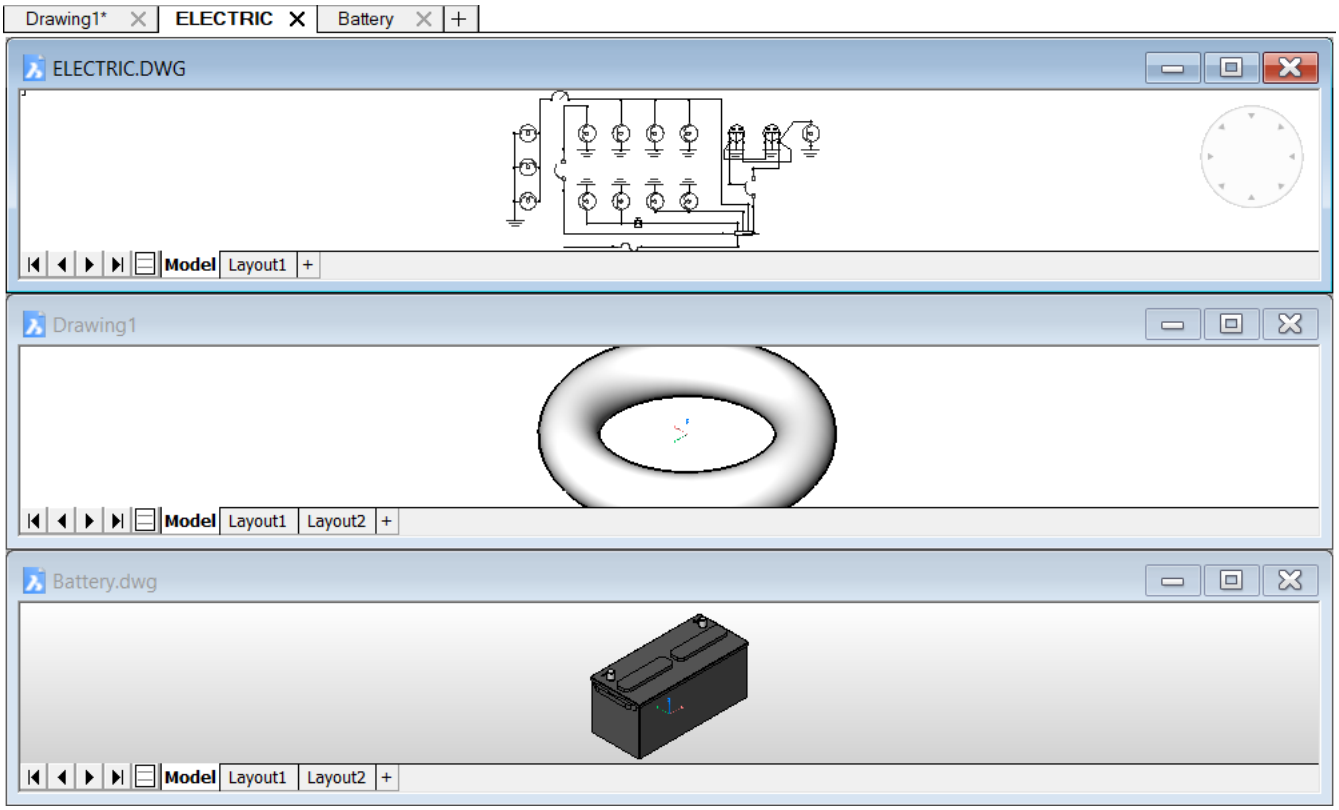
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Tiles drawing windows horizontally (short for "window horizontal tile").

Accessing the Command



command bar: **whtile** menu bar: **Window | Tile Horizontally** ribbon:**View | Windows | Tile Horizontally** : **WhTile** Displays no prompt in the command bar.Windows are tiled horizontally, with the most recent drawing at the top. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/W/whtile-eg.png})

Figure 6.2610



Command Options

No options.

Related Commands

WCascade (6.33.3) - cascades the windows. WClose (6.33.4) - closes the current window. WCloseAll (6.33.5) - closes all windows. WiArrange (6.33.10) - arranges iconized windows. WvTile (6.33.19) - tiles windows vertically. SysWindows (6.27.61) - arranges windows.

Figure 6.2611



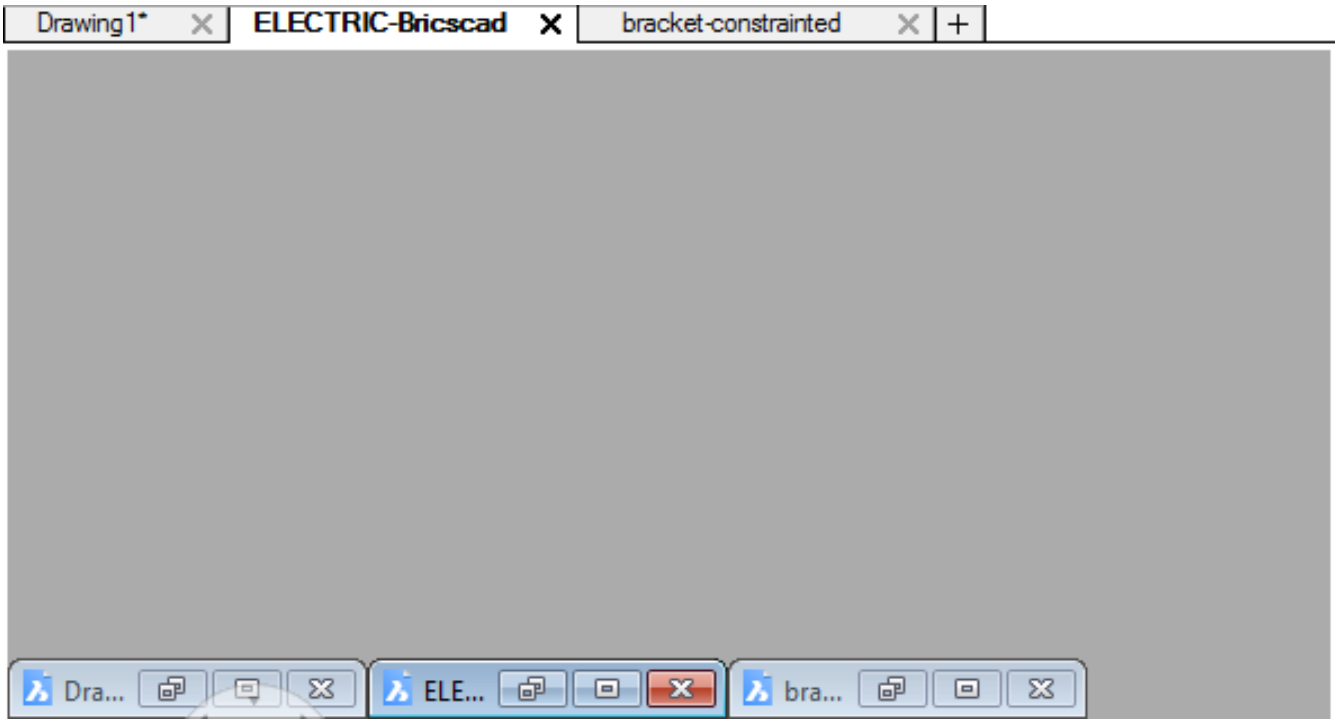
6.33.10 WiArrange

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Lines up iconized drawing windows (short for ”window icon arrange”).

Accessing the Command

command bar: **wiarrange** : **WiArrange** Displays no prompt in the command bar.Arranges the icons of minimized windows at the bottom of the graphical screen.This command works only when windows are minimized as icons. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/W/wiarrange-eg.png})

Figure 6.2612



Command Options

No options.

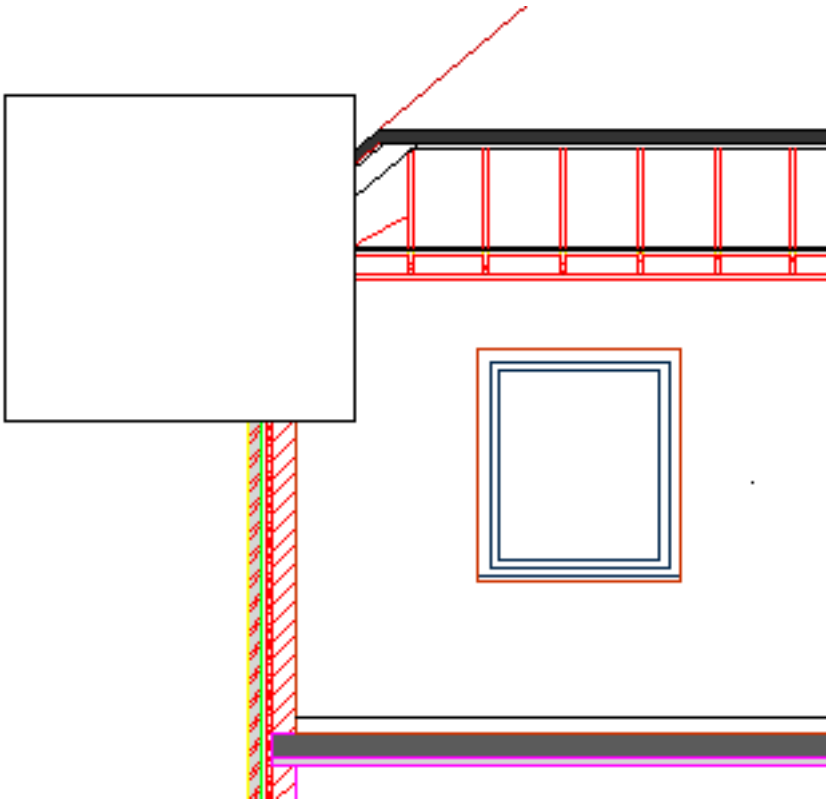
Related Commands

WCascade (6.33.3) - cascades the windows. WClose (6.33.4) - closes the current window. WCloseAll (6.33.5) - closes all windows. WhTile (6.33.9) - tiles windows horizontally. WvTile (6.33.19) - tiles windows vertically. SysWindows (6.27.61) - arranges windows.

Figure 6.2613



Figure 6.2614



6.33.11 WipeOut

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Draws wipeouts in a variety of shapes to mask parts of drawings; controls the display of wipeout frames. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/WipeOut.png

Accessing the Command

 command bar: **wipeout** menu bar: **Draw | WipeOut** ribbon:**Draw | Markup | Wipeout** toolbar: **Draw |**  : **WipeOut Prompts you in the command bar:** Specify start point or [Polyline/Frames]: **(Pick a point, or enter an option)** Specify next point: **(Pick another point)** Specify next point or [Undo]: **(Pick another point, or type U)** Specify next point or [Undo/Close]: **(Enter another point, or type C to end the command)** Use the DrawOrder (6.10.38) command to move the wipeout in front or behind other entities.

Command Options

Option	Description
Specify start point	Specifies the first corner of the wipeout entity: • Pick a point • Enter x,y coordinates

Specify next point	Specifies the second corner of the wipeout entity: • Pick a point • Enter x,y coordinates
Polyline	Uses a closed polyline to construct the wipeout frame. First, draw a closed polyline made of line segments (no arcs or splines) with thePLine (6.24.27)command, then return to this command. Prompts you: 1. **Select a closed polyline** - **(pick a closed polyline composed of line segments.)** 2. **Delete selected polyline? [Yes/No] <No>**- **(enter an option:)** • **Yes** - erases the polyline. • **No** - does not erase the polyline, leaving it in place
Frames	Toggles the display of wipeout frame; this option applies to the frames of all wipeout entities in the drawing; prompts you: **Enter frame setting [ON/OFf/Display but not plot] <ON>** - **(choose an option.)** • **OFf**- turns off all frames; when the fame is off, the wipeout cannot be selected • **ON** - turns on frames; wipeouts can be selected and be printed • **Display but not plot** - turns on the display of frames in the drawing,but they are not printed
Undo	Removes the last segment.
Close	Creates the wipeout.

Grips Editing

Wipeouts can be edited directly through grips:

1. Select the wipeout. Notice that it has one grip at each vertex.
2. Drag a grip to stretch the vertex.

Procedures

- To create a wipeout using points (1.17.27)
- To create a wipeout from a polyline (1.17.27)
- To control the display of the edges of wipeouts (1.17.27)

Related Commands

DrawOrder (6.10.38) - changes the display order of overlapping entities. PLine (6.24.27) - draws polylines. RevCloud (6.26.29) - draws revision clouds.

Figure 6.2615



6.33.12 WmfOut

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) * Command only available on the Windows platform Exports the current view to a file in WMF, EMF, or SLD formats (short for ”windows metafile output”).

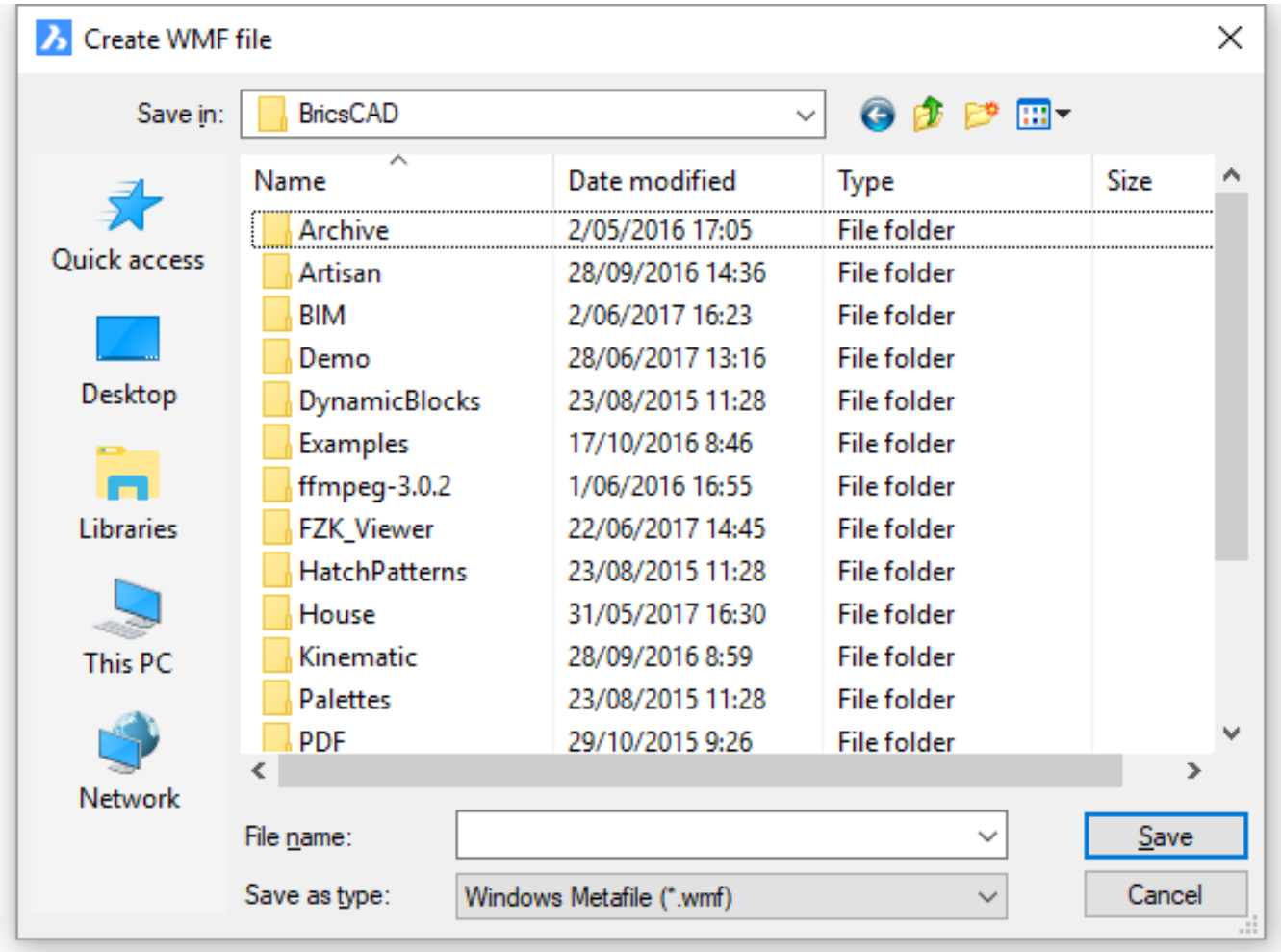
Accessing the Command

command bar: **wmfout** alias: **wo** Before using this command, set the viewport the way you wish it to be saved. : **WmfOut** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/WmfO

1. Enter a file name
2. Choose a file type
3. Click **Save**

WMF, EMF, and SLD files can be viewed in the current viewport with the VSlide (6.31.22) command.

Figure 6.2616



Dialog Box Options

Option	Description
File name	Specifies the name of the slide (snapshot).
Save as type	Specifies the file format: • **SLD** - compatible with AutoCAD • **EMF** - enhanced WMF format • **WMF** - Windows meta format, a hybrid of vector and raster formats The PictureExportScalepreference sets the scale factor when exporting to WMF or EMF.
Save	Saves the file.

Related Commands

BmpOut (6.6.14) - exports drawings in BMP format. Export (6.13.15) - exports drawings in BMP, DWF, DXF, DWT, PDF, WMF, EMF, SLD, and older DWG formats. MSlide (6.21.31) - exports drawings in WMF, EMF, and SLD formats.

Figure 6.2617



6.33.13 WNext

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the next drawing window, useful when multiple drawings are open (sort for ”window next”).

Accessing the Command

command bar: **wnext** keyboard: **Ctrl + Tab** menu bar: **Windows | Next** : **WNext** Bring the next drawing window to the foreground.

Command Options

No command options.

Related Commands

Wprev (6.33.16) - Displays the previous drawing window if multiple drawings are open simultaneously.

Figure 6.2618



6.33.14 WorkSets

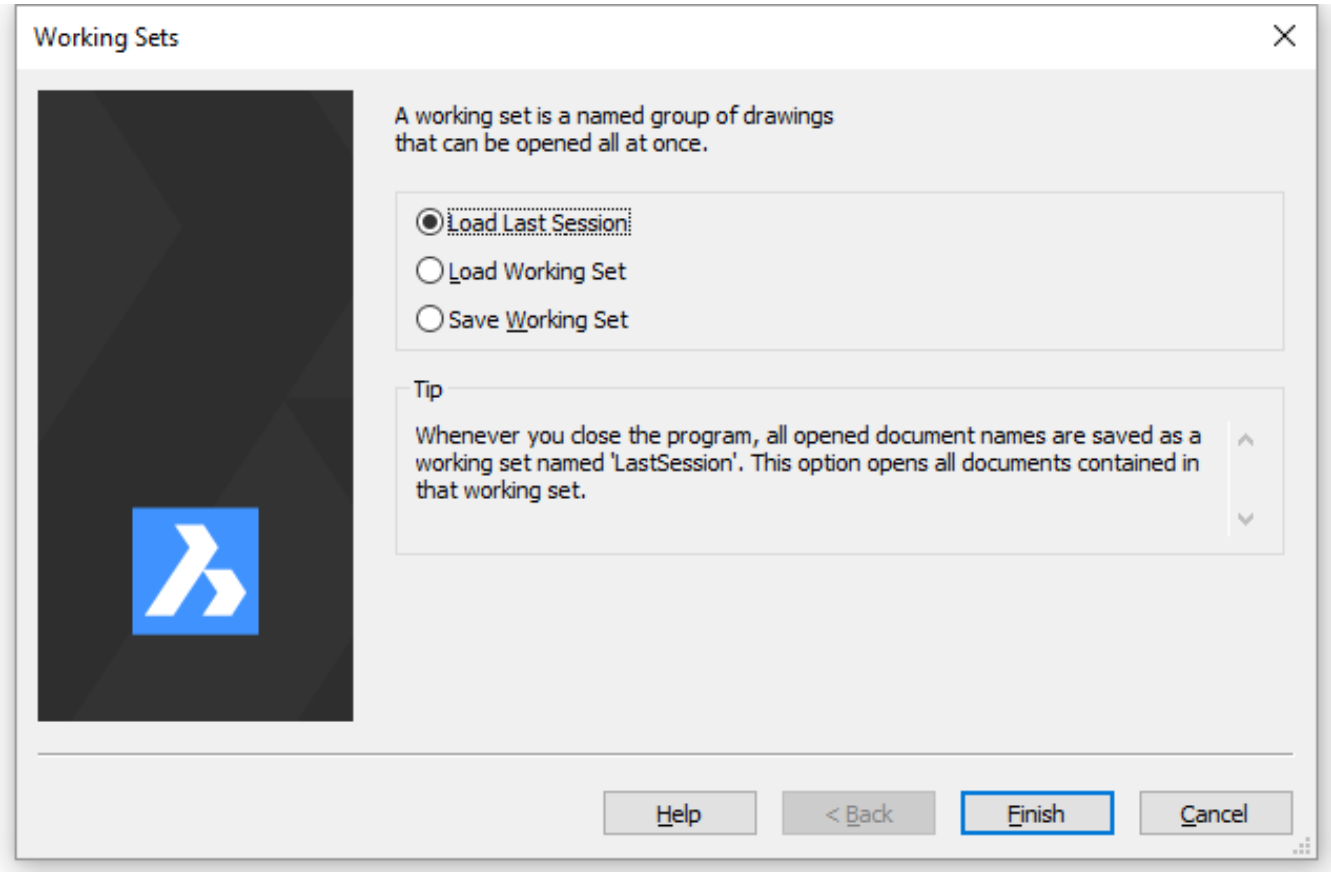
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Creates and loads sets of drawing files, making it convenient to load an entire group of drawings at once.

Accessing the Command

 command bar: **worksets** menu bar: **File | Working Sets** ribbon:**Home | File | Working Sets** : **WorkSets** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/WorkSetsLastDia

1. Select an option.
2. ClickNext or Finish.

Figure 6.2619



Dialog Box Options

Option	Description
Load Last Session	Opens all drawings that where open when the last BricsCAD session was closed: 1. Choose **Load Last Session**. 2. Click**Finish**.BricsCAD opens all the drawings that were open the last time you closed the program.
Load Working Set	Opens an previously saved working set: 1. Choose **Load Working Set**. 2. Click**Next**. 3. Choose the name of a workset from the **Select a Workset** list:(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so 4. Click**Finish** button. The programopens all the drawings that are part of the workset.

Save Working Set	Saves drawingscurrently open by name as a working set: (\vref {fig:C:/help/enUS/V18/CmdRef/source/WorkSetSaveDialog.png}) **Save** - displays the Save Workset dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/V 1. Enter a name for the working set of drawings 2. Click **OK**. The program reports that has saved the workset. 3. Click **Finish** to close the dialog box. Working sets are saved in the registry and so they are defined differently for each user profile.See the ProfileManager (6.24.47) command to learn more about user profiles. **Delete** - removes an named working set: 1. Select a workset name 2. Click **Delete**.The program asks, ”Are you sure you want to delete workset?” 3. Click **Yes**. The name of the workset is removed from the list. 4. Click **Finish** to close the dialog box. Redefine (overwrites) an existing named working set: 1. Click the name of a workset in the list. 2. The program asks, ”The workset already exists. Do you want to replace it?” Click **Yes**. The program saves the current set of open drawings to the named workset. 3. Click **Finish** to close the dialog box.

Related Commands

Open (6.23.8) - opens drawing (DWG), template (DWT), and interchange (DXF) files.

Figure 6.2620

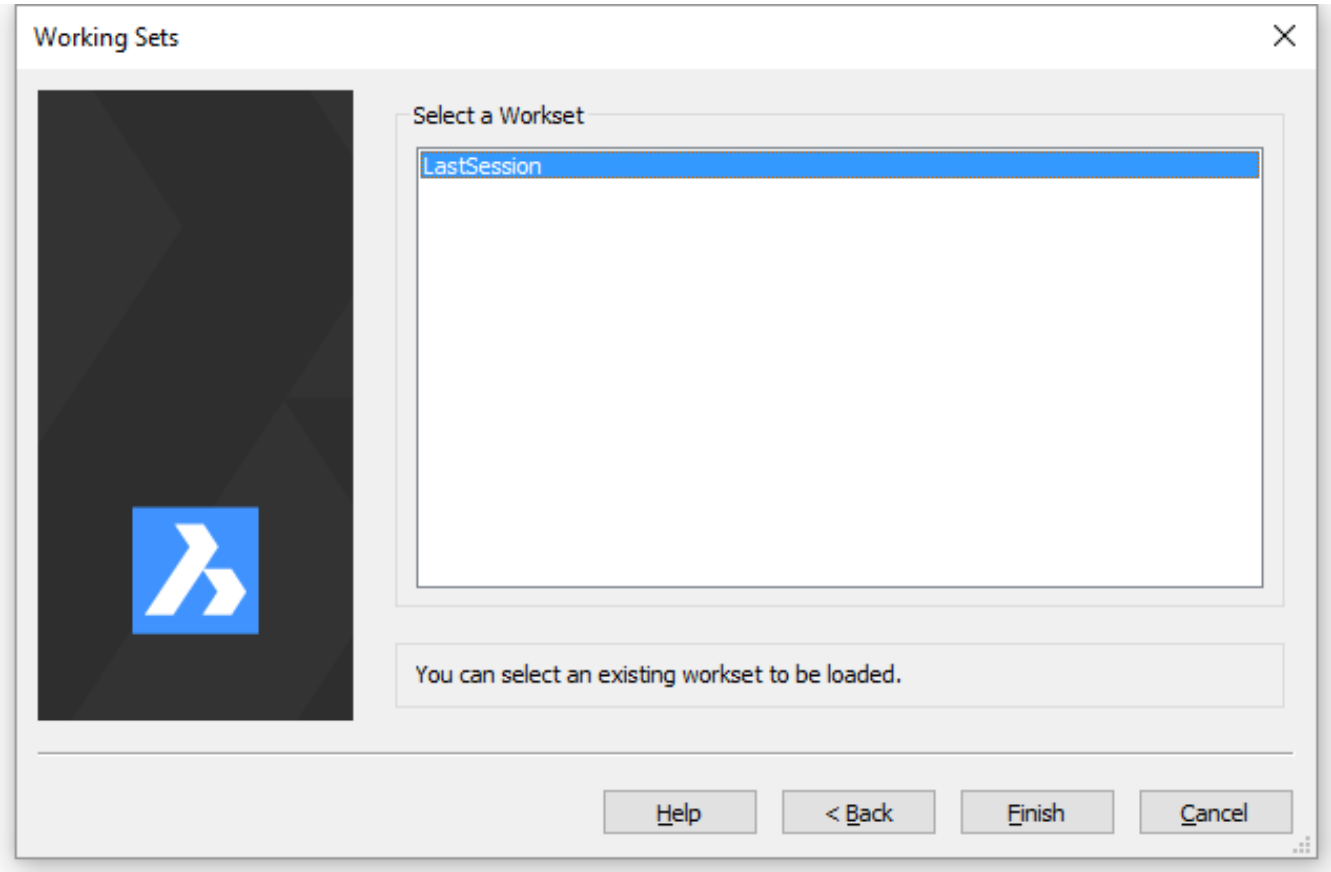


Figure 6.2621

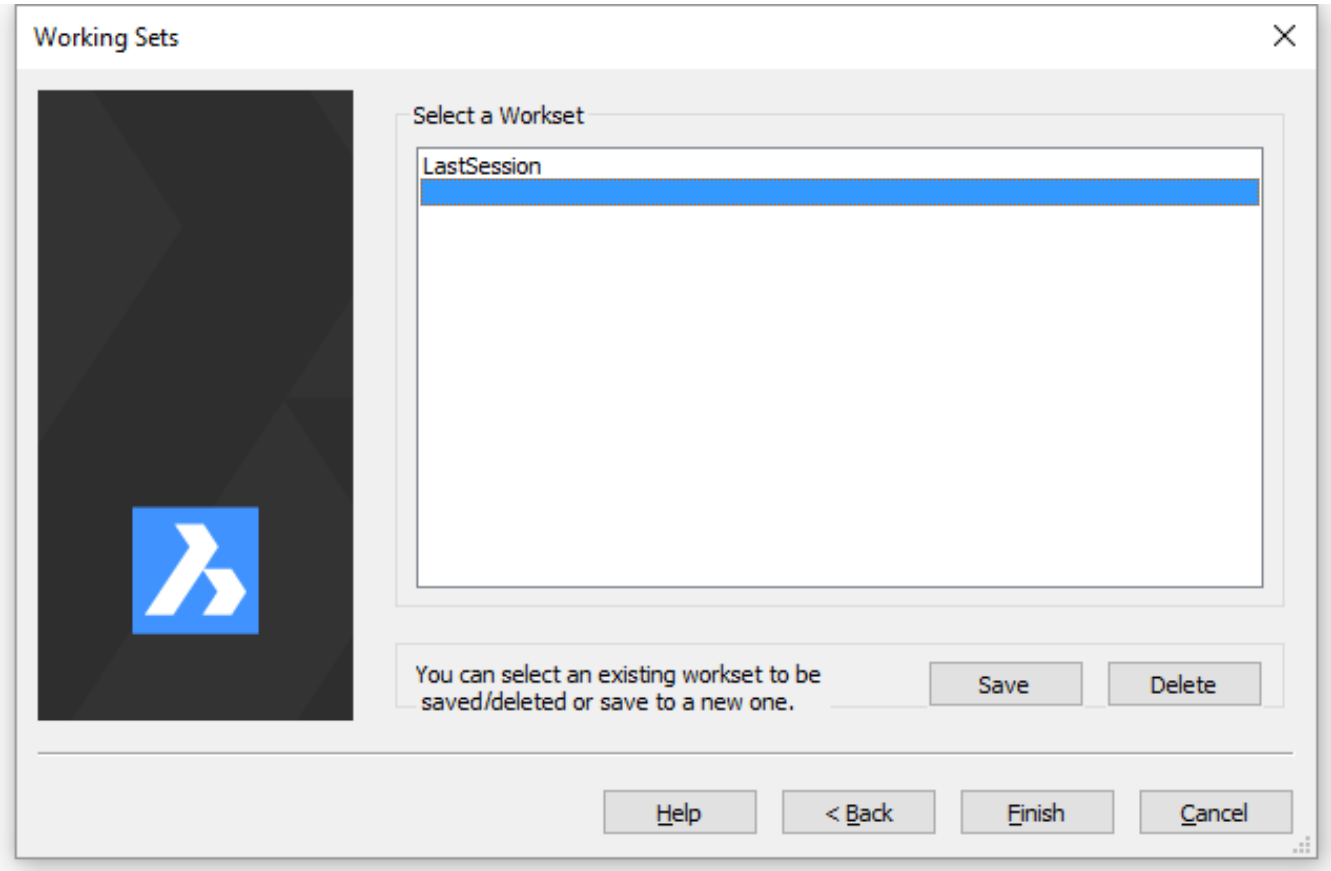


Figure 6.2622

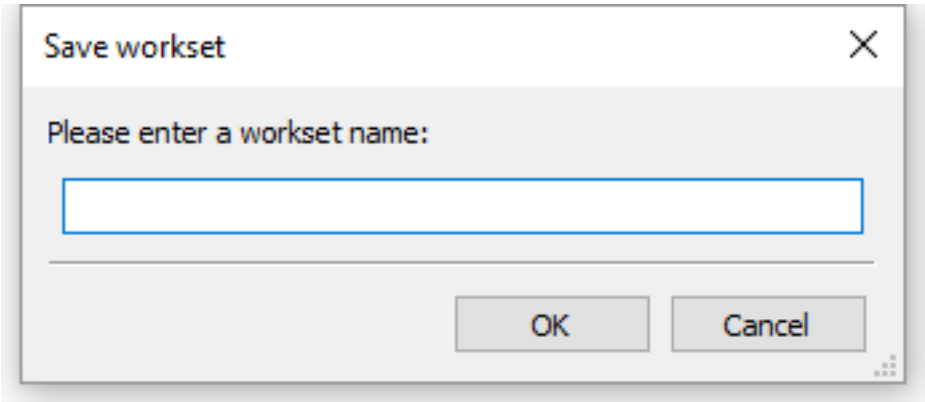


Figure 6.2623



6.33.15 Workspace

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Sets the current workspace and lets you create, modify, and save workspaces.

Accessing the Command

command bar: **workspace** : **Workspace Prompts you in the command bar:** Workspace [set Current/Save As/Rename/Delete/S <Set Current>: **(Select an option)**

Command Options

Option	Description
Set Current	Sets the current workspace. Prompts you: **Enter workspace name to make current [?] <CURRENT_WORKSPACE>** - **(enter an option:)** • Press**?** to list the existing workspaces in the window; displays a list that looks something like this: `Workspace Name -------------- BIM Drafting Drafting (to` • Type the name of the workspace • Press **Enter** to keep the current workspace
Save As	Saves the current settings to a new workspace; prompts you: **Save current workspace as <CURRENT_WORKSPACE>** - **(enter an option:)** • Press**?** to list the existing workspaces in the window • Press **Enter** to save the current workspace, thereby changing it • Type the name of a new workspace
Rename	Renames a workspace. Prompts you: **Enter workspace to rename [?] <CURRENT_WORKSPACE>**-enter an option: • Press**?** to list the existing workspaces in the window • Press **Enter** to rename the current workspace • Type the name of another workspace to rename; prompts you: **Enter new workspace name <CURRENT_WORKSPACE>** - type a new name

Delete	Deletes a workspace. Prompts you: **Enter workspace to delete [?]** - **(enter an option:)** • Press**?** to list the existing workspaces in the window • Type the name of an existing workspace,; prompts you: **Do you really want to delete the workspace "WORKSPACE"? [Yes/No] <N>- (type Yto delete the workspace.)**
Settings	Opens the Workspaces tab of the Customizedialog box.Make adjustments to the workspace of your choice. See theCustomize (6.9.58)command.

Procedures

- Managing workspaces (1.13.10)
- Setting the current workspace (1.13.10)
- Creating a workspace (1.13.10)
- Editing a workspace (1.13.10)
- Deleting a workspace (1.13.10)

Related Commands

Customize (6.9.58) - customizes the user interface of BricsCAD: menus, toolbars, keyboard shortcuts, mouse actions, tablet, Quad settings, workspaces, aliases, and shell commands. WsSave (6.33.17) - saves the current workspace under a different name, thus creating a new workspace.

Figure 6.2624



6.33.16 Wprev

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Displays the previous drawing window, when multiple drawings are open (short for ”window previous”).

Accessing the Command

command bar: **wprev** keyboard: **Ctrl + Shift + Tab** menu bar: **Windows | Previous** : **Wprev** Bring the previous drawing window to the foreground.

Command Options

No command options.

Related Commands

Wnext (6.33.13) - displays the next drawing window if multiple drawings are open simultaneously.

Figure 6.2625



6.33.17 WsSave

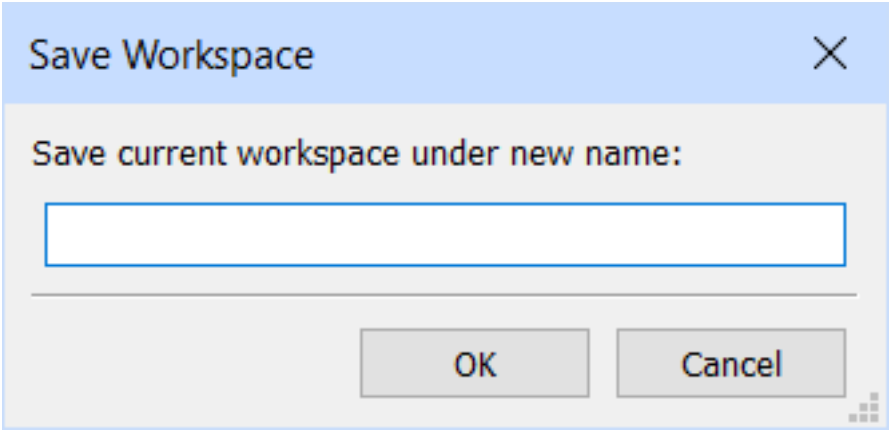
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Saves the current workspace (arrangement of user interface elements) under a different name, creating a new workspace.

Accessing the Command

command bar: **wssave** menu: **Tools | Save Workspace** : **WsSave** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/W/wssave-dlg.png})

1. Type a name.
2. Click**OK**.

Figure 6.2626



Dialog Box Options

Options	Description
Save current workspace under new name	Specifies the name for the new workspace. You can access it later through the Workspaces toolbar or from the status bar.

Related Commands

Customize (6.9.58) - customizes the user interface of BricsCAD: menus, toolbars, keyboard shortcuts, mouse actions, tablet, Quad settings, workspaces, aliases, and shell commands. Workspace (6.33.15) - sets the current workspace and lets you create, modify and save workspaces.

Figure 6.2627



6.33.18 WsSettings

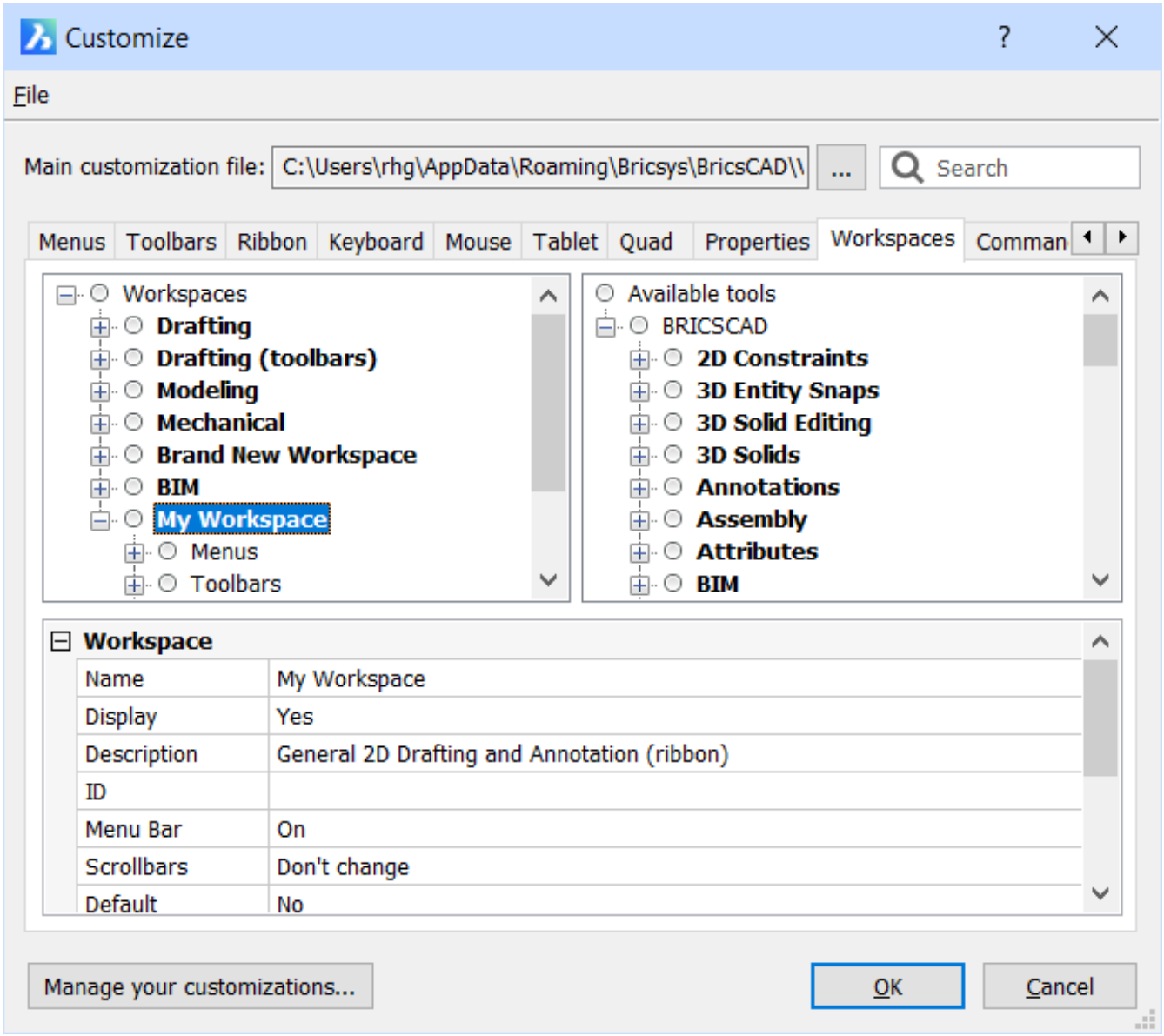
(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the Customize dialog box at the Workspaces tab.

Accessing the Command

command bar: **wssettings** : **WsSettings** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/en
dlg.png})

- Make adjustments to the workspace of your choice. See theCustomize (6.9.58)command.

Figure 6.2628



Command Options

See Customize Workspaces (1.13.10).

Related Commands

Customize (6.9.58) - customizes the user interface of BricsCAD: Menus, Toolbars, Keyboard shortcuts, Mouse actions, tablet, Quad settings, Properties, Workspace settings, Aliases, and Shell Commands.

Figure 6.2629



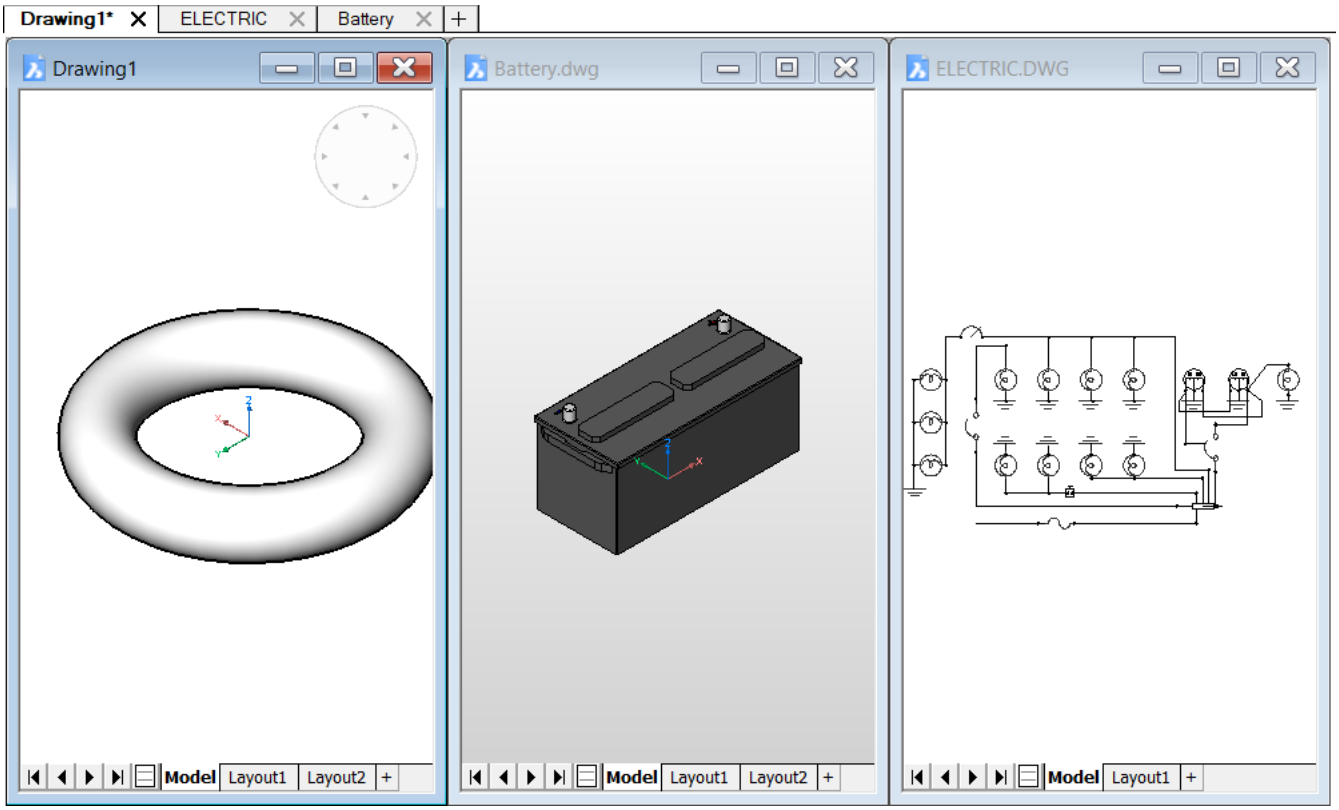
6.33.19 WvTile

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Tiles drawing windows vertically (short for ”window vertical tile”).

Accessing the Command

 command bar: **wvtile** menu bar: **Window | Tile Vertically** ribbon:**View | Windows | Tile Vertically** : **WvTile** Displays no prompt in the command bar.Windows are tiled vertically, with the most recent drawing window highlighted (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/W/wvtile-eg.png})

Figure 6.2630



Command Options

No options.

Related Commands

WCascade (6.33.3) - cascades the windows. WClose (6.33.4) - closes the current window. WCloseAll (6.33.5) - closes all windows. WhTile (6.33.9) - tiles windows horizontally. WiArrange (6.33.10) - arranges iconized windows. SysWindows (6.27.61) - arranges windows.

6.34 X

6.34.1 XAttach

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Attaches externally-referenced drawings to the current drawing, through a dialog box (short for ”external attach”). **Note:**This command is a shortcut to using the -XRef (6.34.9) command’s Attach option.

Figure 6.2631



Accessing the Command

 command bar: **xattach** transparent: **'xattach** alias: **xa** menu bar: **Insert | Attach XRef** ribbon: **Insert | External References | Attach XRef** toolbar: **Insert |**  : **XAttach** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/SelectReferenceFileDialog.png})

1. Choose a DWG file.
2. Click **Open**. The program displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/
3. Choose options.
4. Click **OK**.
5. If you choose **Specify On-screen**, then you are prompted in the command line for the insertion point, scale, and/or rotation angle.

The XDWGFADECTL (1.13.11) system variable controls the fading level for Xref inserts. Non positive values disable fading. Values between -90 and 90 are accepted.

Figure 6.2632

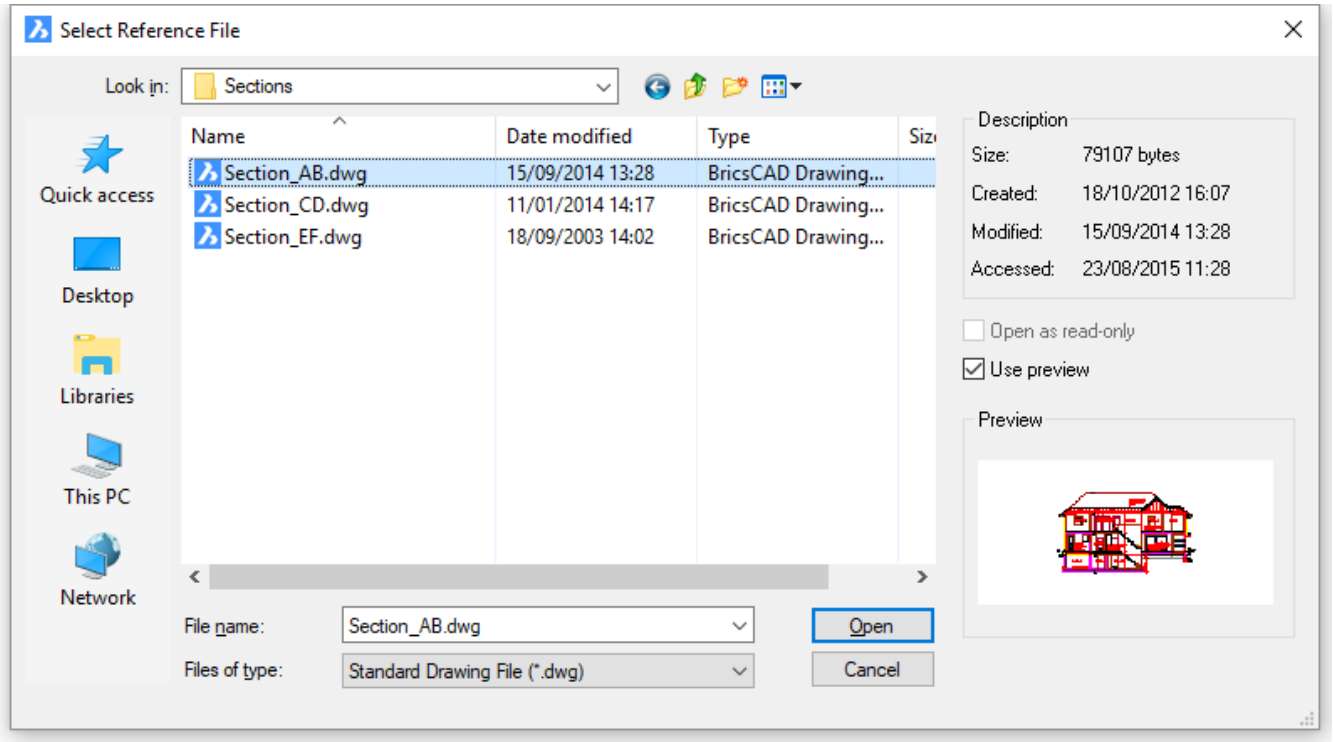
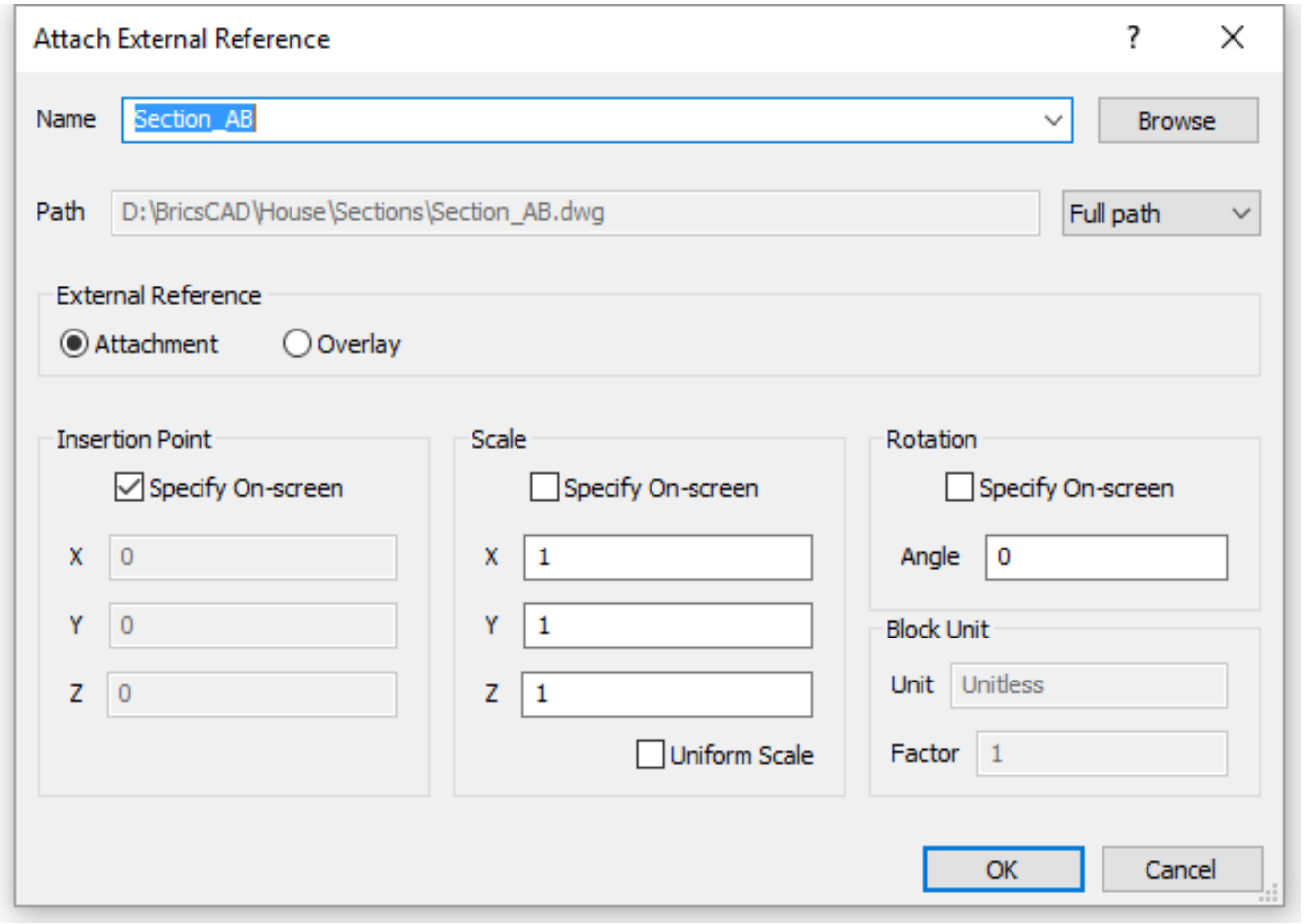


Figure 6.2633



Dialog Box Options

Option	Description
Name	Lists the names of DWG files attached to the current drawing, including the one you just selected: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i Choosing another name lets you modify that xref's properties.
Browse	Attaches additional DWG files without leaving this dialog box.Displays the Select Reference File dialog box: 1. Choose a DWG file 2. Click **Open**.
Path	Reports the path that will be remembered by BricsCAD.You cannot edit this field; instead, use the **Browse** button to modify the path to the file.
Full Path	Changes the way in which the program remembers the path to the attached DWG file: 1. Click the droplist 2. Choose a different path type (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Full path** - stores the drive and folder names of the DWG file as an absolute reference, such as**c:\cad\dwg\filename.dwg** • **Relative path** - stores the relative route to the xref'ed DWF file's folder, such as **..\filename.dwg** (..\ means one folder higher).This option does not work until the drawing has been saved at least once. • **No path** - strips the drive and folder names, leaving just the DWG file name, such as **filename.dwg**
External Reference	Specifies how the external reference is to be attached: • **Attachment** - attaches the xref and nested xrefs • **Overlay** - attaches only the first level of xref
Insertion Point	Specifies the location of the xref's lower-left corner: **Specify On-screen** - determines when the insertion point is specified: 1. **On** - specify the insertion point in the drawing after the dialog box is closed; you will be prompted: **Insertion point for block** - **(pick a point or enter x,y,z coordinates.)** 2. **Off** - specify the insertion point in the dialog box using the X, Y, and Z fields **X**,**Y**, **orZ** - specifies the x, y, and/or z coordinates for the xref's insertion point.Use 0,0,0 to insert the xref at the drawing's origin.

Scale	Specifies the size of the xref: **Specify On-screen** - determines when the scale factors are specified: • **On** - specify the scale factor in the drawing after the dialog box is closed; you will be prompted: 1. **Corner/XYZ/X scale factor** - **(enter an option, pick two points a number, or press Enter for the default value.)** 2. **Y scale factor: < Equal to X scale>** - **(enter an option, a number, or press Enter for the default value.)** • **Off** - specify the scale factor in the dialog box using the X, Y, and Z fields **X**,**Y**, **orZ** - specifies the x, y, and/or zscale factors of the xref in the x, y, and z directions.Use 1,1,1 to keep the xref at its original size. **Uniform Scale** - makes the y and z scale factors equal to that of x
Rotation	Specifies the rotation angle of the xref: **Specify On-screen** - determines when the rotation angle is specified: • **On** - specify the angle in the drawing after the dialog box is closed; you will be prompted: **Rotation angle for block** - **(enter a number, pick two points, or press Enter for the default value.)** • **Off** - specify the angle in the dialog box using the Angle field. Angle - specifies the rotation angle about the insertion point: • Positive angles rotate the xref counterclockwise • Use 0 to keep the xref at it original orientation
Block Unit	Specifies the units with which the drawing will be inserted, usually a form of Imperial or metric units, such as inches or mm: **Unit** - reports the INSUNITS setting of the drawing being attached. **Factor**- reports the calculated scaling factor with respect to the INSUNITS setting of the drawing being attached and the current drawing.
OK	Attaches the DWG file to the current drawing. If you choose **Specify On-screen**, then you are prompted in the command line for the insertion point, scale, and/or rotation angle.

Procedures

- To attach an external reference (1.22.16)

Related Commands

Insert (6.17.13) - inserts drawings as blocks. PlaceView (6.24.24) - places a model view selected from a source drawing into a paper space layout of the current drawing. XRef (6.34.8) - attaches and controls externally referenced drawings through the Drawing Explorer. -XRef (6.34.9) - attaches and controls xrefs through the command bar. XClip (6.34.2) - clips externally-referenced drawings.

Figure 6.2634

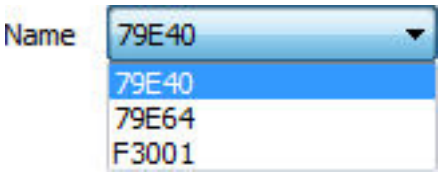


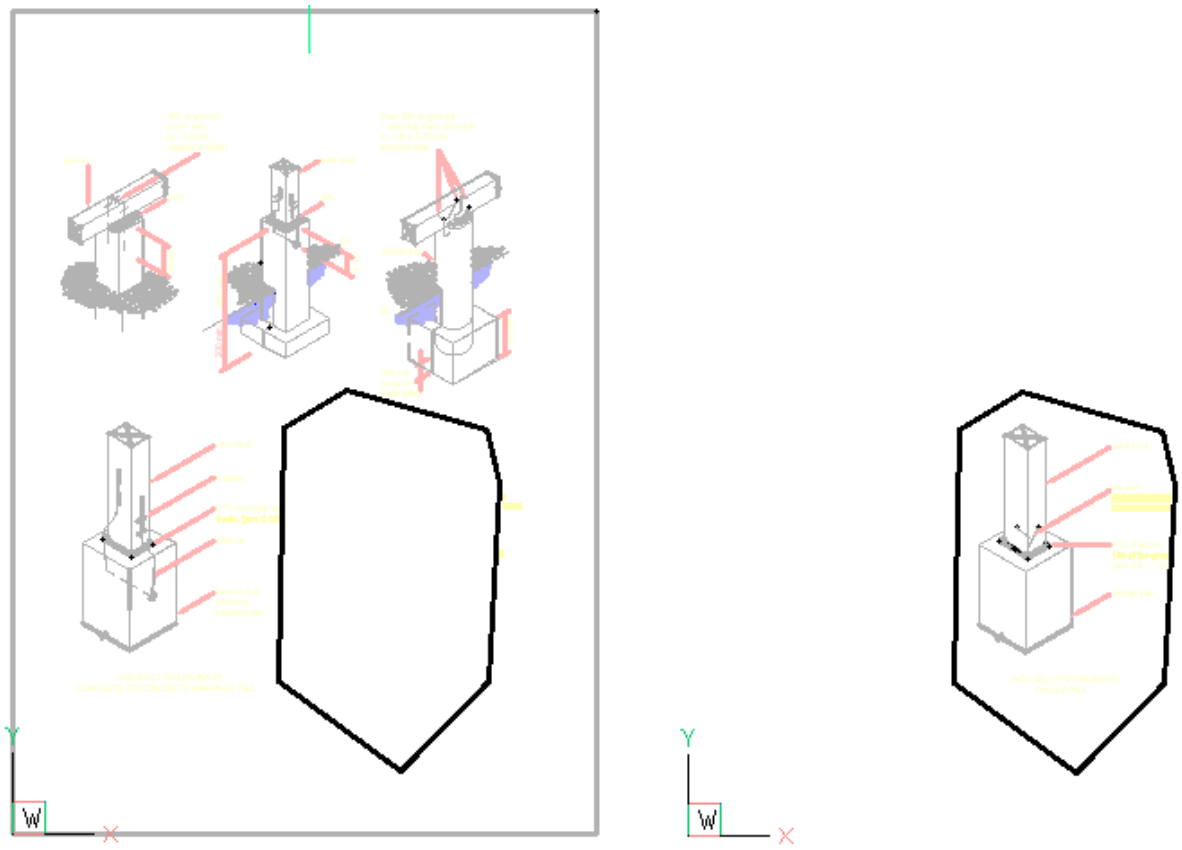
Figure 6.2635



Figure 6.2636



Figure 6.2637



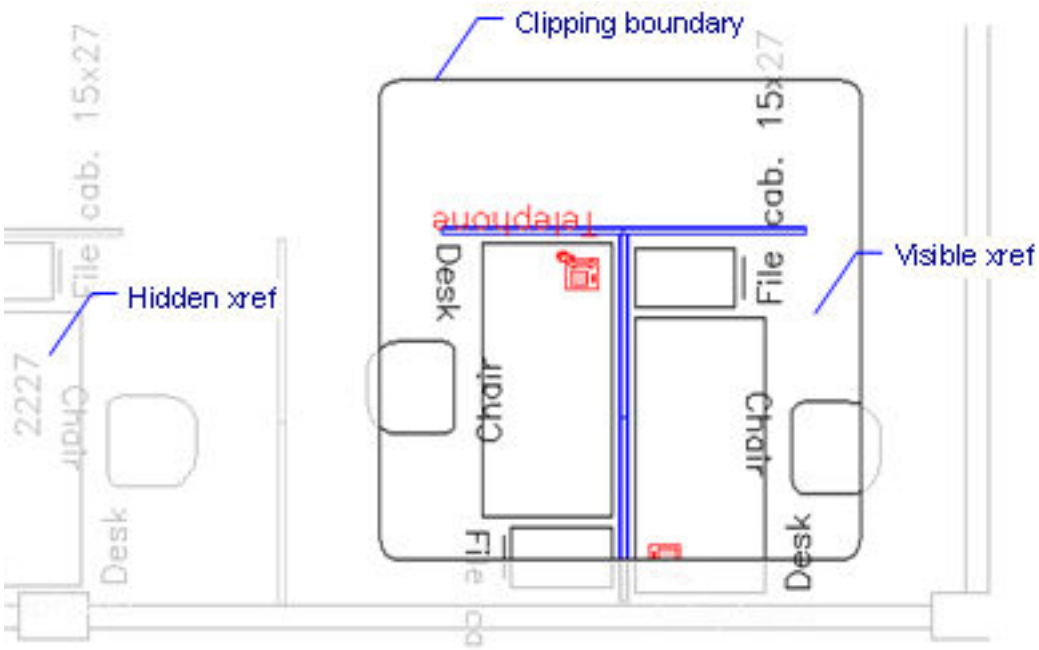
6.34.2 XClip

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Clips externally-referenced drawings with polygons to hide parts of them, and adjusts the front and rear clipping planes (short for "xref clip"). Shown below is a clipped xref (left) and an inverted clipping (right): (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/X/xclip-eg.png})

Accessing the Command

command bar: **xclip** alias: **clip** menu bar: **Modify | XRef Clip** : **XClip Prompts you in the command bar:** Select entities: **(Choose one or more xrefs)** Entities in set: 1 Select entities: **(Choose more xrefs, or press Enter to continue)** [ON/OFF/Clipdepth/Invert/Delete/generate Polyline/New boundary] <New boundary>: **(Enter an option, or else press Enter to create a new clipping boundary)** [Select polyline/Polygonal/Rectangular] <Rectangular>: **(Enter an option, or press Enter to create a rectangular clipping boundary)** In the figure below, the hidden portion of the xref is shown in light gray for illustrative purposes only; it is normally not seen when clipping is turned on. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image380.jpg})

Figure 6.2638



Command Options

Option	Description
Select entities	Chooses the xrefs to be clipped: • Choose one or more xrefs • Enter **all** to choose all xrefs in the current drawing • Press **Enter** to continue
ON	Turns on clipping boundaries that were turned off by the OFF option.
OFF	Turns off clipping so that all of the xref is again visible. The clipping is saved, and is turned on again with the ON option.

Clipdepth	Sets 3D clipping to the front and rear, to hide parts of 3D models that might obscure the front and confuse the back; prompts you: 1. **Specify front clip point or [Distance/Remove]** - **(enter an option.)** 2. **Specify back clip point or [Distance/Remove]** - **(enter an option.)** This option does not operate until you create at least one clipping boundary. It is no useful for 2D xrefs. • **Specify front or back clip point** - specifies the position of the front or back clipping plane: pick a point, or enter coordinates.The cutting plane is parallel to the current viewpoint. • **Distance** - specifies the distance of the clipping point; prompts you: **Specify distance from boundary** - **(enter the distance from the clipping boundary to the front or rear planes.)** • **Remove** - **(remove the front or rear clipping point.)**
Invert	Inverts the clipping mode so that the interior of the clipping boundary is hidden from view (the the outer area is visible); repeat this option to invert again, making the inner portion visible. It is much easier to toggle the Invert status through the Properties panel.
Delete	Erases all clipping boundaries; does not prompt or warn you.
generate Polyline	Draws a polyline over a selected clipping boundary: this allows you to edit the clipping boundary, but in an indirect fashion. Follow these steps: 1. Generate the polyline with this option, and then exit the XClip command. 2. Use the PEdit (6.24.20) command to edit the polyline. 3. Restart the XClip command, and then use the **New | Select polyline** option to turn the polyline back into a clipping boundary.

New boundary	Draws a new clipping boundary: • If a boundary exists, then you are prompted: **Delete old boundary(s)? [Yes/No]** - **(type Y or N.)Caution**! This option replaces all existing clipping boundaries with the new one. The command continuesonly when all previous boundaries are deleted: typing No at this prompt cancels the XClip command. • If no boundary exists, then you are prompted instead: **[Select polyline/Polygonal/Rectangular]** - **(type S, P, or R)** • **Select polyline** - choose any open or closed entity made from polylines, such as splined polylines, rectangles, donuts, and polygons. While the polyline does not need to be closed, it cannot cross over itself. • **Polygonal** - creates non-rectangular clipping boundaries, with a minimum of three sizes; prompts you: 1. **Specify first point**- **(pick a point in the drawing, or specify x,y coordinates.)** 2. **Specify next point or [Undo]**- **(pick another point in the drawing, or specify x,y coordinates.)** 3. **Specify next point or [Undo]**- **(pick more points, or pressEnterto complete the polygon)** • **Rectangular** - creates rectangular clipping boundaries; prompts you: 1. **Specify first corner** - **(pick a point in the drawing, or specify x,y coordinates.)** 2. **Specify opposite corner** - **(pick another point in the drawing, or specify x,y coordinates.)**

Tutorial: Clipping x-refs and blocks

- Click here (1.3.17) to watch

Related Commands

pdfclip (6.24.16) - clips a PDF underlay. XRef (6.34.8) - attaches and controls externally referenced drawings through the Drawing Explorer. -XRef (6.34.9)- attaches and controls externally referenced drawings through the command bar. VpClip (6.31.17) - clips viewports. ImageClip (6.17.7) - clips images.

Figure 6.2639



6.34.3 XEdges

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Generates 2D entities from the edges of 3D solids, 3D flat faces, and regions. **Note:** This command does not extract entities from curved faces, such as a sphere or torus.

Accessing the Command

 command bar: **xedges** menu bar: **Draw | Boundaries | Extract Edges** ribbon: **Solid | Draw | Boundary | Extract Edges** toolbar: **Draw | Boundaries** | Quad: **Draw** | : **XEdges Prompts you in the command bar:** Select entities: **(Select one or more 3D solids, 3D solid faces, or a regions)** Entities in set: 1 Select entities: **(Select more entities, or else press Enter to end the command)** The result are 2D entities drawn in 3D space, such aslines, arcs, circles, ellipses, elliptical arcs, and splines.The entities are placed on the current layer.

Command Options

Option	Description
Select entities	Select one or more of the following entities: 3D solid, 3D solid face, and region: • Pick one of more 3D entities • Type **all** to select all entities in the drawing; the program filters out those that are not eligible for edge extraction • Press **Enter** to end the command

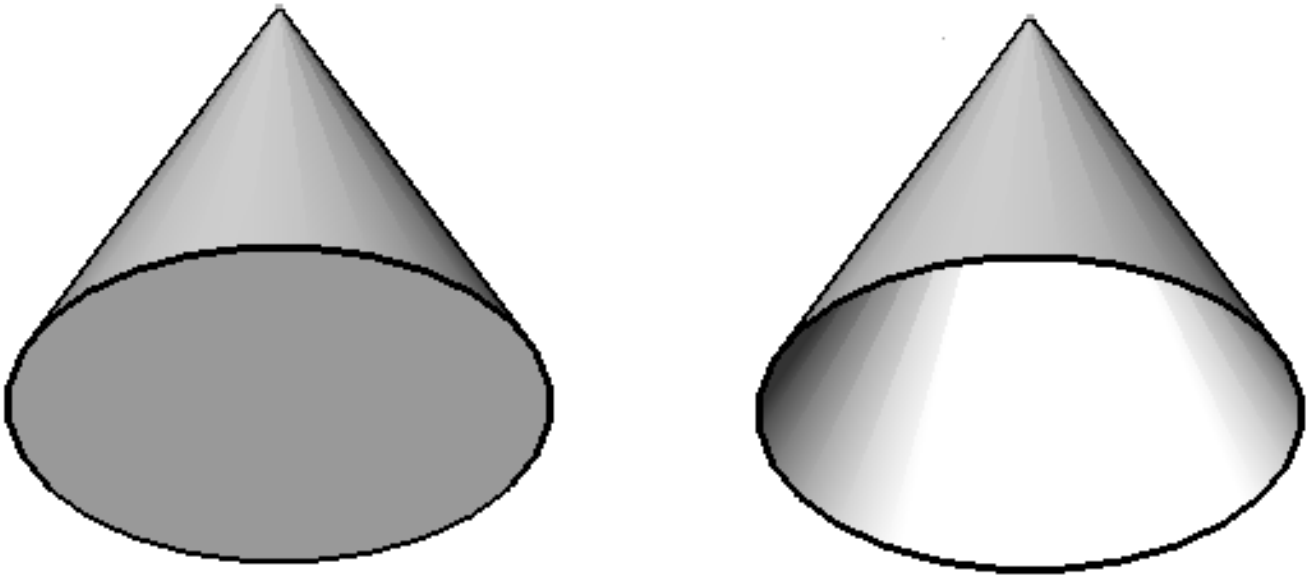
Related Commands

Boundary (6.6.15) - creates a polyline that forms a boundary inside a closed area through a dialog box. -Boundary (6.6.16) - creates a polyline boundary inside closed areas through the command bar. Imprint (6.17.12): imprints 2D entities onto planar faces of 3D solids and surfaces; allows to create additional edges on planar faces. ProjectGeometry (6.24.48) - projects 2D geometry onto a region, a surface or a 3D solid; allows to create additional linework on the faces of solids and surfaces. XFaces (6.34.4) - extracts a set of 3D solid or surface faces as a new 3D surface object.

Figure 6.2640



Figure 6.2641



6.34.4 XFaces

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-shape-pro-platinum-mechanical-bim.png}) Generates 3D surface faces from the faces of 3D solids and 3D surface faces. **Note:** This command works with any kind of solid and surface face. Below, the original 3D solid (at left) and the 3D surface face extracted from it (right). (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/X/xfaces-eg.png})

Accessing the Command

 command bar: **xfaces** transparent: **'xfaces** menu: **Model | Direct Modeling | Extract Faces** toolbar: **Direct Modeling |**  Quad: **Draw |**  : **XFaces Prompts you in the command bar:** Select faces to extract [Move]: **(Select faces*.)** Select faces to extract [Move]: **(Select more faces or press Enter to extract the current selection set.)** The newly created 3D surfaces are placed on the current layer.

Command Options

Option	Description
Select faces to extract	Select one or more faces of 3D solids and 3D surfaces: • Pick one of more faces • Type **all** to select all entities in the drawing; the program filters out those that are not eligible for edge extraction • Press **Enter** to end the command

Move	Moves the newly created 3D surface entities; prompts you: 1. **Enter base point [Displacement] <Displacement>- (specify a point from which to move the entities.)** 2. **Enter second point <Use base point as displacement>- (specify another point at which to place the entities.)**

Related Commands

XEdges (6.34.3) - creates 2D linear entities (lines, arcs, circles, ellipses, elliptical arcs and splines) from the edges of 3D solids, 3D solid faces, and regions.

Figure 6.2642



Figure 6.2643



6.34.5 XLine

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Draws infinitely long lines. **Note:** Xlines have no starting point and no ending point as both ends stretch to infinity, and are useful as construction lines; they areignored when the program determines the extents of the drawing. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image691.png})

Accessing the Command



command bar: **xline** aliases:**xl** **infline** menu bar: **Draw | Infinite Line** ribbon: **Draw | Draw | Line | Construction Line** toolbar: **Draw | Lines |**  : **XLine Prompts you in the command bar:** Infinite line: Set point along line or [Horizontal/Vertical/Angle/Bisect/Parallel]: **(Pick a point, or else enter an option)** Direction: **(Pick a point, or enter x,y coordinates)** Direction: **(Pick another point, or else press Enter to exit command)**

Command Options

Option	Description
Point along line	Specifies a point on the xline: pick a point in the drawing, or enter x,y coordinates; prompts you: **Direction** - **(specify the direction of the xline.)** The Direction prompt repeats so that you can draw multiple xline from the same starting point.Press Enter to exit the command.

Bisect	Draws an xline that bisects entities perpendicularly; prompts you: **Entity/<Vertex point>** - **(enter an option:)** • **Entity** - draws xlines perpendicular to the midpoint of the selected line or polyline segment, and the center point of arcs and polyarcs; prompts you: **Select line,arc, or polyline segment to bisect** - **(select a line, an arc, a polyline segment, or a polyarc:)** – When you select a line or polyline segment, the command draws the xline perpendicular to the segment's midpoint: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRe – When you select an arc or polyarc, the command draws the xline perpendicular to the arc's center point and through its midpoint: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i This option works with splined polylines, but not with spline entities. • **Vertex point** - draws xlines that start at the vertex point and bisect two pick points; prompts you: 1. **Bisection angle start point** - **(specify one part of the bisection angle.)** 2. **Bisection angle start point** - **(specify the other part of the bisection angle.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Horizontal	Draws xlines horizontal to the current UCS; prompts you: **Location** - **(specify the horizontal location of the xline; pick a point in the drawing, or enter x,y coordinates. The 'Location' prompt repeats for drawing multiple horizontal xlines at different starting points.Press Enter to exit the command.)**
Vertical	Draws xlines vertical to the current UCS; prompts you: **Location** - **(specify the vertical location of the xline; pick a point in the drawing, or enter x,y coordinates.The 'Location' prompt repeats for drawing multiple vertical xlines at different starting points.Press Enter to exit the command.)**

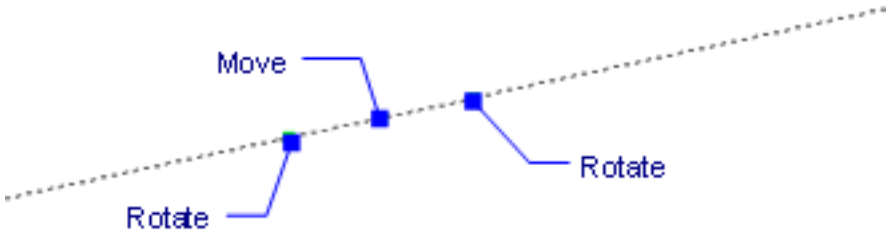
Angle	Draws xlines at angles; prompts you: **Reference/<Enter angle (0)> - (enter an option:)** • **Reference** - Draws xlines at an angle relative to another entity; prompts you: 1. **Select reference entity** - **(choose a line thatprovides the reference (base) angle.)** 2. **Enter angle** - **(specify the angle, measured relative to the entity.)** 3. **Location** - **(specify the angle's start point; need not coincide with the entity's end points.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRe The 'Location' prompt repeats for drawing multiple infinite lines at the same angle. • **Enter angle -** specifies the angle; enter an angle or pick two points to indicate the angle; prompts you: **Location** - **(specify the starting point of the xline.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/so The 'Location' prompt repeats for drawing multiple infinite lines at the same angle.
Parallel	Draws xlines parallel to lines and polyline segments; prompts you: **Through point/<Offset distance for parallel infinite line> (Through)** - **(enter an option:)** • **Through point** - specifies the point through which to draw the xline; prompts you: 1. **Select entity for parallel infinite line** - **(choose the entity to offset with a xline.)** 2. **Through the point** - **(specify the point through which to draw the xline; this point also specifies the xline's start point.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i This option works only with line segments. • **Offset distance for parallel infinite line** - specifies the offset distance: enter a value or pick two points; prompts you: 1. **Select entity for parallel infinite line** - **(choose the entity to offset with an xline.)** 2. **Side for parallel infinite line** - **(specify the side on which to place the xline.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i The xline's start point is parallel to the line's start point.The 'Select entity for parallel infinite line' prompt repeats for drawing multiple xlines at different offset distances.

Grips Editing

XLines can be edited directly through grips:

1. Select the xline. Notice that it has three grips.

Figure 6.2644



2. Drag the grips to perform the following actions:
   - Drag the midpoint grip to move the xline.
   - Drag the other grips to rotate the xline.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image766.png})

Procedures

- Drawing infinite lines (1.17.11)

Related Commands

Ray (6.26.1) - draws semi-infinite lines. Properties (6.24.49) - modifies the properties of rays and xlines.

Figure 6.2645

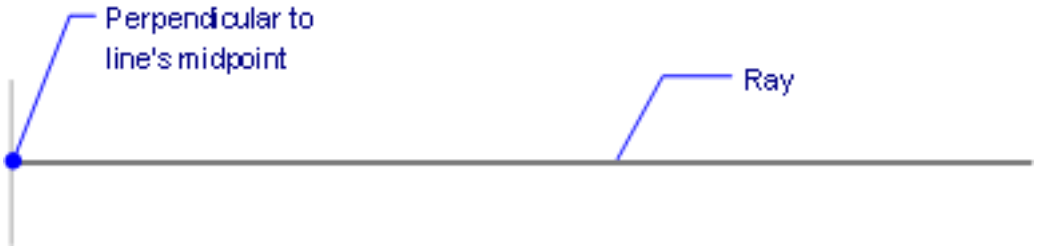


Figure 6.2646

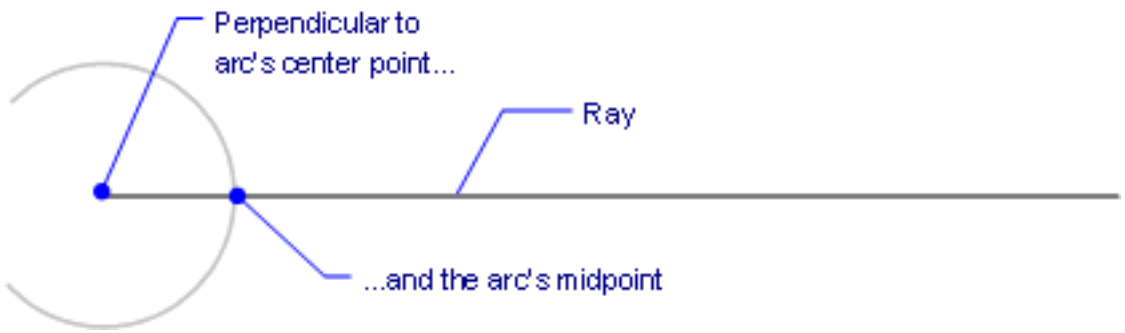


Figure 6.2647

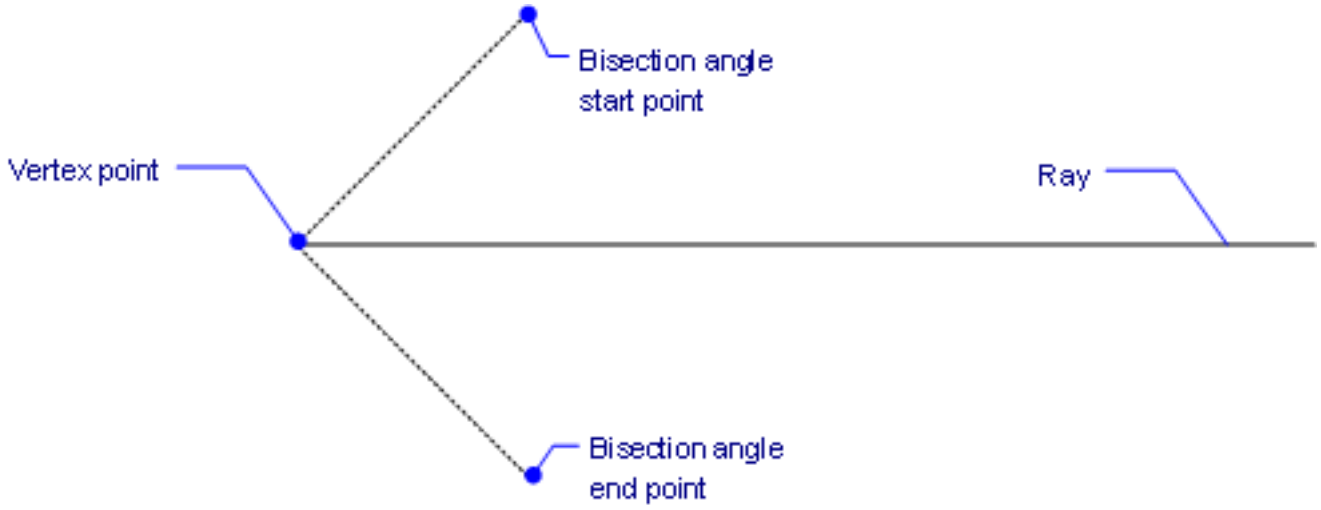


Figure 6.2648

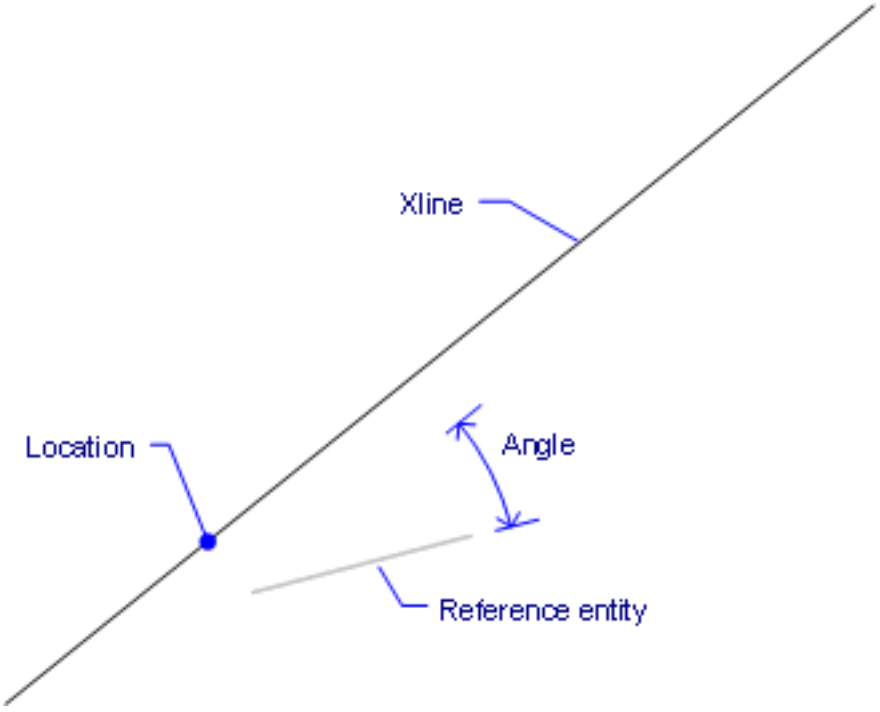


Figure 6.2649

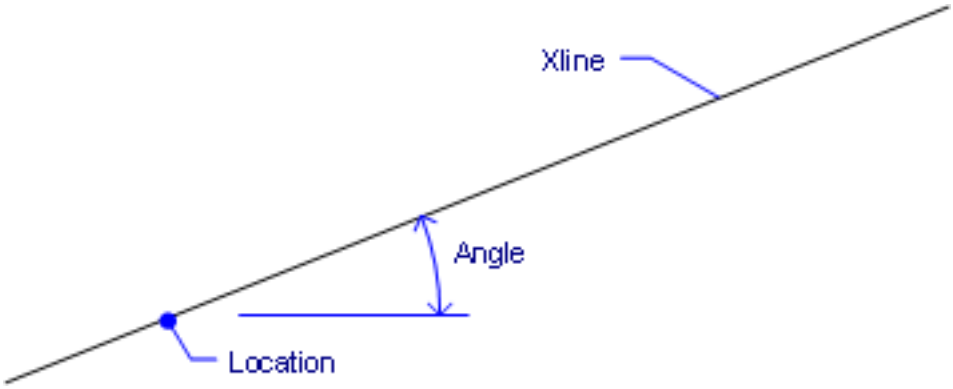


Figure 6.2650

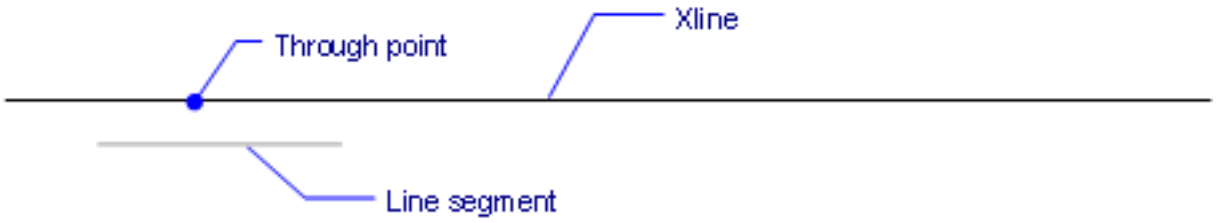


Figure 6.2651

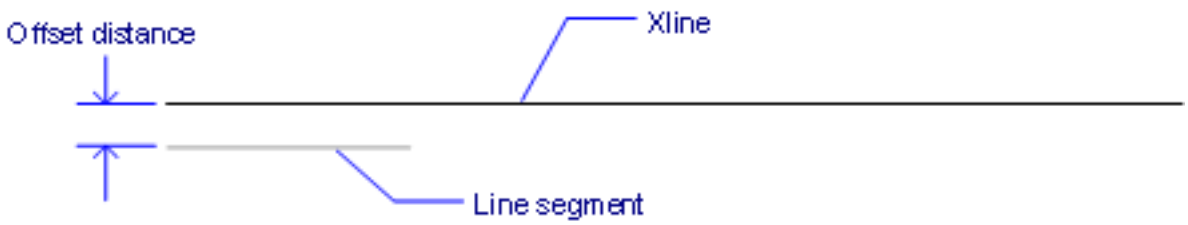


Figure 6.2652



6.34.6 XOpen

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens an externally-referenced drawing (xXref) attached to the current drawing, for editing in a separate window.

Accessing the Command

 command bar: **xopen** menu: **Xref context menu | Open Xref** ribbon: **Insert | External References | Open XRef** toolbar: **Drawing Explorer - External References |**  : **XOpen Prompts you in command bar:** Select reference: **(Click an externally-referenced drawing in the current drawing)**

1. The drawing opens in a new drawing tab.
2. When done editing, save the drawing.
3. Close the tab.

Command Options

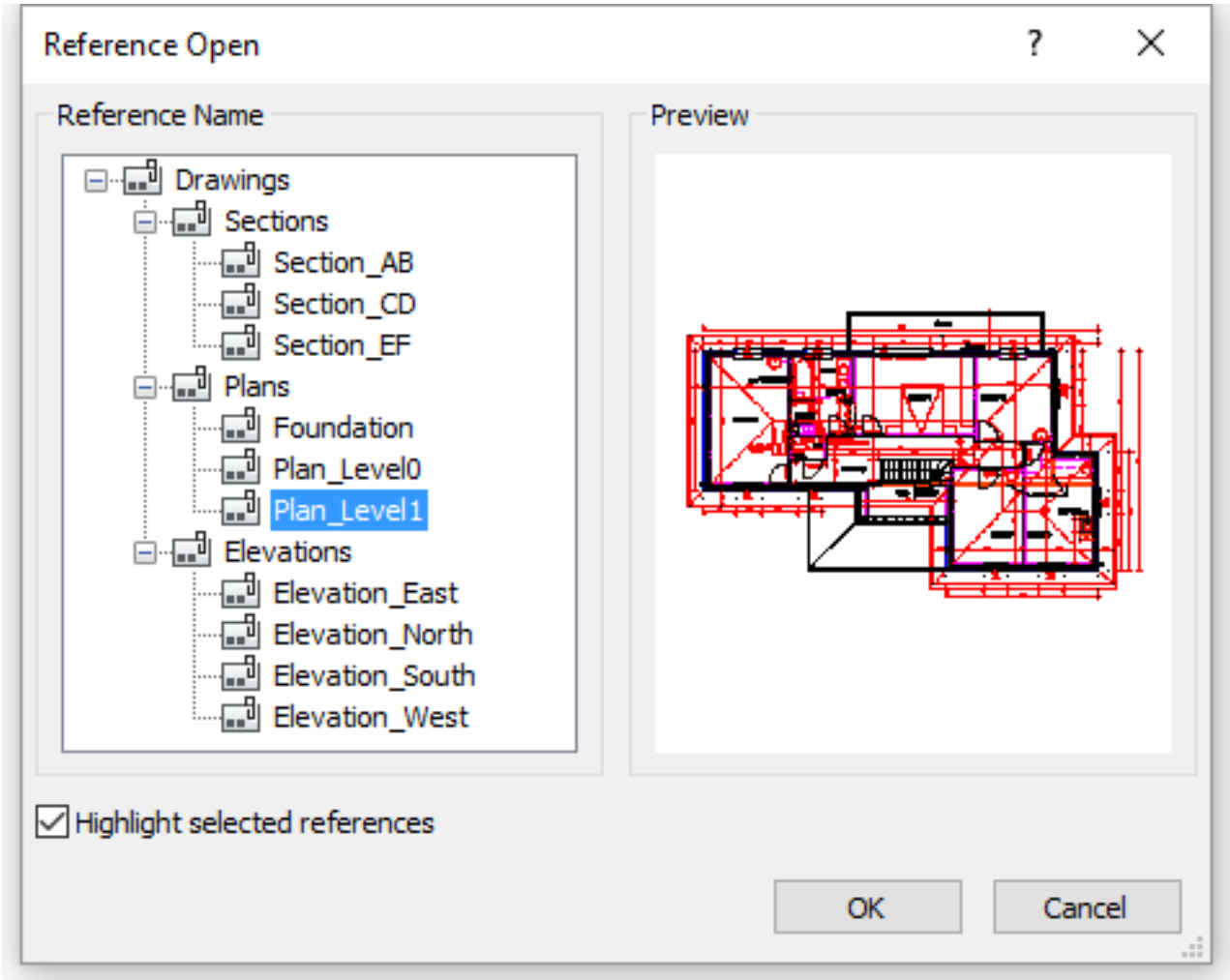
Option	Description
Select reference	Click an externally-referenced drawing in the current drawing

Dialog Box Options

When the selected externally-referenced drawing contains nested references a dialog box displays to let you choose the drawing to be opened: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/Reference

Dialog Box Option	Description
Reference Name	Select an xref to open for editing
Highlight Selected References	Toggles the highlighting of xrefs in the drawing: • **On** - the xref you select in this dialog box is highlighted in the drawing • **Off** - the xref is not highlighted
OK	Closes the dialog box and opens the selected xref in a new drawing tab

Figure 6.2653



Related Commands

Refedit (6.26.12) - edits block references and externally-referenced drawings through a dialog box. BmOpen (6.8.17) - opens the source drawing of an external mechanical component.

Figure 6.2654



6.34.7 Xplode

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Explodes entities, providing control over the resulting entities (short for "explode"). **Note:**This command does not explode attached references, entities on frozen layers, or basic entities, such as lines, circles, and arcs.

Accessing the Command

command bar: **xplode** transparent: **'xplode** : **Xplode Prompts you in the command bar:** Select entities to explode: **(Choose one or more entities, but usually you wan to explode one at a time)** Entities in set: 1 Select entities to explode: **(Choose more entities, or else press Enter to continue)** Xplode entities [Separately/All] <All>: **(Enter an option)** [All listed/Multiple/LAyer/Color/LType/ltScale/Inherit all from parent] <Xplode>: **(Enter an option)**

Command Options

Option	Description
Select entities to explode	Selects the entities to explode. Choose one or more entities; use any selection method. Entities on frozen layers are not selected and not frozen.
Separately	Explodes one entity at a time; the prompts repeat themselves for each entity.
All	Explodes all selected entities together.
All listed	Allows to specify all properties for the selected entities.
Multiple	Specifies more than one property for the exploded entities; prompts you: **[LAyer/Color/LType/ltScale/Exit] <Xplode>** - **(choose properties one by one. For instance, whenyou want to specify the color and the linetype scale:)** 1. **[LAyer/Color/LType/ltScale/Exit] <Xplode>** - **(pressCandthe program responds:) Altering: Color** 2. **[LAyer/Color/LType/ltScale/Exit] <Xplode>** - **(type LT,andthe program responds:) Altering: Color LTScale** 3. **[LAyer/Color/LType/ltScale/Exit] <Xplode>** - **(press Enter to proceed;andthe program responds:)** 4. **Color for drawing new entities <BYLAYER>** - **(specify a color name or number.)** 5. **Linetype scale for new entities <1>** - **(specify the linetype scale factor.)**
Color	Specifies the color for the exploded entities; prompts you: **Color for new entities** - **(specify a color name or number.)**
LAyer	Specifies the layer for the exploded entities; prompts you: **Layer for new entities** - **(specify the name.)** The layer name must exist in the drawing.
LType	Specifies the linetype for the exploded entities; prompts you: **Linetype for new entities** - **(specify the name.)** The linetype name must exist in the drawing.

ltScale	Specifies the linetype scale for the exploded entities; prompts you: **Linetype scale for new entities - (specify the scale factor.)**
Inherit all from parent	Inherits the color, layer, and linetype scale from the source entities.
Xplode	Explodes the selected entities.

Related Commands

Explode (6.13.13) - explodes complex entities into simpler ones. TxtExp (6.29.27) - explodes text into polylines. U (6.30.1) - undoes the effect of the explosion.

Figure 6.2655



6.34.8 XRef

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) A command that combines several xref-related commands through the Drawing Explorer; attaches DWG files to the current drawing and controls the state of the attachments (short for "external reference").

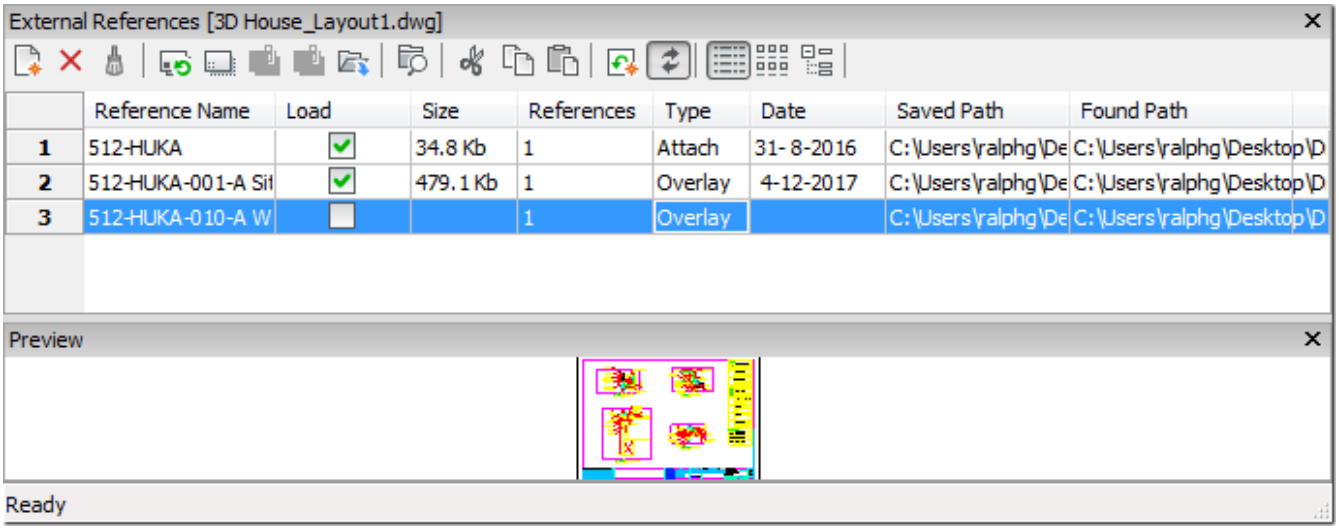
Accessing the Command

command bar: **xref** alias: **xr xa xattach expxrefs** menu bar: **Settings | XRefs** ribbon:**Home | Settings | Drawing Explorer | Xrefs Insert | External References | XRefs** : **XRef** Displays the External Reference section of the Drawing Explorer: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/X/xref-dlg.png})

1. Click **New** to attach a DWG file, or change the properties of existing xrefs.
2. Click **X** to close the dialog box.

Press **F1** to access help. The XDWGFADECTL (1.13.11) system variable sets the fading of external references. Values between 0 (no fading) and 90 are accepted.

Figure 6.2656



Toolbar Options

Option	Description
Attach Xref	Attaches DWG files as xrefs; prompts you in the command bar: **External References: ? to list/Bind/Insert/Detach/Path/Reload/Unload/Overlay/<Attach> - (enter an option.)** See the-XRef (6.34.9) command.
Detach Xref	Erases the selected xref from the drawing without warning. This is equivalent to using the **Detach** option.
Reload Xref	Reloads the selected xref. This is useful when the original DWG file has changed, and you want to see the updated version in your drawing.
Unload Xref	Unloads the selected xref. This hides the xref from view.You can also click the checkboxes in the Loaded column.
Bind Xref	Binds the xref to the current drawing to make the xref part of the drawing. References to the xref disappear from the Drawing Explorer. (This option is not available for xrefs that are unloaded.) This option sets the BINDTYPE settings variable OFF (Traditional binding behavior), thus influencing the behavior of the Bind (6.34.9) option of the -xref command.
Insert Xref	Converts xrefs into blocks.This is like using the Insert (6.17.13)command to insert external DWG files in drawings. References to the xref disappear from the Drawing Explorer. (This option is not available for xrefs that are unloaded.)
Open Xref	Opens the xref for editing. See the XOpen (6.34.6) command.
Detail View	Displays the details of xrefs: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Icon View	Displays the xrefs as icons: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Tree View	Displays the relationship between xrefs in a tree-like view: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i • **Current** - the current drawing • **Column and Floor** - xrefs attached to the current drawing

External Reference Options

Option	Description
Reference Name	Specifies the name of the xref. This name is assigned by BricsCAD, but can be changed by the user. Click to rename.

Loaded	Toggles the loading of the xref; click to change the load status: • **On** - loads the xref, and makes it visible • **Off** - unloads the xref, making it invisible
Size	Reports the size of the xref DWG files. Very large files can slow down the system.
References	Reports the number of times the xref is attached to the drawing.
Type	Reports whether the xref is attached or overlaid: • **Attach** - all xrefs are displayed. • **Overlay** - only the first xref of nested xrefs is displayed. To change type: 1. Unload the xref 2. Reload it as attached or overlaid Overlay and Attach operate the same way. The difference, however, is in how they treat xrefs that have xrefs themselves" • **Overlay**- attaches only the first level of xref • **Attach**- attaches all xrefs
Date	Reports the date of the file. This can be useful in determining if you ware working with the most recent revision.
Saved Path	Reports the original path to the xref's DWG file when first loaded: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i If the path cannot be found, click this field, and then click the  button. BricsCAD displays the Choose a File dialog box: 1. Locate the missing DWG file 2. Click **Open** To convert the Xref path type, do the following: 1. Right click the Xref name 2. Choose the path type in the context menu: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou or (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/sou
Found Path	Reports the current path to the xref's DWG file. This path should match the Saved Path in most cases.

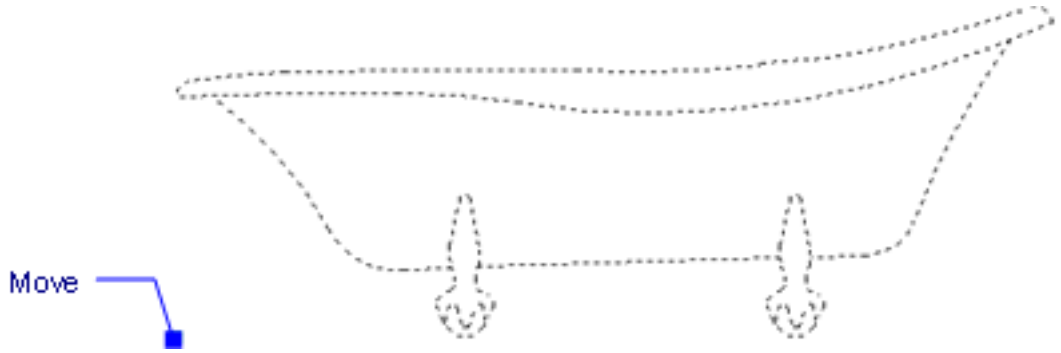
Grips Editing

External references can be edited directly through grips:

1. Select the xref. Notice that it has one grip at its insertion point.
2. Drag the grip to move the xref.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image765.png})

Figure 6.2657



Procedures

- Managing Xrefs (1.22.17)
- To detach external references (1.22.17)
- To unload external references (1.22.17)
- To reload external references (1.22.17)

Related Commands

Insert (6.17.13) - inserts drawings as blocks. Explorer (6.13.14) - opens the Drawing Explorer dialog box PdfAttach (6.24.14) - inserts a PDF file as an underlay into the current drawing. -PdfAttach (6.24.15) - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar. Pdfclip (6.24.16) - allows clipping of PDF underlays. Pdflayers (6.24.18) - allows to control the display of layers in a PDF underlay. Pdf (6.24.12) - attaches PDF files to the current drawing through the Drawing Explorer. PlaceView (6.24.24) - places a model view selected from a source drawing into a paper space layout of the current drawing. -XRef (6.34.9)- attaches and controls externally referenced drawings through the command bar. XAttach (6.34.1) - attaches xrefs through dialog boxes. XClip (6.34.2) - clips externally-referenced drawings. VpClip (6.31.17) - clips viewports.

Figure 6.2658

	Reference Name	Loaded	Size	References	Type	Date	Saved Path	Found Path
1	COLUMN	☑	14.0 Kb	1	Attach	22-12-1998	\\Ralph\d\CAD\dwg\LTSAM	\\Ralph\d\CAD\dwg\LTS
2	FLOOR	☑	189.5 Kb	1	Attach	22-12-1998	\\Ralph\d\CAD\dwg\LTSAM	\\Ralph\d\CAD\dwg\LTS

Figure 6.2659



Figure 6.2660

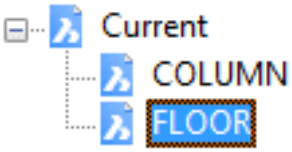


Figure 6.2661

Saved Path	Found Path
\\Ralph\d\CAD\dwg\LTSAM	\\Ralph\d\CAD\dwg\LTS
xref4	(! Not found)

Figure 6.2662

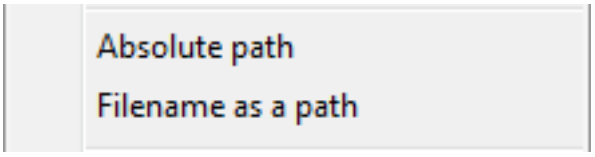


Figure 6.2663

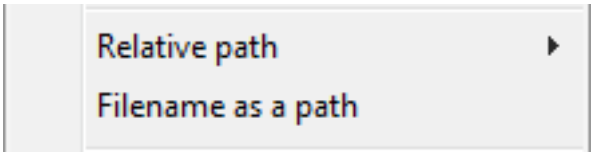


Figure 6.2664



6.34.9 -XRef

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Attaches DWG files to the current drawing through the command bar (short for "external reference").

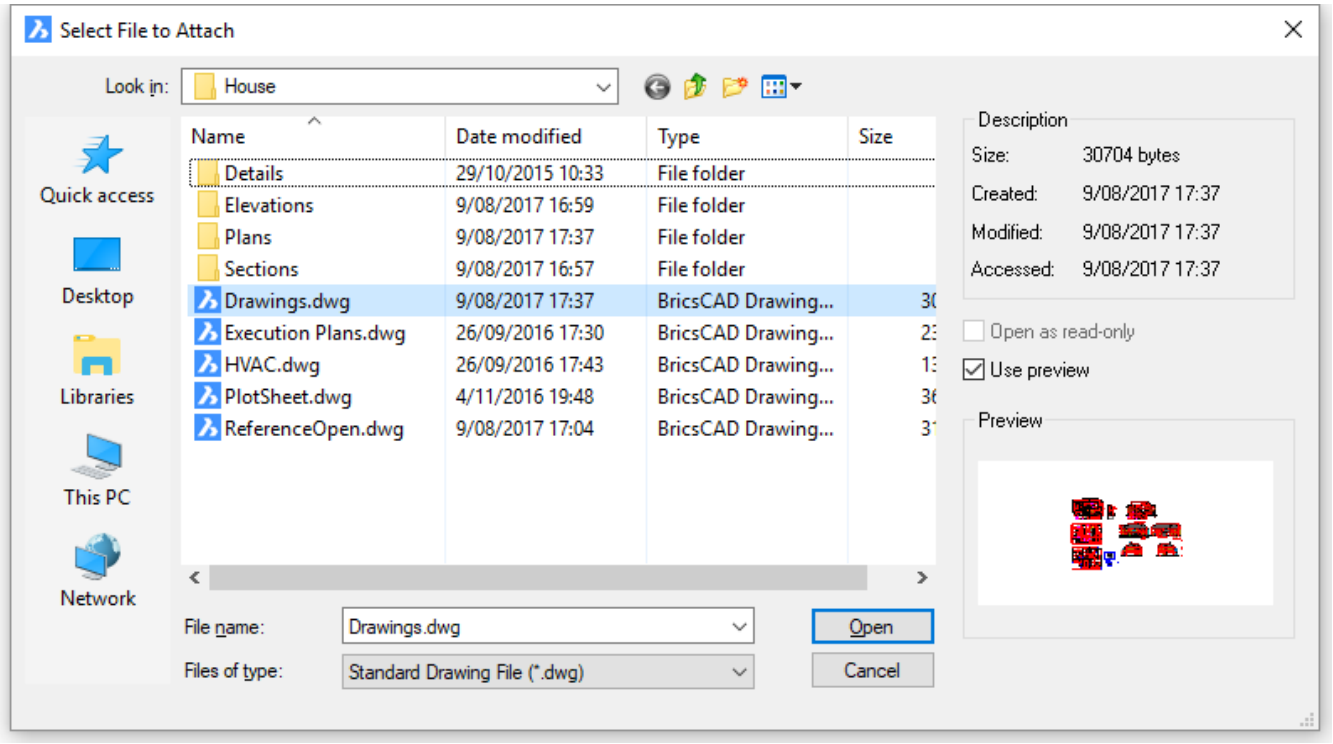
Accessing the Command

command bar: **-xref** alias: **-xr** : **-XRef Prompts you in the command bar:** External References [? to list external references/ Attach/ Overlay/ Bind/ Detach/ Path /Reload/ Unload] <Attach>: **(Enter an option)** Displays a dialog box: (\vref {fig:C:/Dev/Task-2448/images/wwwbricsyscom/bricscad/help/enUS/V18/CmdRef/source/SelectFileToAttachDialog.p

1. Choose a DWG file
2. Click Open.The program prompts you in the command bar (see -Insert (6.17.14) command):
3. Insertion point for block: **(Pick a point, or enter x,y coordinates)**
4. Corner/XYZ/X scale factor <1.000000>: **(Enter an option, a number, or press Enter for the default value)**
5. Y scale factor: < Equal to X scale (1.000000)>: **(Enter an option, a number, or press Enter for the default value)**
6. Rotation angle for block <0>: **(Enter a number, or press Enter for the default value)**

TheXDWGFADECTL (1.13.11)system variable sets the fading of external references. Values between 0 (no fading) and 90 are accepted.

Figure 6.2665



Command Options

Option	Description
? to list	Lists the names of all blocks in the current drawing. Prompts you: **Xref(s) to list <* for all>** - **(press Enter to list xrefs already attached, or enter part of a name to view some xrefs.)** Sample report: `Xref name Xref Type Path ---------- -------` If necessary, press **F2** to view the full list. See TextScr (6.29.13) command.
Bind	You are prompted differently, depending on the setting of the BINDTYPE variable: • If BINDTYPE = on(Insert-like behavior): Converts xrefs into blocks; prompts you: **Externally referenced drawing(s) to insert** - **(enter the names of one or more xrefs; separate multiple names with commas.)** • If BINDTYPE = off(traditional binding behavior): Binds the xref; makes the xref part of the drawing;prompts you: **Externally referenced drawing(s) to bind** - **(enter the names of one or more xrefs; separate multiple names with commas.)**
Insert	Converts xrefs into blocks; prompts you: **Externally referenced drawing(s) to insert** - **(enter the names of one or more xrefs; separate multiple names with commas.)** This option is hidden, but you can type **i** or **insert** at the command prompt.

Detach	Detaches xrefs from the drawing; prompts you: **Externally referenced drawing(s) to detach** - **(enter the names of one or more xrefs; sSeparate multiple names with commas.)** Detached xrefs are removed from the drawing, and are no longer displayed.Use the Attach option to reattach them.
Path	Changes the path to the xref's source DWG file; prompts you: 1. **Externally referenced drawing(s) to edit path** - **(enter the names of one or more xrefs; separate multiple names with commas;)** 2. Displays the New Location for XREF dialog box: choose a new path 3. Click **Open**.The New Location for XREF dialog box repeats for each xref named. When the path is incorrect, the program cannot display the xref, so use this option when the source DWG file is located in a different folder or on another computer.
Reload	Reloads the xrefs from their source DWG files; prompts you: **Externally referenced drawing(s) to reload** - **(enter the names of one or more xrefs; separate multiple names with commas.)** The program reports whether the DWG file has changed since the xref was last loaded: `-- Reload Xref XREF1: d:\CAD \XREF 1.DWG -- XREF1 has n` Use this option when the source DWG file has changed, and you want to see the updated drawing.
Unload	Unloads the xrefs from the drawing; prompts you: **Externally referenced drawing(s) to unload** - (**enter the names of one or more xrefs;separate multiple names with commas.)**BricsCAD reports the xrefs that were unloaded: `-- Xref XREF2: d:\CAD \XREF 2.DWG -- XREF2 is unloaded` This option keeps the links to the source DWG files, and hides the display of xrefs in the drawing.
Overlay	Overlays xrefs in the current drawing; displays the Select File to Overlay dialog box. 1. Choose a DWG file 2. Click **Open**. BricsCAD prompts you in the command bar: 3. **Insertion point for block** - **(pick a point, or enter x,y coordinates.)** 4. **Corner/XYZ/X scale factor** - **(enter an option, a number, or press Enter for the default value.)** 5. **Y scale factor: <Equal to X scale>** - **(enter an option, a number, or press Enter for the default value.)** 6. **Rotation angle for block** - **(enter a number, or press Enter for the default value.)** See theInsert (6.17.13) command.

Attach	Attaches xrefs to the current drawing; displays the Select File to Attach dialog box. 1. Choose a DWG file 2. Click **Open**. BricsCAD prompts you in the command bar: 3. **Insertion point for block** - **(pick a point, or enter x,y coordinates.)** 4. **Corner/XYZ/X scale factor** - **(enter an option, a number, or press Enter for the default value.)** 5. **Y scale factor: <Equal to X scale>** - **(enter an option, a number, or press Enter for the default value.)** 6. **Rotation angle for block** - **(enter a number, or press Enter for the default value.)** See theInsert (6.17.13) command.

Overlay and Attach operate the same way. The difference is in how they treat xrefs that have xrefs themselves (nested xrefs).

- **Overlay**-nattaches only the first level of xref, nested xrefs are not attached.
- **Attach**- attaches all xrefs, including nested xrefs.

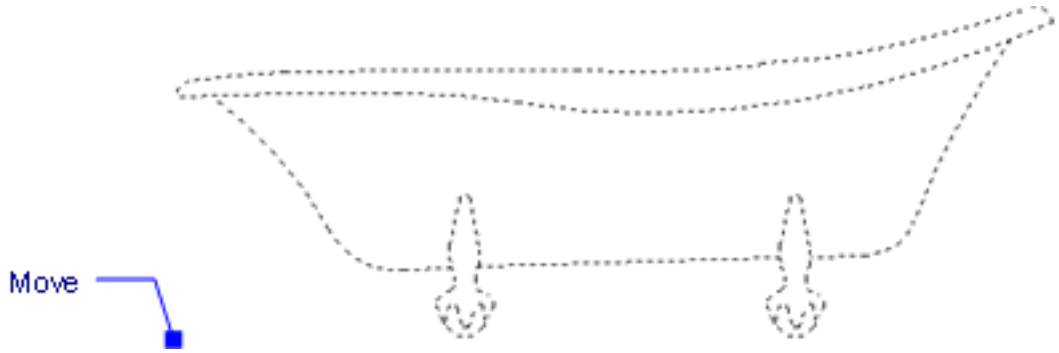
Grips Editing

External references can be edited directly through grips:

1. Select the xref. Notice that it has one grip at its insertion point.
2. Drag the grip to move the xref.

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/image765.png})

Figure 6.2666



Related Commands

Pdfattach (6.24.14) - inserts a PDF file as an underlay into the current drawing. -PdfAttach (6.24.15) - inserts a PDF file as an underlay into the current drawing; displays prompts in the command bar. XRef (6.34.8) - attaches and controls externally referenced drawings through the Drawing Explorer. XAttach (6.34.1) - attaches xrefs through a dialog box. XClip (6.34.2) - clips externally-referenced drawings. Insert (6.17.13) - inserts drawings as blocks.

6.35 Z

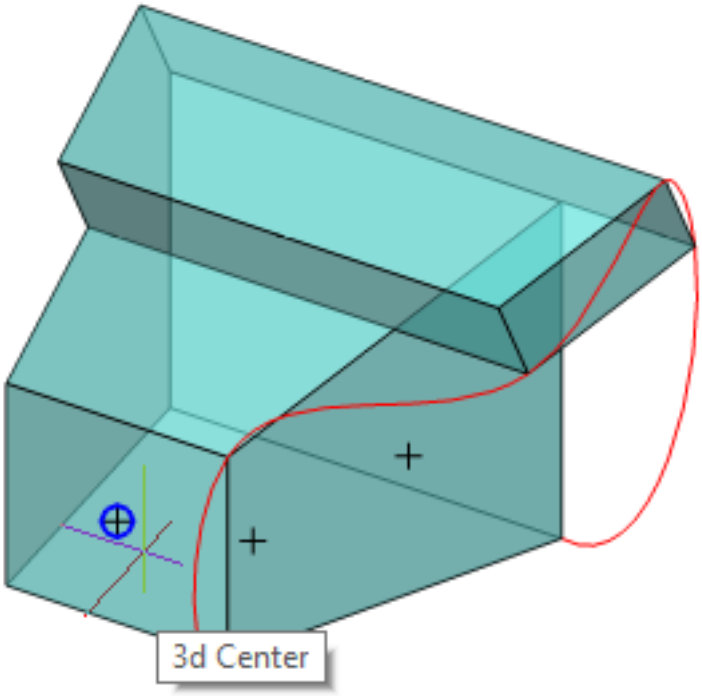
6.35.1 ZCenter

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the 3D center entity snap, which snaps to the centers of 3D faces. **Note:** The GCenter (6.15.4) entity snap also snaps to the centers of faces. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/hel

Figure 6.2667



Figure 6.2668



Accessing the Command

command bar: **zcenter** transparent: **zce** toolbar: **3D Entity Snaps** |

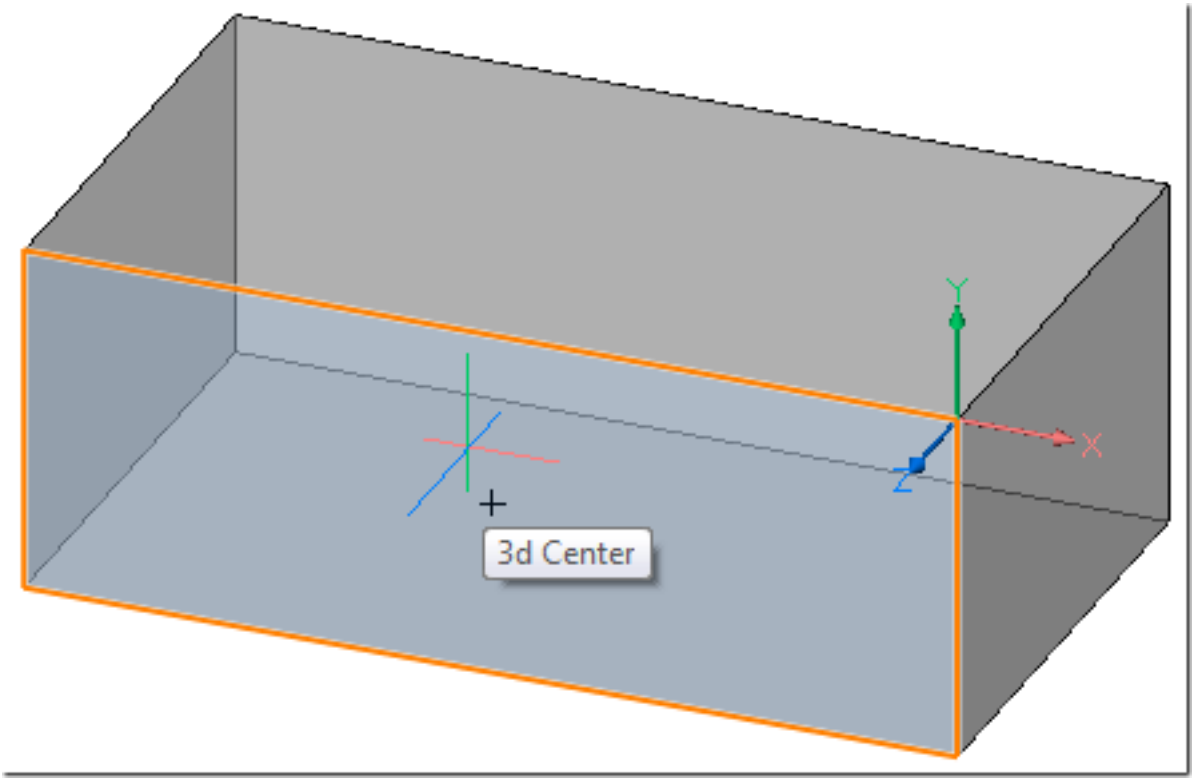
- When used at the command prompt:

: **ZCenter** Displays no prompts in the command bar

- When used as a transparent toggle during another command:

: **Line** Start of line: zce Snap to face center of:**(Pick a point)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/eg.png})

Figure 6.2669



Command Options

Option	Description
zcenter	This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').This can be expensive on complex curved faces.
Snap to face center of	Move cursor onto the face whose center you want to snap to. The program shows a **+** target glyph at the center point, and displays a tooltip reading "3d Center".

Related Commands

3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. -3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. OSnap (6.23.12) - sets entity snaps through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command line . Aperture (6.4.10) - sizes the aperture cursor for entity snaps. GCenter (6.15.4) - toggles Geometric Center (GCenter) entity snap; snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids. Zintersection (6.35.2) - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces. Zknot (6.35.3) - toggles the 3D knot entity snap. Snaps to a knot on a spline (6.27.65). Zmidpoint (6.35.4) - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge. Znearest (6.35.5) - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor. Znone (6.35.6) - disables all 3d snap modes. Zperpendicular (6.35.8) - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face. Zvertex (6.35.9) - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Figure 6.2670



6.35.2 ZIntersection

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the 3D intersection snap; snaps to the intersections of linear entities, edges, and polar or entity snap tracking lines with faces.

Accessing the Command

 command bar: **zintersection** transparent: **zint** toolbar: **3D Entity Snaps** | 

- When used at the command prompt:

: **ZIntersection** Displays no prompts in the command bar.

- When used as a transparent toggle during another command:

: **Line** Start of line: zint Snap to face intersection of: **(Pick a point)** Snapping to the intersection point of a line and the face of a 3D solid: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3dOsnapIntersect Snapping to the intersection point of a polar tracking line and the face of a 3D solid: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3dOsnapIntersectionTrackingLine.png})

Figure 6.2671

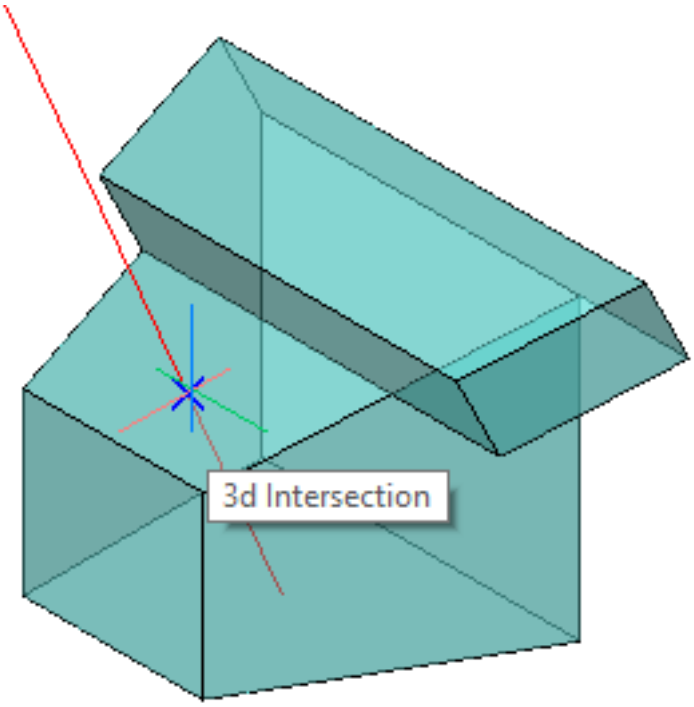
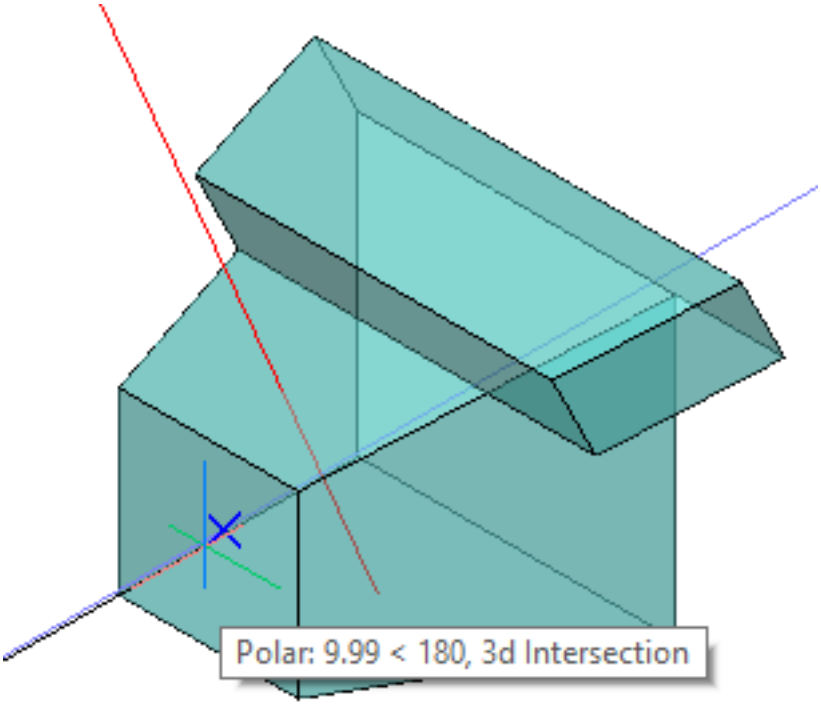


Figure 6.2672



Command Options

Option	Description
zint	This keyword can be entered transparently during commands, but must be entered without the apostrophe prefix (').
Snap to face intersection of	Move cursor near the elements whose intersection you want to snap to. The program shows a + target glyph at the center point, and displays a tooltip reading "3d Intersection".

Related Commands

3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. -3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. OSnap (6.23.12) - sets entity snaps through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command line . Aperture (6.4.10) - sizes the aperture cursor for entity snaps. Midpoint (6.21.16) - toggles Midpoint entity snap; snaps to the middle of lines, arcs, and other open entities, and edges of 3D solids. Zcenter (6.35.1) - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces. Zknot (6.35.3) - toggles the 3D knot entity snap. Snaps to a knot on a spline (6.27.65). Zmidpoint (6.35.4) - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge. Znearest (6.35.5) - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor. Znone (6.35.6) - disables all 3d snap modes. Zperpendicular (6.35.8) - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face. Zvertex (6.35.9) - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Figure 6.2673



6.35.3 ZKnot

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the 3D knot entity snap: snaps to knots onspline (6.27.65)s.

Accessing the Command

command bar: **zknot** transparent: **zkn** toolbar: **3D Entity Snaps** |

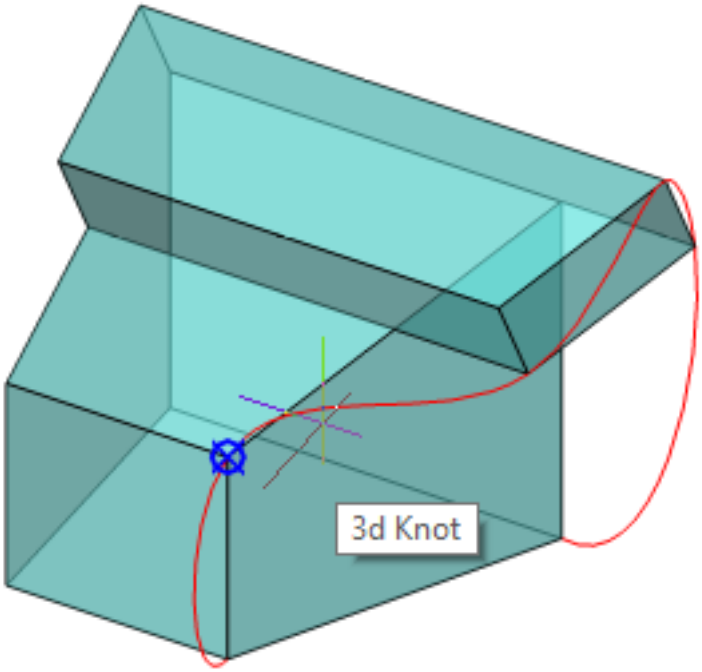
- When used at the command prompt:

: **ZKnot** Displays no prompts in the command bar.

- When used as a transparent toggle during another command:

: **Line** Start of line: zkno Snap to spline knot:**(Pick a point)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V

Figure 6.2674



Command Options

Option	Description
zknot	This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').
Snap to spline knot	Move cursor near the spline whose knot you want to snap to. The program shows a + target glyph at the center point, and displays a tooltip reading "3d Knot".

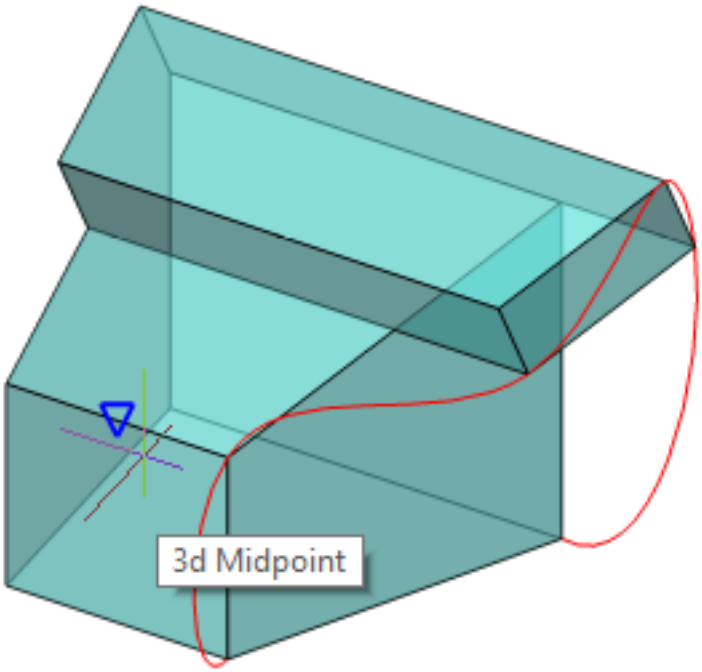
Related Commands

3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. -3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. OSnap (6.23.12) - sets entity snaps through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command line . Aperture (6.4.10) - sizes the aperture cursor for entity snaps. Zcenter (6.35.1) - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces. Zintersection (6.35.2) - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces. Zmidpoint (6.35.4) - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge. Znearest (6.35.5) - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor. Znone (6.35.6) - disables all 3d snap modes. Zperpendicular (6.35.8) - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face. Zvertex (6.35.9) - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Figure 6.2675



Figure 6.2676



6.35.4 ZMidpoint

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the 3D midpoint snap: snaps to the midpoint of a face edge. **Note:** You can also use the 2D MIDpoint entity snap to snap to face edges. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/

Accessing the Command

command bar: **zmidpoint** transparent: **zmi** toolbar: **3D Entity Snaps** |

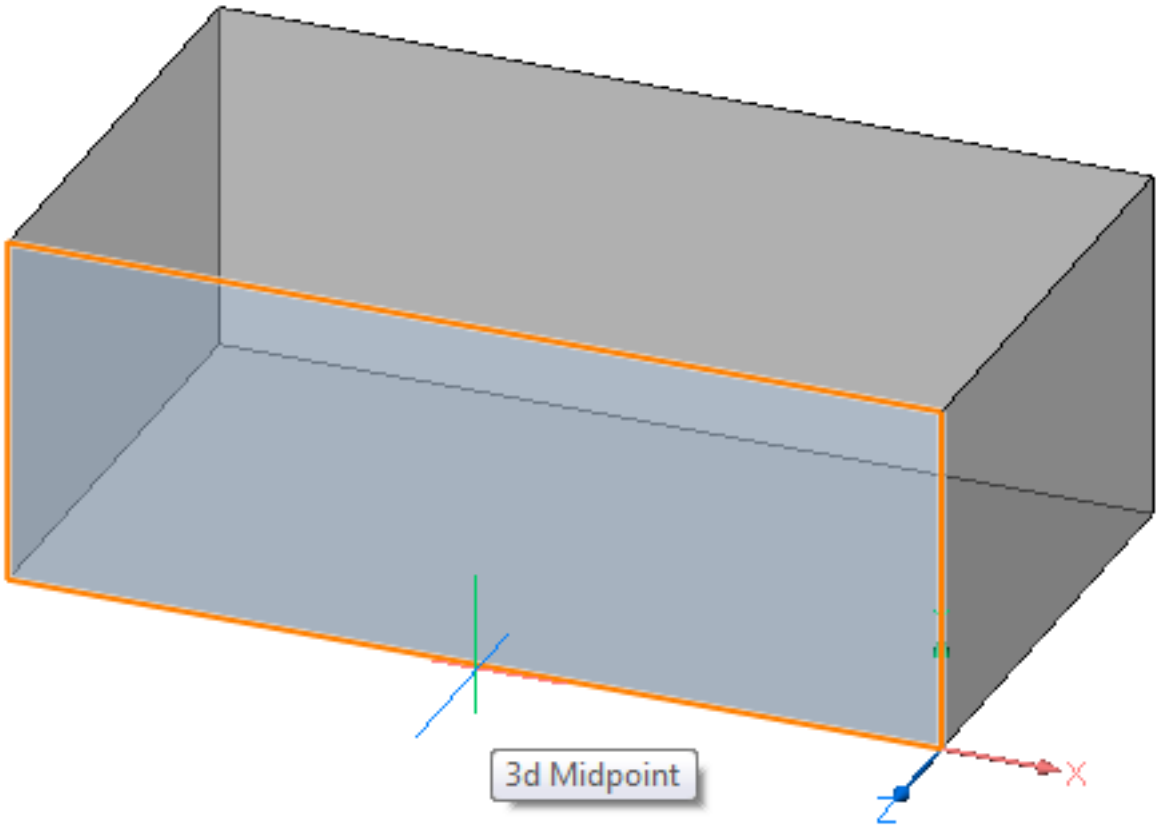
- When used at the command prompt:

: **ZMidpoint** Displays no prompts in the command bar.

- When used as a transparent toggle during another command:

: **Line** Start of line: zmi Snap to edge midpoint of:**(Pick a point)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/eg.png})

Figure 6.2677



Command Options

Option	Description
zmidpoint	This command can be entered transparently during commands, but must be entered without the apostrophe prefix ('). • When 3D snap ZMidpoint is off, the 2D snap MIDpoint snap still works on 3D entities • When 2D snapping is off, 3D snap for midpoints does not work on 2D entities
nap to edge midpoint of	Move cursor onto the edge whose midpoint you want to snap to. The program shows a **+** target glyph at the mid point, and displays a tooltip reading "3d Midpoint".

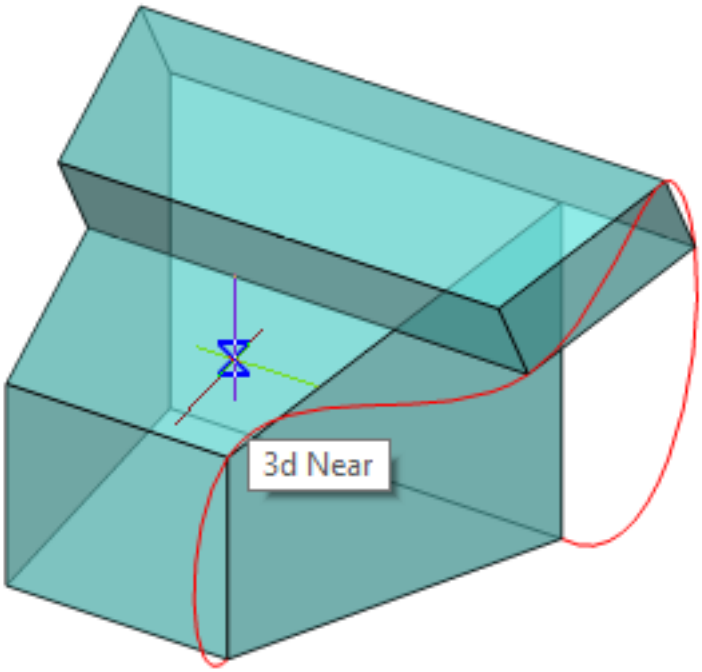
Related Commands

3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. -3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. OSnap (6.23.12) - sets entity snaps through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command line . Aperture (6.4.10) - sizes the aperture cursor for entity snaps. Midpoint (6.21.16) - toggles Midpoint entity snap; snaps to the middle of lines, arcs, and other open entities, and edges of 3D solids. Zcenter (6.35.1) - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces. Zintersection (6.35.2) - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces. Zknot (6.35.3) - toggles the 3D knot entity snap. Snaps to a knot on a spline (6.27.65). Znearest (6.35.5) - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor. Znone (6.35.6) - disables all 3d snap modes. Zperpendicular (6.35.8) - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face. Zvertex (6.35.9) - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Figure 6.2678



Figure 6.2679



6.35.5 ZNearest

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the 3D nearest entity snap, which snaps to a point on the face of a 3D entity that is nearest to the cursor. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3dOsnap

Accessing the Command

 command bar: **znearest** transparent: **zne** toolbar: **3D Entity Snaps** | 

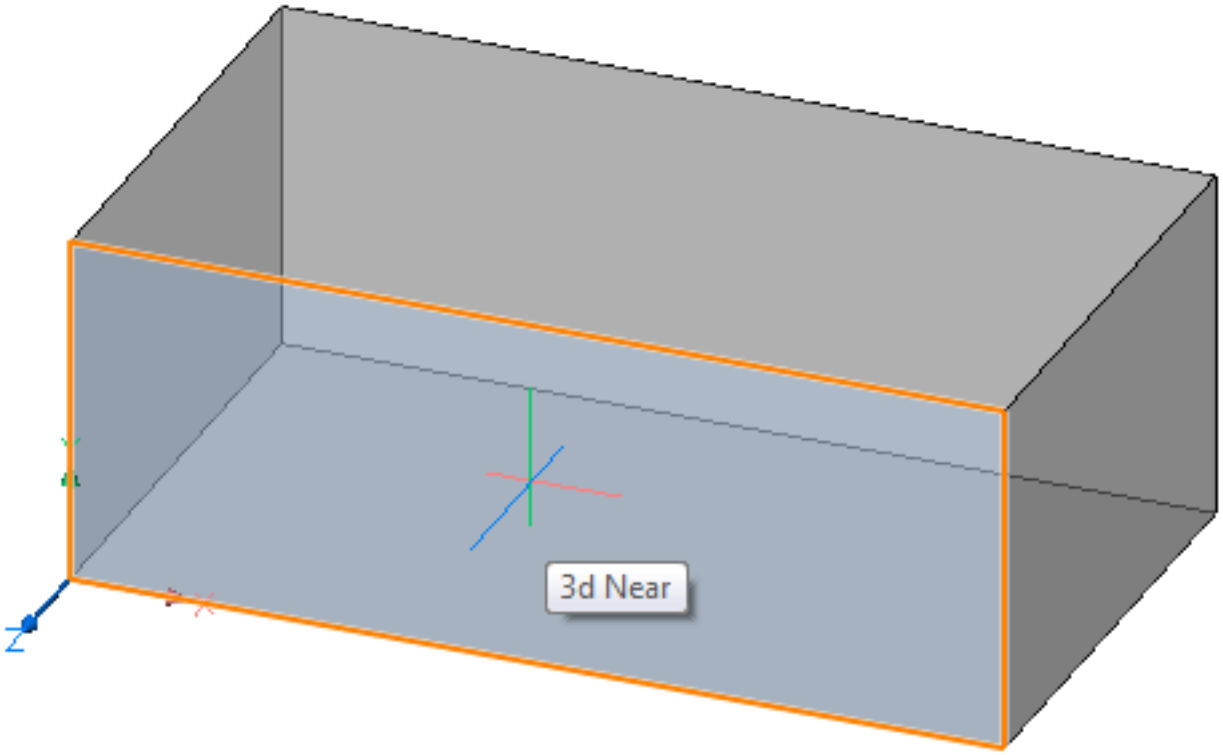
- When used at the command prompt:

: **ZNearest** Displays no prompts in the command bar.

- When used as a transparent toggle during another command:

: **Line** Start of line: zne Snap to face nearest of:**(Pick a point)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS eg.png})

Figure 6.2680



Command Options

Option	Description
znearest	This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').
Snap to face nearest of	Move cursor onto the nearest face to which you want to snap. The program displays a tooltip reading "3d Near".

Related Commands

3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. -3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. OSnap (6.23.12) - sets entity snaps through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command line . Aperture (6.4.10) - sizes the aperture cursor for entity snaps. Nearest (6.22.2) - toggles Nearest entity snap mode; snaps to the nearest geometry on entities. Zcenter (6.35.1) - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces. Zintersection (6.35.2) - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces. Zknot (6.35.3) - toggles the 3D knot entity snap. Snaps to a knot on a spline (6.27.65). Zmidpoint (6.35.4) - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge. Znone (6.35.6) - disables all 3d snap modes. Zperpendicular (6.35.8) - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face. Zvertex (6.35.9) - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Figure 6.2681



6.35.6 ZNone

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Disables all 3D snap modes.

Accessing the Command

command bar: **znone** transparent: **znon** toolbar: **3D Entity Snaps** | : **ZNone** Displays no prompts in the command bar.

Command Options

Option	Description
znone	This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').

Related Commands

3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. -3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. OSnap (6.23.12) - sets entity snaps through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command line . Aperture (6.4.10) - sizes the aperture cursor for entity snaps. Zcenter (6.35.1) - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces. Zintersection (6.35.2) - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces. Zknot (6.35.3) - toggles the 3D knot entity snap. Snaps to a knot on a spline (6.27.65). Zmidpoint (6.35.4) - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge. Znearest (6.35.5) - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor. Zperpendicular (6.35.8) - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face. Zvertex (6.35.9) - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Figure 6.2682



6.35.7 Zoom

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-all.png}) Changes the size of the drawing larger and smaller visually within the current viewport. **Note:**To zoom in and out at any time, roll the mouse's wheel back and forth; the zoom is centered at the cursor location, which is handy for zooming into a specific entity.

Accessing the Command



command bar: **zoom** transparent: **'zoom** alias: **z** menu bar: **View | Zoom** ribbon:**View | Zoom | Zoom** toolbar: **Zoom |** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/ART/0105ZoomButtons.png} : **Zoom Prompts you in the command bar:** Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/Scale (nx/nxp)/Window/OBject] <Scale (nX/nXP)>: **(Enter an option, or pick two points for a windowed zoom, or enter a zoom factor)**

Figure 6.2683



Command Options

Option	Description
Scale	Specifies the absolute zoom scale factor: • Less than 1 - makes the drawing appear smaller. • **1** - keeps the drawing the same size. • More than 1 - makes the drawing appear larger.
nX	Specifies the zoom factor relative to the current zoom- For example, 2x makes the current view twice as large.
nXP	Specifies the zoom factor in paper space relative to model space.For example, 2xp makes the model space view twice as large as the surrounding paper space.
In	Zooms in by 50%, equivalent to Zoom 0.5
Out	Zooms out by 50%, equivalent to Zoom 2
All	Displays the entire drawing; zooms to the larger of: • Drawing limits - see the Limits (6.20.30) command • Extents of drawing - excludes entities on frozen layers
Center	Zooms in or out about a specified center point; prompts you: 1. **Center of view** - **(pick a point or enter x,y coordinates.)** 2. **Enter zoom factor (nX/nXP), or view height** - **(enter a zoom factor or view height.)** The view height defines the zoom; it is the height of the current viewport: (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Dynamic	Pans and zooms using a rectangular box which represents the viewport; this command is largely replaced by real-time zooming. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Commands/Z/zo dynamioc-eg.png}) This option first executes a zoom extents or zoom limits to show the original size of the viewport as a dashed rectangle. Adjust the size of the view box dynamically, then pan it to the part of the drawing you want to view.Click to switch between the two modes. • **X** - zoom mode • **->** (arrow) - pan mode
Extents	Displays the drawing to the extents of entities; limits and frozen entities are ignored. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i

Left	Zooms in or out with the lower left corner specified by a point; prompts you: 1. **Lower left corner** - **(pick a point or enter x,y coordinates.)** 2. **Enter zoom factor (nX/nXP), or view height** - **(enter a zoom factor or view height.)** The view height defines the zoom; it is the height of the current viewport. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Previous	Displays the previous view, whether set by the Zoom, Pan, or View commands.
Right	Zooms in or out with the upper right corner specified by a point; prompts you: 1. **Upper right corner** - **(pick a point or enter x,y coordinates.)** 2. **Enter zoom factor (nX/nXP), or view height** - **(enter a zoom factor or view height.)** The view height defines the zoom; it is the height of the current viewport. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
Window	Zooms in to a rectangular area; prompts you: 1. **First corner** - **(pick a point or enter x,y coordinates.)** 2. **Opposite corner** - **(pick another point or enter x,y coordinates.)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/i
OBject	Zooms in on a single entity or a selection set; prompts you: 1. **Select an entity** - **(select one or more entities.) Entities in set: 1** 2. **Select an entity** - **(select more entities or eles press Enter to zoom in on the selection set.)**

Related Commands

RtZoom (6.26.44) - performs real-time zooming of the drawing. Pan (6.24.2) and -Pan (6.24.3) - move the drawing within the current viewport. RtPan (6.26.39) - performs real-time panning of the drawing. View (6.31.1) and -View (6.31.2) - set up named views of drawings. DdVPoint (6.10.22) - changes the 3D viewpoint of the drawing through a dialog box. VPoint (6.31.19) - changes the 3D viewpoint of the drawing through the command bar. RtRot (6.26.38) - rotates the 3D viewpoint in realtime. RtRotX (6.26.49) - rotates the 3D viewpoint about the x axis in realtime. RtRotY (6.26.50) - rotates the 3D viewpoint about the y axis in realtime. RtRotZ (6.26.41) - rotates the 3D viewpoint about the z axis in realtime.

Figure 6.2684

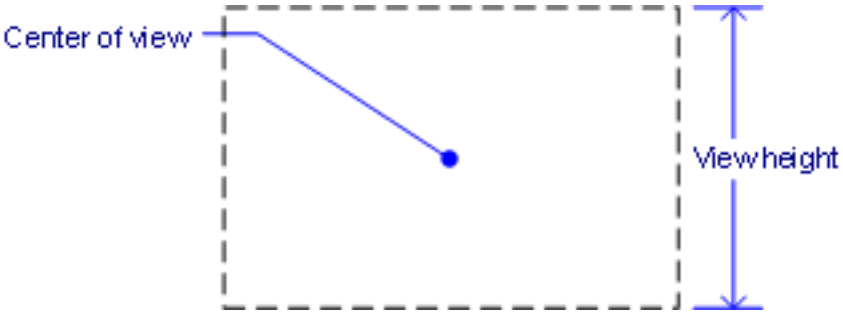


Figure 6.2685

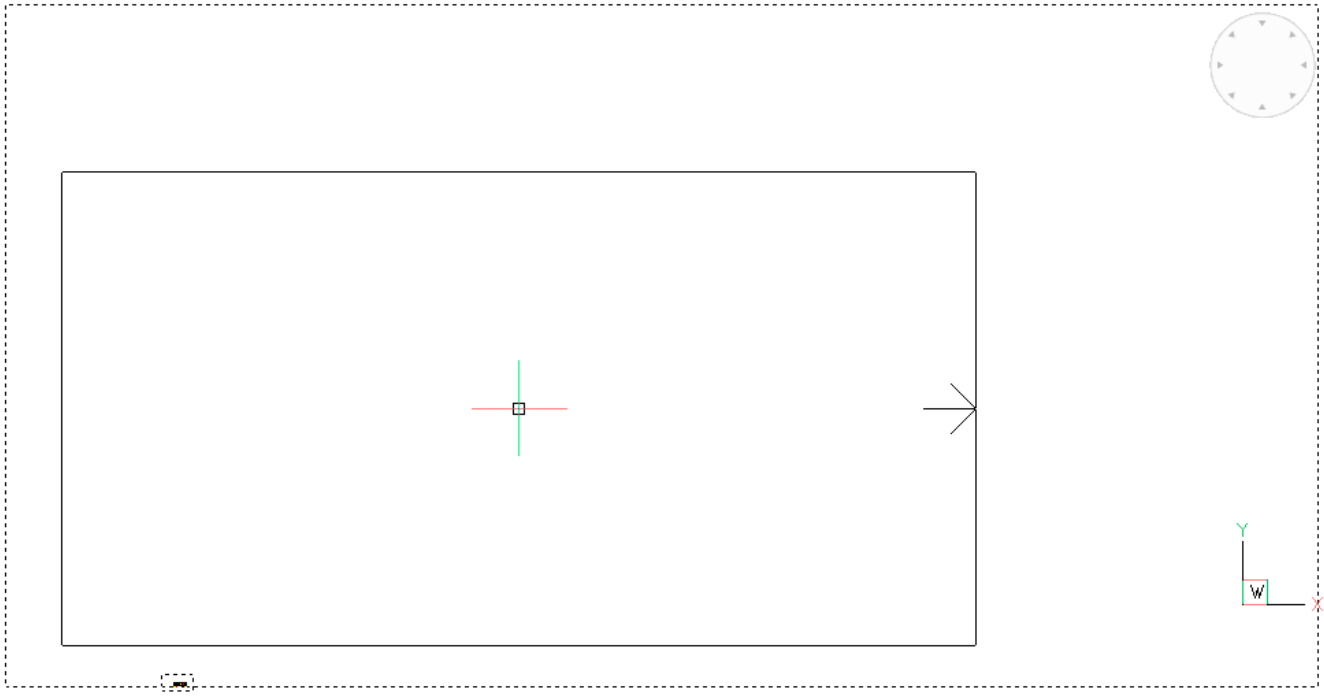


Figure 6.2686

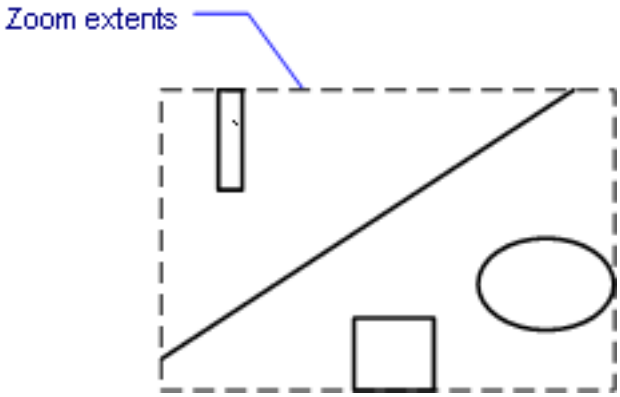


Figure 6.2687

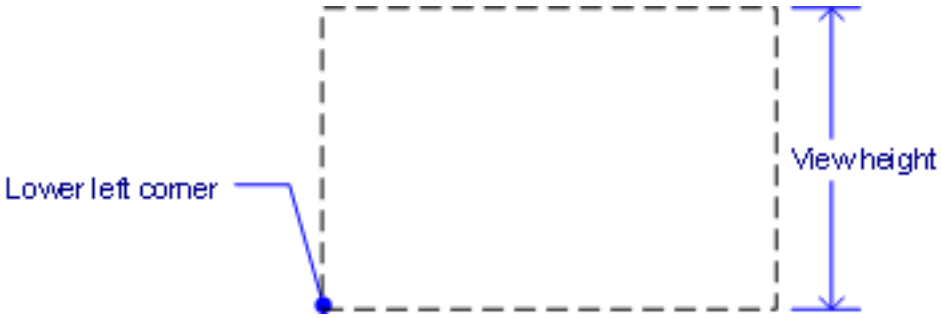


Figure 6.2688

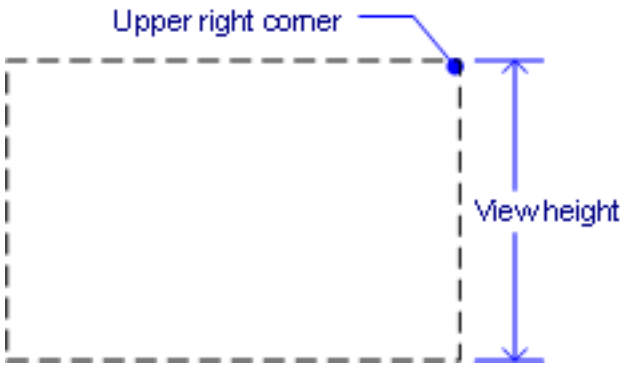


Figure 6.2689

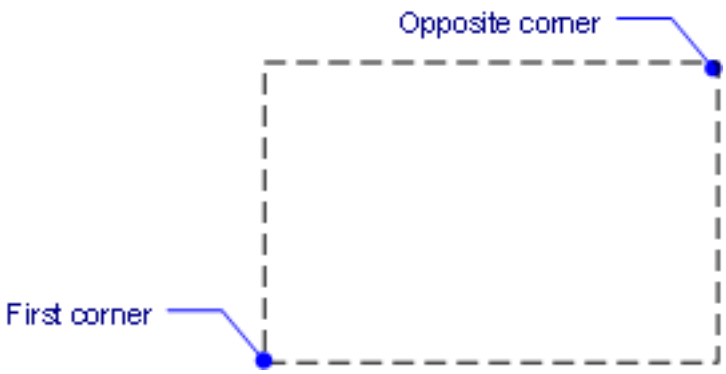
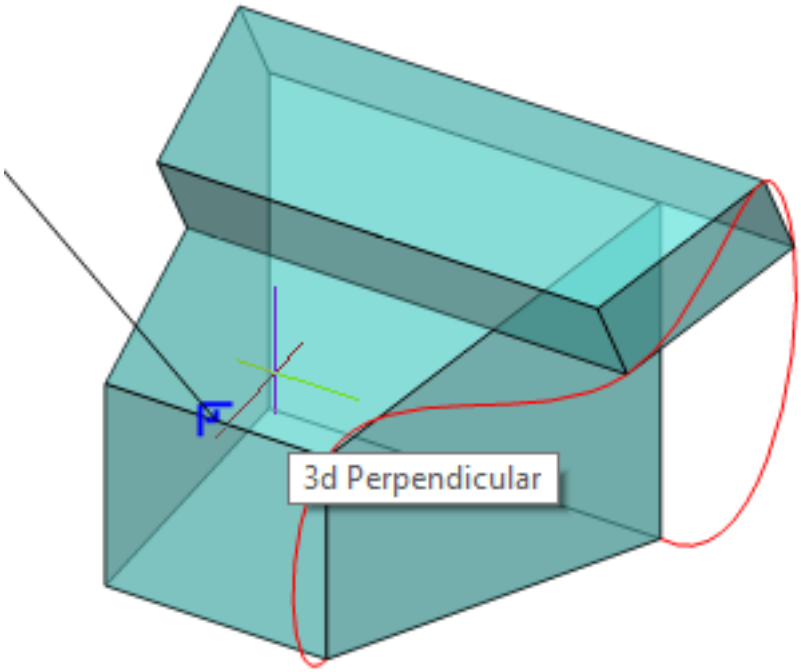


Figure 6.2690



Figure 6.2691



6.35.8 ZPerpendicular

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the 3D perpendicular entity snap, which snaps to a point perpendicular to a face on a 3D entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3dOsnapPerpend

Accessing the Command

command bar: **zperpendicular** transparent: **zpe** toolbar: **3D Entity Snaps** |

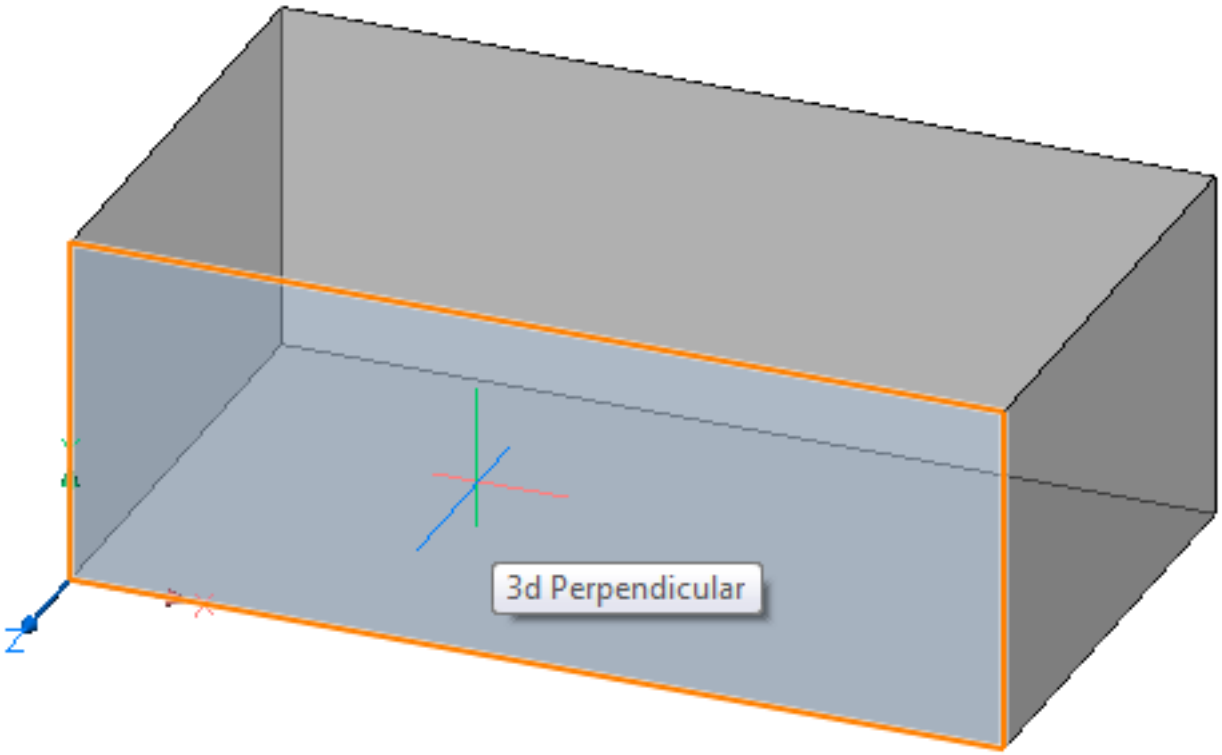
- When used at the command prompt:

: **ZPerpendicular** Displays no prompts in the command bar

- When used as a transparent toggle during another command:

: **Line** Start of line: zpe Snap to face perpendicular of:**(Pick a point)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help eg.png})

Figure 6.2692



Command Options

Option	Description
zperpendicular	This command can be entered transparently during commands, but must be entered without the apostrophe prefix (').
Snap to face perpendicular of	Move cursor onto the body whose face you want to snap to perpendicularly. The program displays a tooltip reading "3d Perpendicular".

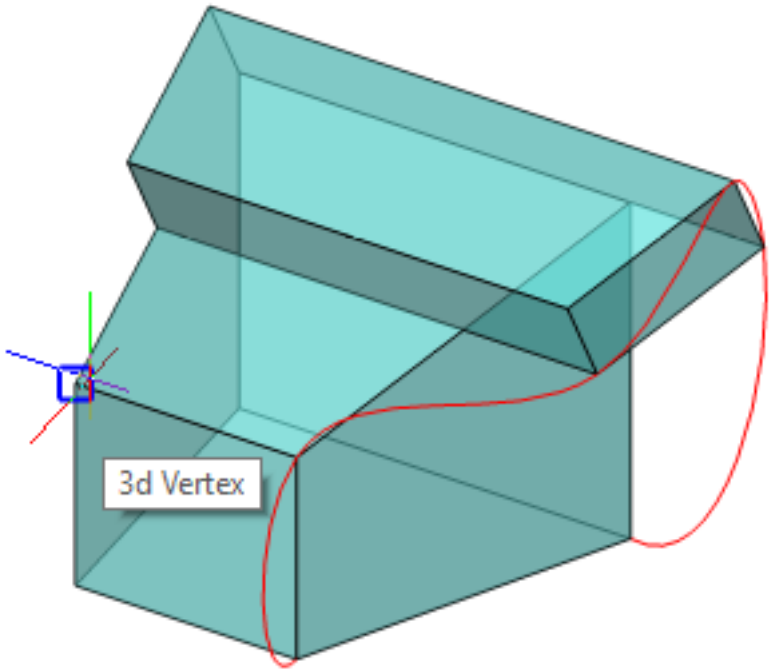
Related Commands

3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. -3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. OSnap (6.23.12) - sets entity snaps through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command line . Aperture (6.4.10) - sizes the aperture cursor for entity snaps. Perpendicular (6.24.22) - toggles perpendicular entity snap mode. Zcenter (6.35.1) - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces. Zintersection (6.35.2) - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces. Zknot (6.35.3) - toggles the 3D knot entity snap. Snaps to a knot on a **spline**. Zmidpoint (6.35.4) - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge. Znearest (6.35.5) - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor. Znone (6.35.6) - disables all 3d snap modes. Zvertex (6.35.9) - toggles the 3D vertex entity snap. Snaps to the closest vertex of a 3d entity.

Figure 6.2693



Figure 6.2694



6.35.9 ZVertex

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Toggles the 3D vertex entity snap, so that the cursor snaps to the closest vertex of a 3D entity. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V18/CmdRef/source/3dOsnapVertex.png})

Accessing the Command

 command bar: **zvertex** transparent: **zve** toolbar: **3D Entity Snaps** | 

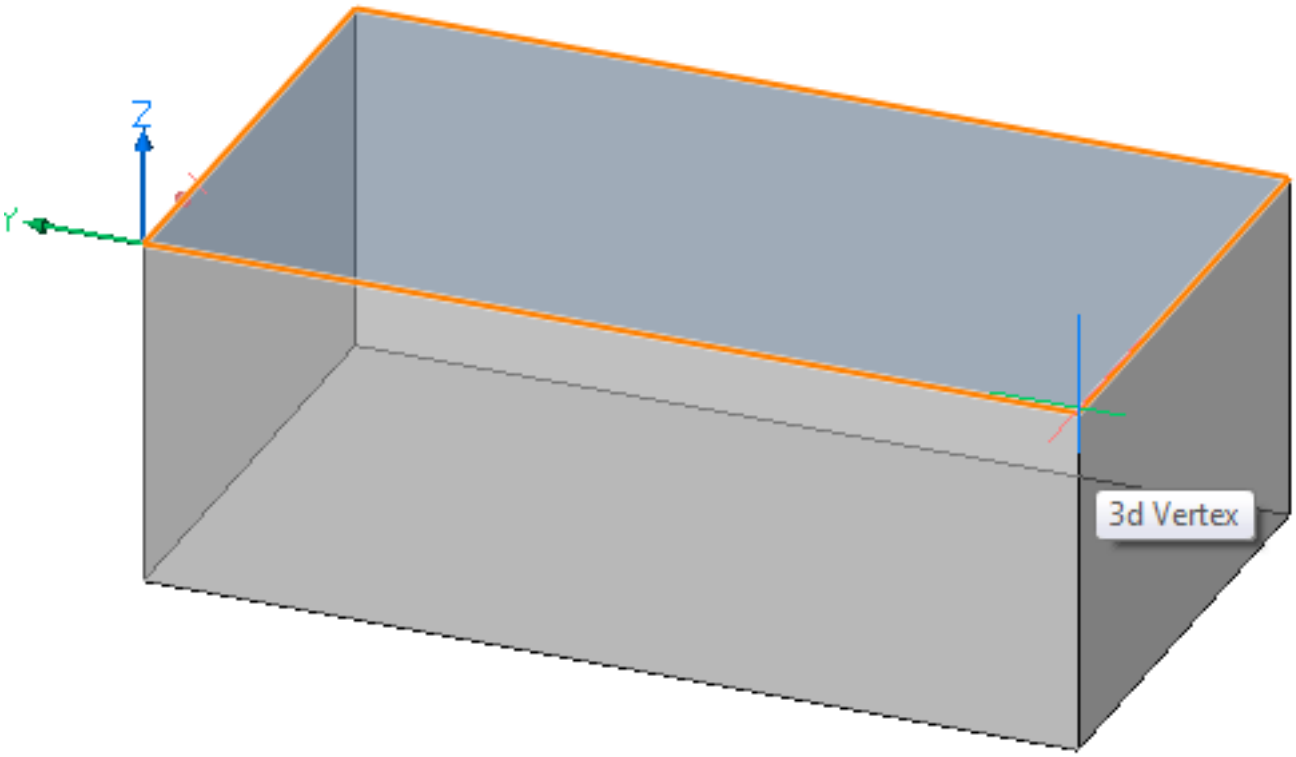
- When used at the command prompt:

: **ZVertex** Displays no prompts in the command bar

- When used as a transparent toggle during another command:

: **Line** Start of line: zve Snap to vertex of:**(Pick a point)** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/eg.png})

Figure 6.2695



Command Options

Option	Description
zvertex	This command can be entered transparently during commands, but must be entered without the apostrophe prefix ('). • When 3D snapping (Zvertex) is off, the 2D snap ENDpoint still works on 3D entities • When 2D snapping is off, 3D snap does not work on 2D objects
Snap to vertex of	Move cursor near the vertex of the 3D solid which you want to snap. The program and displays a tooltip reading "3d Vertex". A vertex is where two or more edges come together.

Related Commands

3dOsnap (6.3.8) - sets the entity snap modes for 3D entities through the Settings dialog box. -3dOsnap (6.3.9) - toggles the entity snap modes for 3D entities at the command line. OSnap (6.23.12) - sets entity snaps through the Settings dialog box. -OSnap (6.23.13) - toggles entity snap modes at the command line . Aperture (6.4.10) - sizes the aperture cursor for entity snaps. Endpoint (6.13.7) - toggles endpoint entity snap; snaps to the ends of open entities, such as line, arcs, and open polylines and vertices of 3D solids. Zcenter (6.35.1) - toggles the 3D center entity snap. Snaps to the center of planar or curved 3d faces. Zintersection (6.35.2) - Toggles the 3D intersectionpoint snap. Snaps to the intersection of linear entities, edges and polar or entity snap tracking lines with faces. Zknot (6.35.3) - toggles the 3D knot entity snap. Snaps to a knot on a **spline**. Zmidpoint (6.35.4) - toggles the 3D midpoint snap. Snaps to the midpoint of a face edge. Znearest (6.35.5) - toggles the 3D nearest entity snap. Snaps to a point on the face of a 3D entity that is nearest to the cursor. Znone (6.35.6) - disables all 3d snap modes. Zperpendicular (6.35.8) - toggles the 3D perpendicular entity snap. Snaps to a point perpendicular to a face.

6.36 ?

6.36.1 ?

(\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Common/command-reference-classic-pro-platinum-mechanical-bim.png}) Opens the BricsCAD Online Help on the Bricsys website. 

Figure 6.2696



Accessing the Command

command bar: **?** transparent: **'?** shortcut: **F1** alias: help (6.16.8)menu bar: **Help | Help** toolbar: **Standard |**  ribbon: **Home | Help | Help** : **?** Opens the BricsCAD online help on the Bricsys website; see the Help (6.16.8) command. When used transparently during a command, '? displays help about the command, such as during the Line command: : **Line** Start of line or [Follow] <Last point>:'?

Command Options

See the Help (6.16.8) command.

Related Commands

Help (6.16.8) - also opens the BricsCAD online help on the Bricsys website. HelpSearch (6.16.9) - searches BricsCAD Help and then displays a list of matching topics in a dialog box.

Chapter 7

Settings

Chapter 8

Guided Lessons

8.1 2D Drafting in BricsCAD

8.1.1 Lesson 1: Getting Started with BricsCAD

Welcome to Learning BricsCAD. A fundamentals level course for the practical use and application of BricsCAD in a design environment. This course is intended for the learner who is new to computer-aided design (CAD). No experience with BricsCAD or any other CAD product is necessary.

8.1.2 Course chapters

After completing this course you will be able to:

1/ Identify key elements of the User Interface .

2/ Work with drawing files .

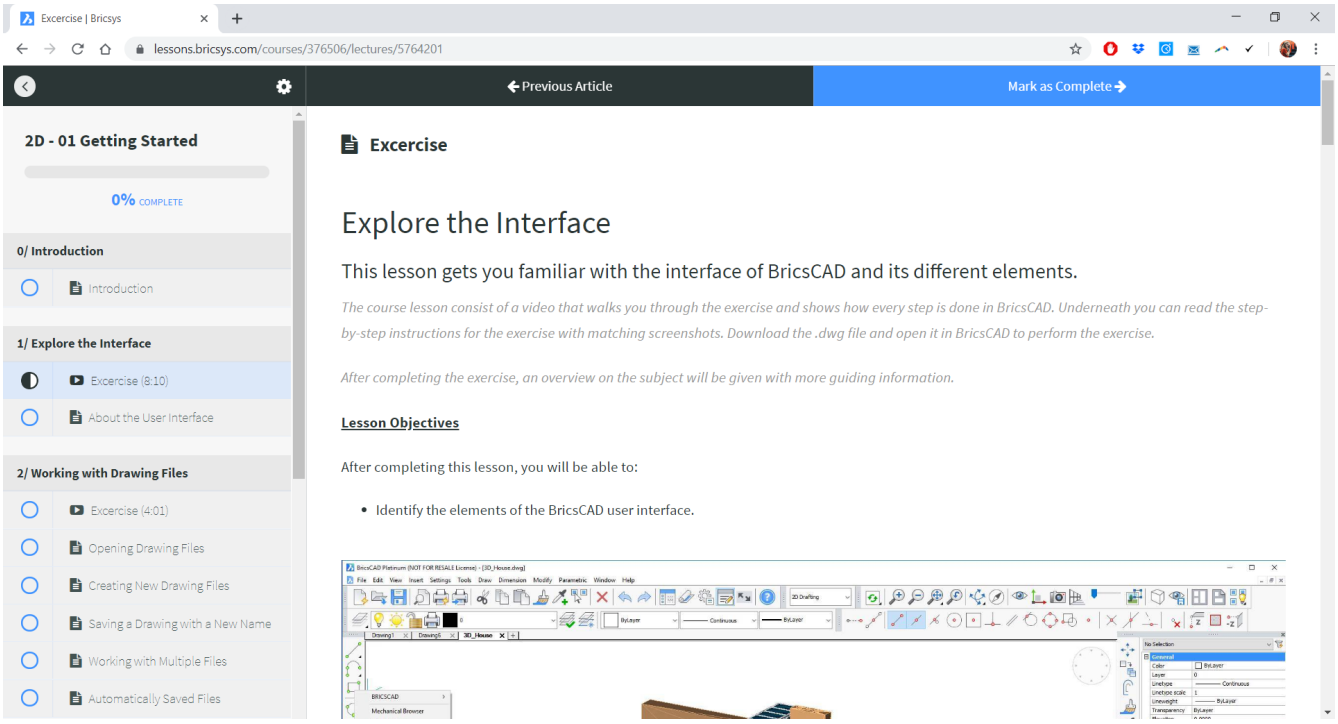
- Describe the methods available to open drawing files in BricsCAD.
- Explain how to create a new drawing file using a template file.
- Save a drawing file with a new name.
- Add shortcuts to the Places list.
- Explain how to use tabs to work with multiple files.
- Describe the importance of and how to set up autosave and backup settings.

3/ Use different tools to view your drawings .

- Identify what zooming tools are used for.
- Illustrate the various ways for accessing the Zoom tools including the Navigation bar, the right-click menu, and the command line.
- Describe how to use Real-Time Zoom and other Zoom options such as Window, Extents, and Previous.
- Change the view without changing the magnification of the view using the Pan command.
- View the contents of the drawing window from any direction using the LookFrom control.

This is only a preview Get access to the full course materials and step by step exercises at lessons.bricsys.com
The course lesson consists of a videothat walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screen-shots. Download the .dwg file and open it in BricsCAD to perform the exercise. After completing the exercise, an overview on the subject will be given with more guiding information. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h

Figure 8.1


Excercise | Bricsys
lessons.bricsys.com/courses/376506/lectures/5764201
Previous Article
Mark as Complete
2D - 01 Getting Started
0% COMPLETE
0/ Introduction
Introduction
1/ Explore the Interface
Excercise (8:10)
About the User Interface
2/ Working with Drawing Files
Excercise (4:01)
Opening Drawing Files
Creating New Drawing Files
Saving a Drawing with a New Name
Working with Multiple Files
Automatically Saved Files
Excercise
Explore the Interface
This lesson gets you familiar with the interface of BricsCAD and its different elements.
The course lesson consist of a video that walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screenshots. Download the .dwg file and open it in BricsCAD to perform the exercise.
After completing the exercise, an overview on the subject will be given with more guiding information.
Lesson Objectives
After completing this lesson, you will be able to:
Identify the elements of the BricsCAD user interface.

8.1.3 Lesson 2: Primary Drawing Techniques

Welcome to Learning BricsCAD. A fundamentals level course for the practical use and application of BricsCAD in a design environment. This course is intended for the learner who is new to computer-aided design (CAD). No experience with BricsCAD or any other CAD product is necessary.

8.1.4 Course chapters

After completing this course you will be able to:

1/ Drawbasic objects .

- Draw a line by defining the start and endpoint.
- Add a rectangle.
- Create a circle.
- Draw an arc by selecting three consecutive points.
- Create a polygon.

2/ Work withEntity Snaps and Tracking .

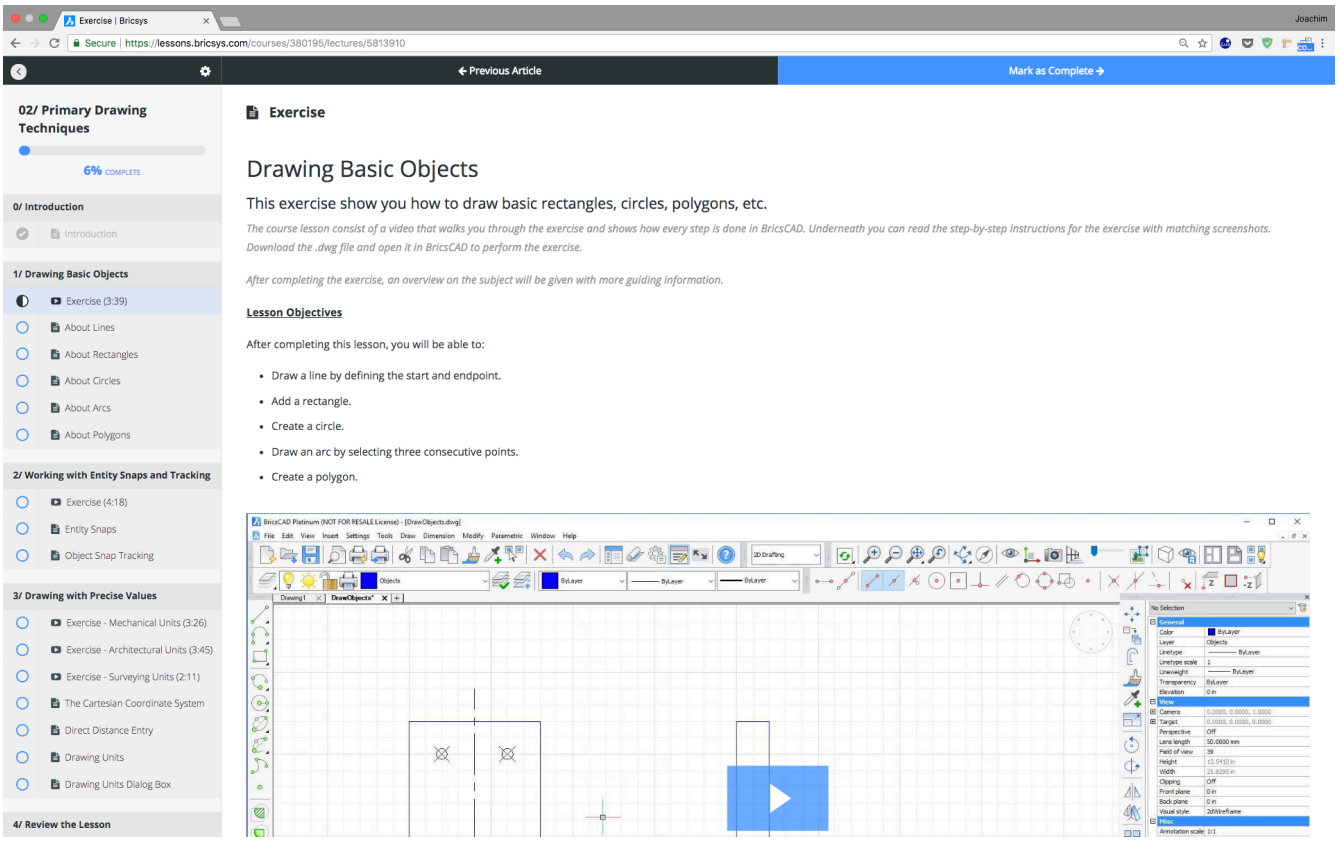
- Improve the accuracy of your drawings by using entity snaps to create and edit geometry.
- Incorporate theoretical construction geometry when you create and edit geometry.

3/ Draw usingprecise values.

- Understand the Cartesian Coordinate System upon which the BricsCAD drawing window is based.
- Recognize the advantage of using Direct Distance Entry while creating and modifying objects.
- Understand how drawing units determine the way measurement values are returned.
- Assign specific drawing unit values to a drawing.

This is only a preview Get access to the full course materials and step by step exercises at lessons.bricsys.com **The course lesson consists of a videothat walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screen-shots. Download the .dwg file and open it in BricsCAD to perform the exercise. After completing the exercise, an overview on the subject will be given with more guiding information.** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/h

Figure 8.2


Exercise | Bricsys
Joachim
Secure | https://lessons.bricsys.com/courses/380195/lectures/5813910
← Previous Article
Mark as Complete →
02/ Primary Drawing Techniques
6% COMPLETE
0/ Introduction
Introduction
1/ Drawing Basic Objects
Exercise (3:39)
About Lines
About Rectangles
About Circles
About Arcs
About Polygons
2/ Working with Entity Snaps and Tracking
Exercise (4:18)
Entity Snaps
Object Snap Tracking
3/ Drawing with Precise Values
Exercise - Mechanical Units (3:26)
Exercise - Architectural Units (3:45)
Exercise - Surveying Units (2:11)
The Cartesian Coordinate System
Direct Distance Entry
Drawing Units
Drawing Units Dialog Box
4/ Review the Lesson
Exercise
Drawing Basic Objects
This exercise show you how to draw basic rectangles, circles, polygons, etc.
The course lesson consist of a video that walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screenshots. Download the .dwg file and open it in BricsCAD to perform the exercise.
After completing the exercise, an overview on the subject will be given with more guiding information.
Lesson Objectives
After completing this lesson, you will be able to:
Draw a line by defining the start and endpoint.
Add a rectangle.
Create a circle.
Draw an arc by selecting three consecutive points.
Create a polygon.

8.1.5 Lesson 3: Essential Drawing Techniques

Welcome to Learning BricsCAD. A fundamentals level course for the practical use and application of BricsCAD in a design environment. This course is intended for the learner who is new to computer-aided design (CAD). No experience with BricsCAD or any other CAD product is necessary.

8.1.6 Course chapters

After completing this course you will be able to:

1/ Use differentselection modes and erase objects.

- Manually select objects in a drawing.
- Select objects using an Implied Window or an Implied Crossing.
- Use the Polygon selection mode to select objects.
- Use the Circle selection mode to select objects.
- Use the Point selection mode to select objects.
- Use the Fence selection mode to select objects.
- Use other selection modes to select objects.
- Use the Erase command to remove objects from a drawing.

2/Move and Copy objects.

- Use the Move command to reposition objects in your drawings.
- Duplicate objects in your drawings using the Copy command.

3/Offset objects.

- Create geometry at a specific distance from the selected geometry using the Offset command.

4/Trim and Extend geometry.

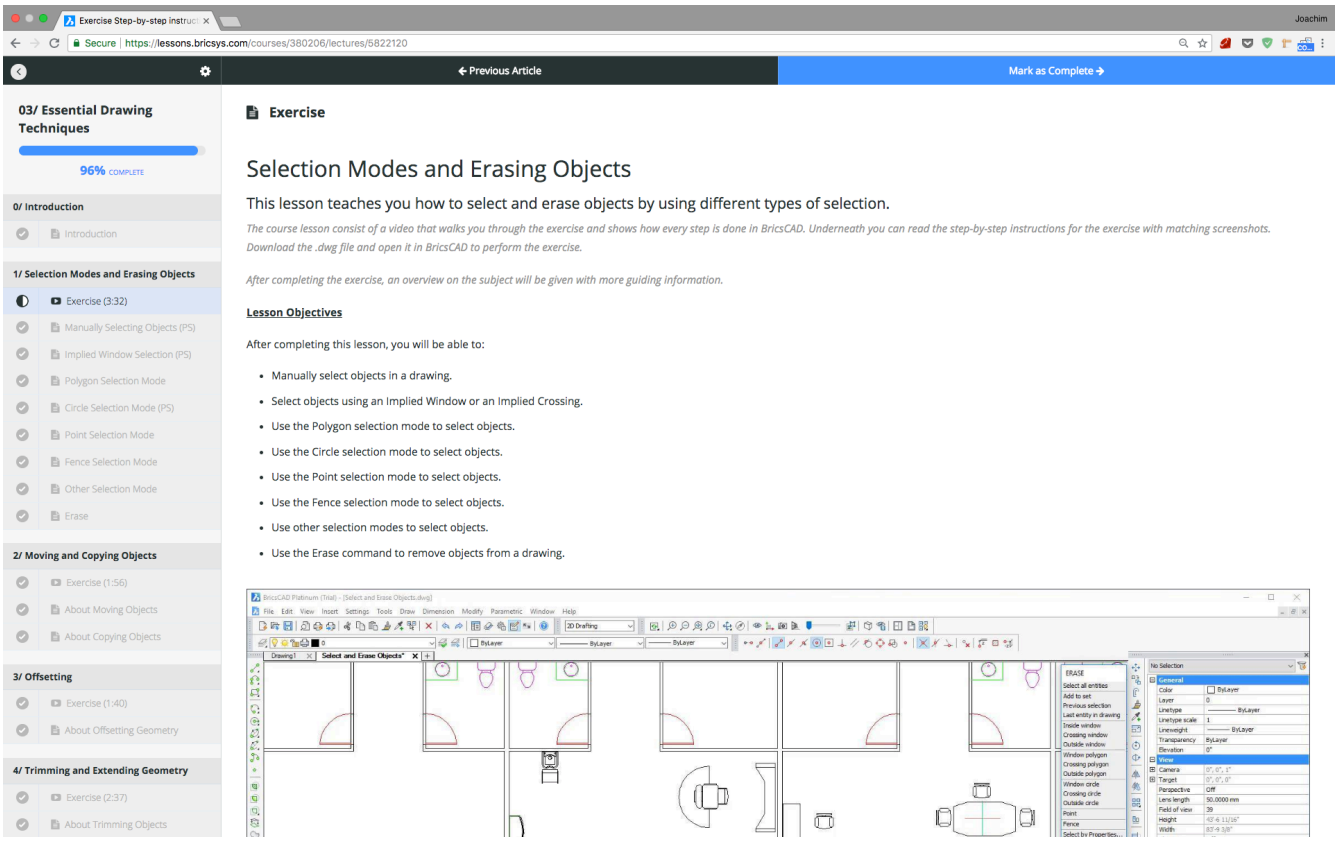
- Understand how to change the length of geometry based on other geometry in the drawing with the Trim command.
- Utilize the Extend command to change the length of geometry in relation to other geometry.

5/ Applyfillets and chamfers to geometry.

- Use the Fillet command to create round and sharp corners where geometry intersects.
- Create angled corners with the Chamfer command.

This is only a preview Get access to the full course materials and step by step exercises at lessons.bricsys.com **The course lesson consists of a videothat walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screen-shots. Download the .dwg file and open it in BricsCAD to perform the exercise. After completing the exercise, an overview on the subject will be given with more guiding information.** (\vref{fig:C:/Dev/Task-2448/images/staticbricsyscom/h

Figure 8.3


Exercise Step-by-step instruc
Joachim
Secure | https://lessons.bricsys.com/courses/380206/lectures/5822120
← Previous Article
Mark as Complete →
03/ Essential Drawing Techniques
96% COMPLETE
0/ Introduction
Introduction
1/ Selection Modes and Erasing Objects
Exercise (3:32)
Manually Selecting Objects (PS)
Implied Window Selection (PS)
Polygon Selection Mode
Circle Selection Mode (PS)
Point Selection Mode
Fence Selection Mode
Other Selection Mode
Erase
2/ Moving and Copying Objects
Exercise (1:56)
About Moving Objects
About Copying Objects
3/ Offsetting
Exercise (1:40)
About Offsetting Geometry
4/ Trimming and Extending Geometry
Exercise (2:37)
About Trimming Objects
Exercise
Selection Modes and Erasing Objects
This lesson teaches you how to select and erase objects by using different types of selection.
The course lesson consist of a video that walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screenshots. Download the .dwg file and open it in BricsCAD to perform the exercise.
After completing the exercise, an overview on the subject will be given with more guiding information.
Lesson Objectives
After completing this lesson, you will be able to:
Manually select objects in a drawing.
Select objects using an Implied Window or an Implied Crossing.
Use the Polygon selection mode to select objects.
Use the Circle selection mode to select objects.
Use the Point selection mode to select objects.
Use the Fence selection mode to select objects.
Use other selection modes to select objects.
Use the Erase command to remove objects from a drawing.

8.1.7 Lesson 4: Additional Drawing Techniques

Welcome to Learning BricsCAD. A fundamentals level course for the practical use and application of BricsCAD in a design environment. This course is intended for the learner who is new to computer-aided design (CAD). No experience with BricsCAD or any other CAD product is necessary.

8.1.8 Course chapters

After completing this course you will be able to:

1/ UseGrips

- Utilize Grips on geometry to initiate editing commands.
- Change the behavior and appearance of grips using the Options dialogue box.

2/Stretch Objects

- Use the Stretch command to change the shape of entities in the drawing.

3/Break andJoinObjects

- Use the Break command to separate one piece of geometry into two.
- Create a gapless break in geometry using the Same as First Point option.
- Take two pieces of geometry and combine them into one.

4/Mirror Objects

- Create duplicate geometry based on a defined centerline.

5/Rotate Objects

- Rotate geometry in relation to a specified base point.

6/Scale Objects

- Change the size of an object in the drawing.

7/Pattern Geometry

- Make copies of selected geometry in a specific pattern using one of three layouts.
- Create an array pattern based on rows, columns, and spacing.
- Use the Polar Array to arrange geometry in a circular pattern around a center point.
- Update the layout of a specified pattern using grips, the Array tab, and the Properties window.

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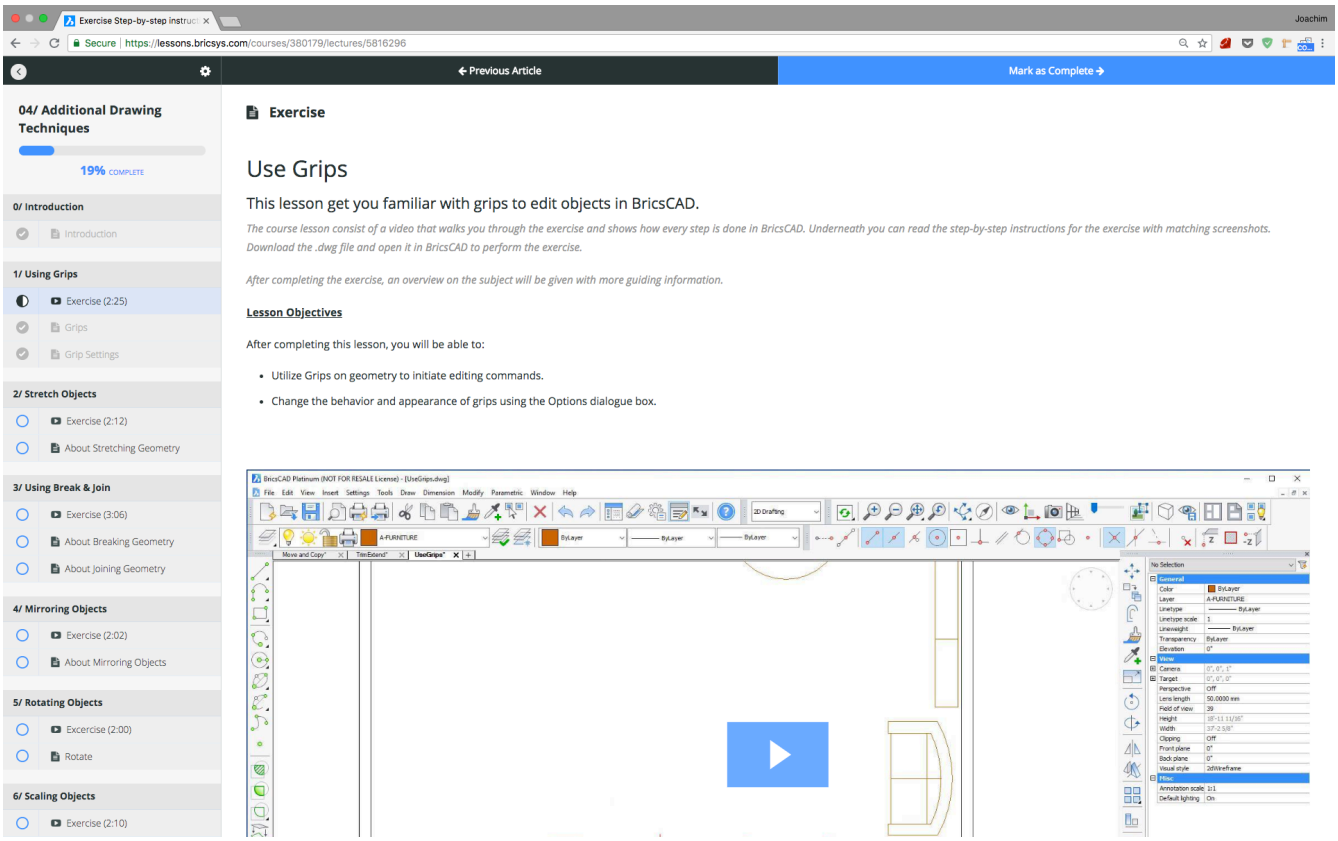
Figure 8.4


Exercise Step-by-step instruc
Joachim
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← Previous Article
Mark as Complete →
04/ Additional Drawing Techniques
19% COMPLETE
0/ Introduction
Introduction
1/ Using Grips
Exercise (2:25)
Grips
Grip Settings
2/ Stretch Objects
Exercise (2:12)
About Stretching Geometry
3/ Using Break & Join
Exercise (3:06)
About Breaking Geometry
About Joining Geometry
4/ Mirroring Objects
Exercise (2:02)
About Mirroring Objects
5/ Rotating Objects
Excercise (2:00)
Rotate
6/ Scaling Objects
Exercise (2:10)
Exercise
Use Grips
This lesson get you familiar with grips to edit objects in BricsCAD.
The course lesson consist of a video that walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screenshots. Download the .dwg file and open it in BricsCAD to perform the exercise.
After completing the exercise, an overview on the subject will be given with more guiding information.
Lesson Objectives
After completing this lesson, you will be able to:
Utilize Grips on geometry to initiate editing commands.
Change the behavior and appearance of grips using the Options dialogue box.

8.1.9 Lesson 5: Organizing Objects in a Drawing

Welcome to Learning BricsCAD. A fundamentals level course for the practical use and application of BricsCAD in a design environment. This course is intended for the learner who is new to computer-aided design (CAD). No experience with BricsCAD or any other CAD product is necessary.

8.1.10 Course chapters

After completing this course you will be able to:

1/ Create anduse Layers.

- Utilize separate virtual sheets to organize a drawing.
- Use the Drawing Explorer to create, delete, or change layers in a drawing.
- Change the appearance of objects in a drawing.
- Add layers into a drawing using the Drawing Explorer.
- Utilize the Layer List to change the current layer and alter certain layer properties.
- Understand how a layer can be changed.
- Delete a specific layer.

2/ Modifyobject properties.

- Understand how general properties affect an object.
- Make changes to an object using the Properties palette.
- Select an object and dynamically change it using Quad.
- Use a source object to apply congruent properties to another object.

3/ Obtaininformation about objects.

- Show specific information in a drawing using an Inquiry Command.
- Identify specifics about an objectusing a List command.
- Measure objects in a drawing for distance, angle, radius, and area.
- Use a Boundary Area to accurately measure an area of a curved or irregular object.
- Determine the location of an object on the coordinate plane.

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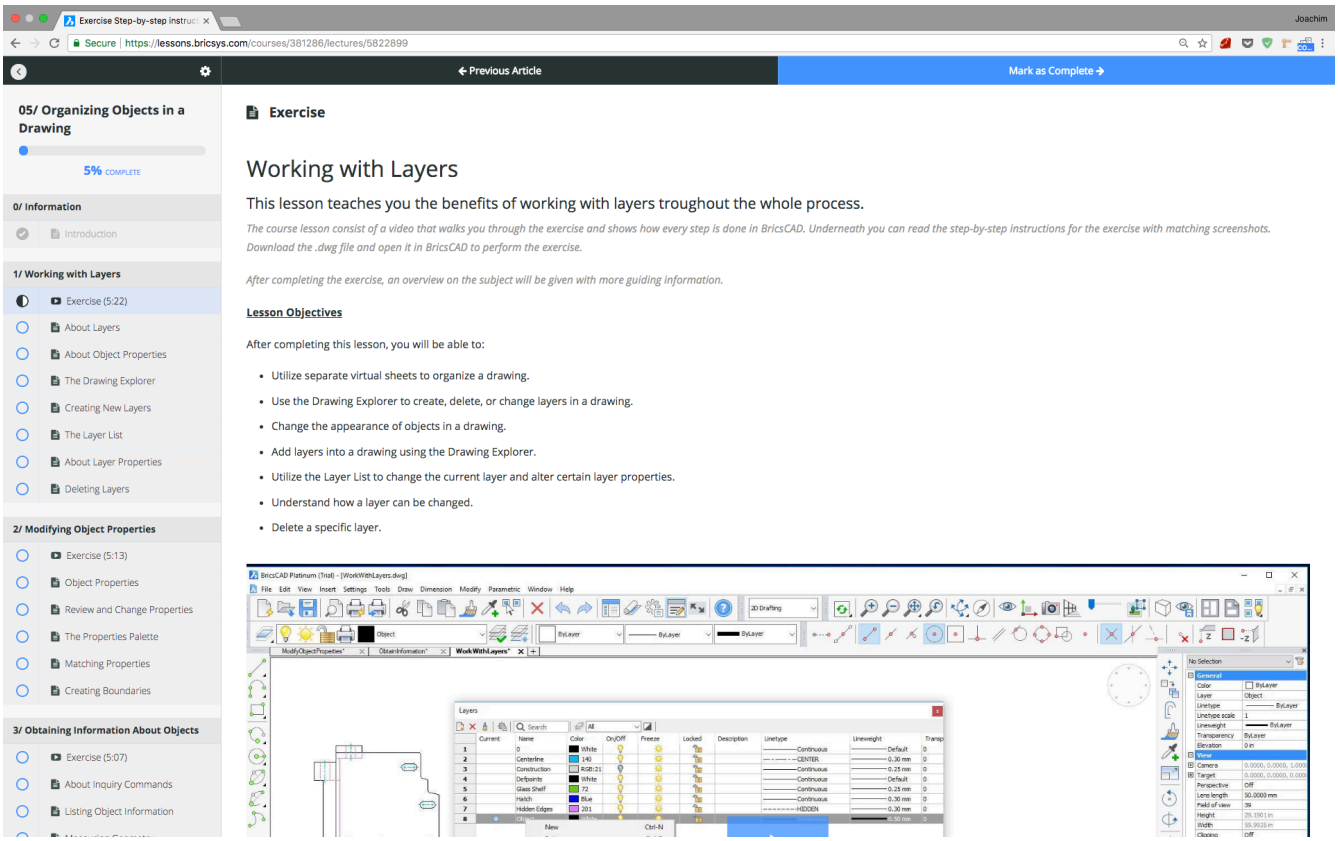
Figure 8.5


Exercise Step-by-step instruc
Joachim
Secure | https://lessons.bricsys.com/courses/381286/lectures/5822899
← Previous Article
Mark as Complete →
05/ Organizing Objects in a Drawing
5% COMPLETE
0/ Information
Introduction
1/ Working with Layers
Exercise (5:22)
About Layers
About Object Properties
The Drawing Explorer
Creating New Layers
The Layer List
About Layer Properties
Deleting Layers
2/ Modifying Object Properties
Exercise (5:13)
Object Properties
Review and Change Properties
The Properties Palette
Matching Properties
Creating Boundaries
3/ Obtaining Information About Objects
Exercise (5:07)
About Inquiry Commands
Listing Object Information
Exercise
Working with Layers
This lesson teaches you the benefits of working with layers troughout the whole process.
The course lesson consist of a video that walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screenshots. Download the .dwg file and open it in BricsCAD to perform the exercise.
After completing the exercise, an overview on the subject will be given with more guiding information.
Lesson Objectives
After completing this lesson, you will be able to:
Utilize separate virtual sheets to organize a drawing.
Use the Drawing Explorer to create, delete, or change layers in a drawing.
Change the appearance of objects in a drawing.
Add layers into a drawing using the Drawing Explorer.
Utilize the Layer List to change the current layer and alter certain layer properties.
Understand how a layer can be changed.
Delete a specific layer.

8.1.11 Lesson 6: Drawing Annotations

Welcome to Learning BricsCAD. A fundamentals level course for the practical use and application of BricsCAD in a design environment. This course is intended for the learner who is new to computer-aided design (CAD). No experience with BricsCAD or any other CAD product is necessary.

8.1.12 Course chapters

After completing this course you will be able to:

1/ Create and Edit Text

- Create a single line of text to add annotation to a drawing.
- Create a paragraph of information to use as annotation in a drawing.
- Change Mtext settings and options using the Text Editor tab.
- Assign word processor type settings to a Mtext.
- Utilize grips and the Properties palette to edit Mtext.

2/ Use Spell Check

- Correct any spelling errors in the drawing using Check Spelling.
- Change and edit dictionaries used by the Check Spelling Command.

3/ Add Dimensions

- Determine the size and location of objects with dimensions.
- Use a dimension style to control the appearance of dimensions in a drawing.
- Use dimensions to quantify objects in a drawing.
- Find the horizontal or vertical distance from one point to another.
- Show the linear distance between two points regardless of relative angle.
- Find the relative angle between two lines.
- Determine the exact length of an arc.
- Apply a radius to an arc or a circle.
- Apply a diameter to an arc or circle.
- Identify the center of an arc or circle using a Center Mark.
- Create a series of dimensions based on an existing dimension.
- Determine the distance between a start point and multiple items following a baseline.
- Create dimensioning in a drawing for multiple objects simultaneously.

4/ Modify Dimensions

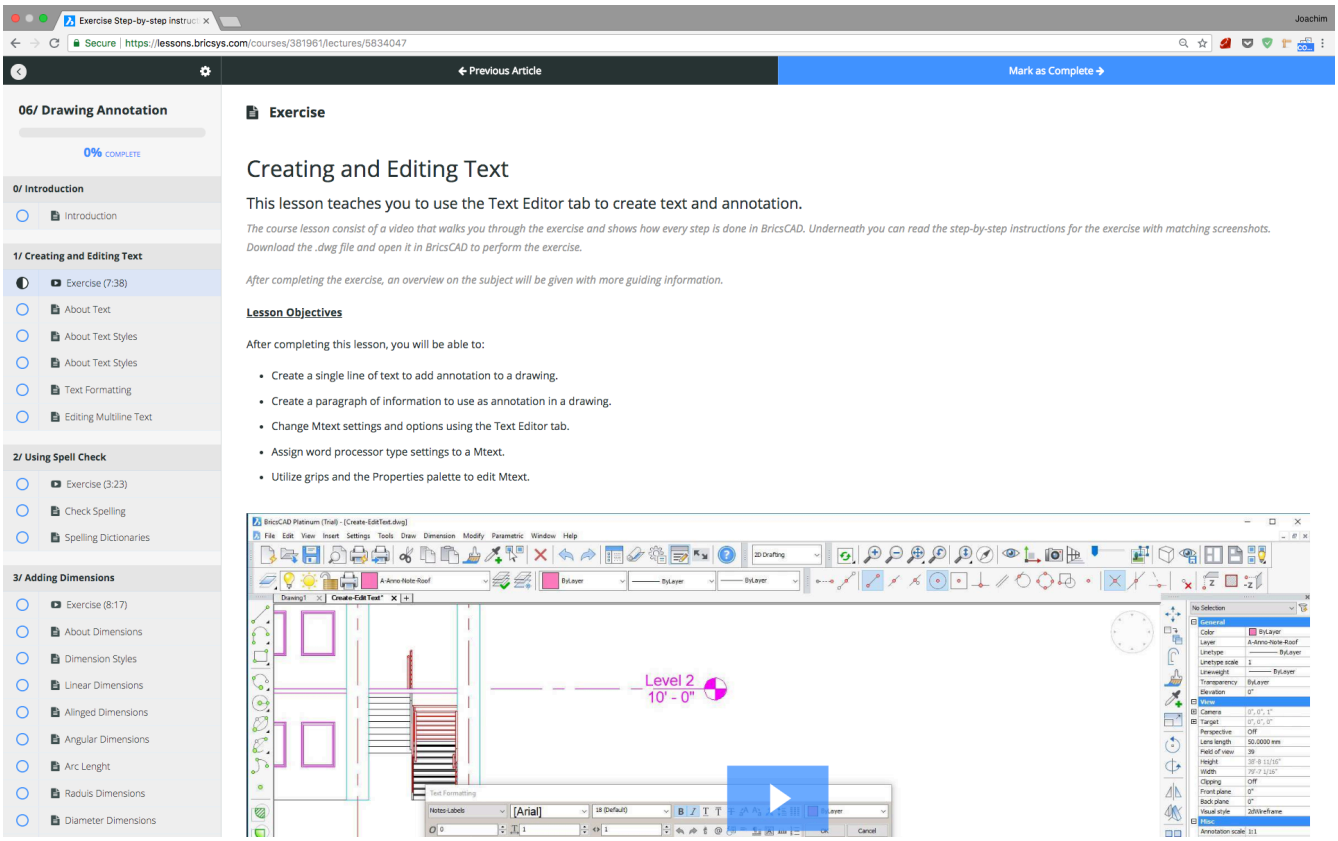
- Use the Properties palette to change the appearance of dimensions.
- Add text information to a text value.
- Add or remove a break in extension or dimension lines.
- Change the spacing between dimension lines.

5/ Add and Modify Multileaders

- Add information to a drawing in the form of multileaders.
- Add and position a mutileader in a drawing.
- Change the appearance of a multileader line.

This is only a preview Get access to the full course materials and step by step exercises at lessons.bricsys.com **The course lesson consists of a videothat walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screen-shots. Download the .dwg file and open it in BricsCAD to perform the exercise. After completing the exercise, an overview on the subject will be given with more guiding information.** (\vref{fig:C:/Dev/Task-2448/images/staticbricsyscom/h

Figure 8.6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← Previous Article
Mark as Complete →
06/ Drawing Annotation
0% COMPLETE
0/ Introduction
Introduction
1/ Creating and Editing Text
Exercise (7:38)
About Text
About Text Styles
About Text Styles
Text Formatting
Editing Multiline Text
2/ Using Spell Check
Exercise (3:23)
Check Spelling
Spelling Dictionaries
3/ Adding Dimensions
Exercise (8:17)
About Dimensions
Dimension Styles
Linear Dimensions
Alinged Dimensions
Angular Dimensions
Arc Lenght
Raduis Dimensions
Diameter Dimensions
Exercise
Creating and Editing Text
This lesson teaches you to use the Text Editor tab to create text and annotation.
The course lesson consist of a video that walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screenshots. Download the .dwg file and open it in BricsCAD to perform the exercise.
After completing the exercise, an overview on the subject will be given with more guiding information.
Lesson Objectives
After completing this lesson, you will be able to:
Create a single line of text to add annotation to a drawing.
Create a paragraph of information to use as annotation in a drawing.
Change Mtext settings and options using the Text Editor tab.
Assign word processor type settings to a Mtext.
Utilize grips and the Properties palette to edit Mtext.
Level 2
10' - 0"

8.1.13 Lesson 7: Reusing Existing Content

Welcome to Learning BricsCAD. A fundamentals level course for the practical use and application of BricsCAD in a design environment. This course is intended for the learner who is new to computer-aided design (CAD). No experience with BricsCAD or any other CAD product is necessary.

8.1.14 Course chapters

After completing this course you will be able to:

1/ Working withBlocks .

- Use a collection of stored objects repeatedly in a drawing.
- Place blocks in a drawing using the Insert command.
- Add text information to a block when it is inserted.
- Change dynamic blocks to fit a drawing.

2/ Reuse content with theDrawing Explorer .

- Import data or drawings into another drawing.

3/ Reuse content withTool Palettes .

- Organize tools and blocks used repeatedly in a Tool Palette.

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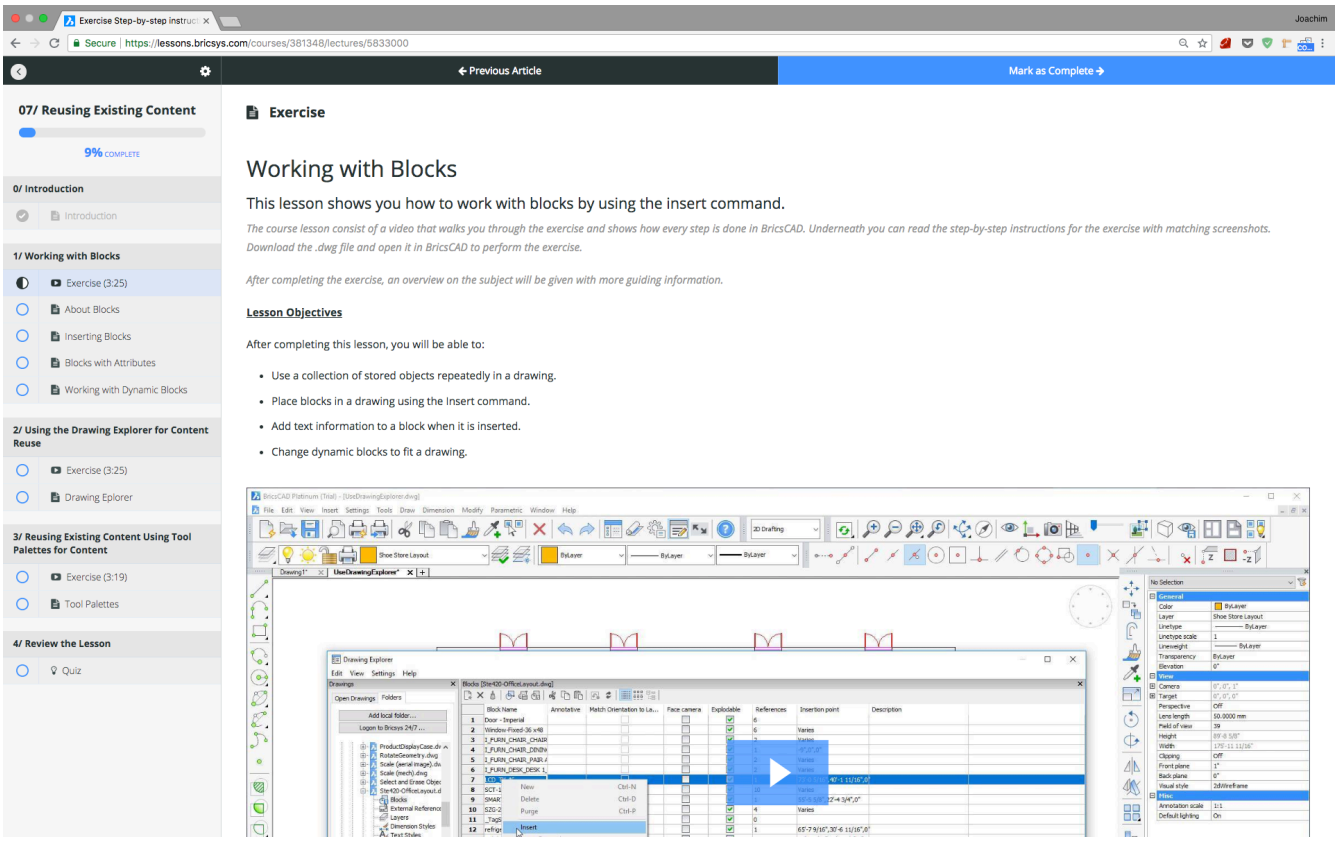
Figure 8.7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Joachim
← Previous Article
Mark as Complete →
07/ Reusing Existing Content
9% COMPLETE
0/ Introduction
Introduction
1/ Working with Blocks
Exercise (3:25)
About Blocks
Inserting Blocks
Blocks with Attributes
Working with Dynamic Blocks
2/ Using the Drawing Explorer for Content Reuse
Exercise (3:25)
Drawing Explorer
3/ Reusing Existing Content Using Tool Palettes for Content
Exercise (3:19)
Tool Palettes
4/ Review the Lesson
Quiz
Exercise
Working with Blocks
This lesson shows you how to work with blocks by using the insert command.
The course lesson consist of a video that walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screenshots. Download the .dwg file and open it in BricsCAD to perform the exercise.
After completing the exercise, an overview on the subject will be given with more guiding information.
Lesson Objectives
After completing this lesson, you will be able to:
Use a collection of stored objects repeatedly in a drawing.
Place blocks in a drawing using the Insert command.
Add text information to a block when it is inserted.
Change dynamic blocks to fit a drawing.

8.1.15 Lesson 8: Supplemental Drawing Techniques

Welcome to Learning BricsCAD. A fundamentals level course for the practical use and application of BricsCAD in a design environment. This course is intended for the learner who is new to computer-aided design (CAD). No experience with BricsCAD or any other CAD product is necessary.

8.1.16 Course chapters

After completing this course you will be able to:

1/ Create and edithatch patterns.

- Add a pattern to a specific area of a drawing.
- Add color fill to closed shapes in a drawing.
- Edit hatch and gradient patterns.

2/ Create and editpolylines.

- Create a single object using multiple line segments.
- Use grips and PEDIT command to edit a polyline.

3/ Createsplines and ellipses.

- Create the smoothest possible curve between two or more points
- Make an ellipse using a major and minor axis.

4/ Work withtables.

- Insert a table filled with information into a drawing.

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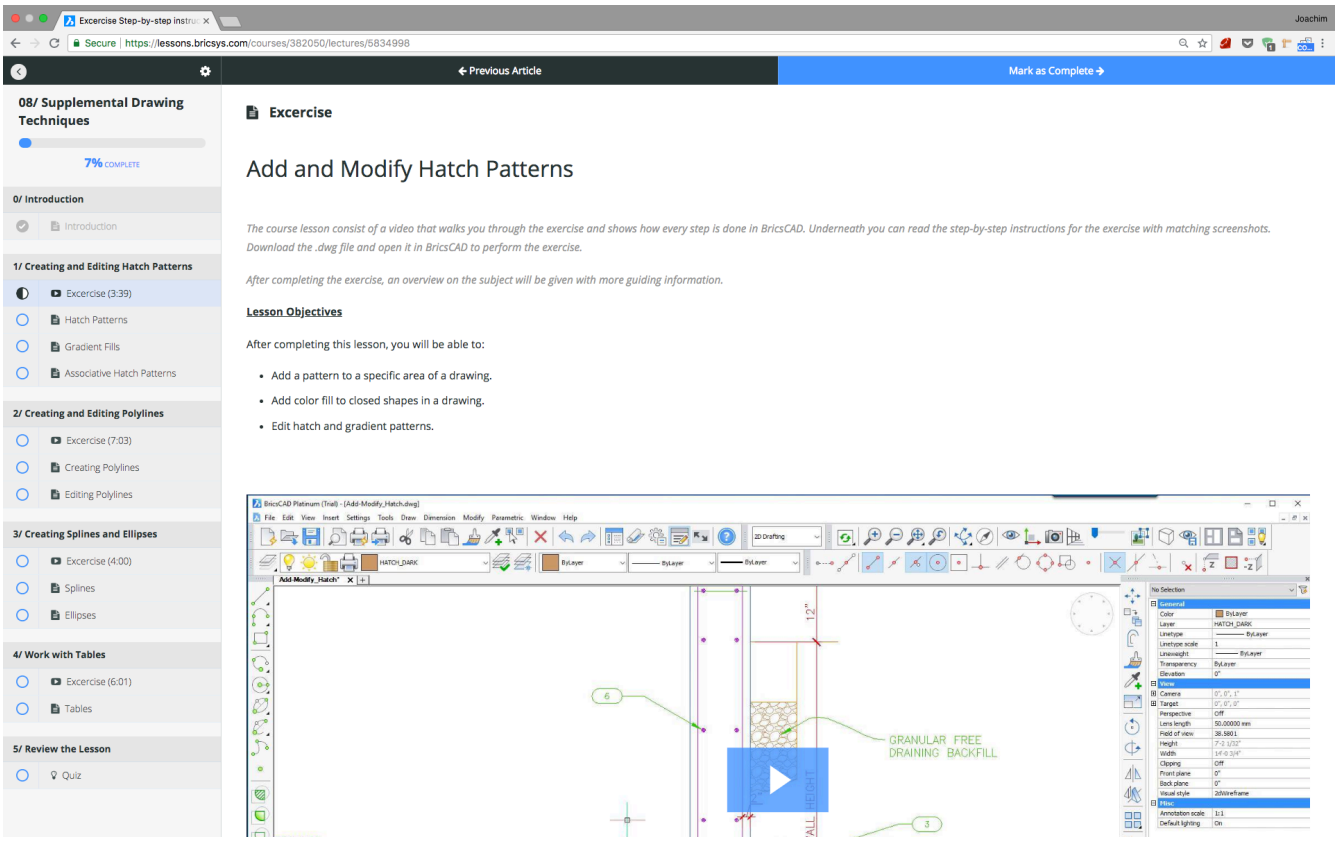
Figure 8.8


Excercise Step-by-step instruc
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← Previous Article
Mark as Complete →
08/ Supplemental Drawing Techniques
7% COMPLETE
0/ Introduction
Introduction
1/ Creating and Editing Hatch Patterns
Excercise (3:39)
Hatch Patterns
Gradient Fills
Associative Hatch Patterns
2/ Creating and Editing Polylines
Excercise (7:03)
Creating Polylines
Editing Polylines
3/ Creating Splines and Ellipses
Excercise (4:00)
Splines
Ellipses
4/ Work with Tables
Excercise (6:01)
Tables
5/ Review the Lesson
Quiz
Excercise
Add and Modify Hatch Patterns
The course lesson consist of a video that walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screenshots. Download the .dwg file and open it in BricsCAD to perform the exercise.
After completing the exercise, an overview on the subject will be given with more guiding information.
Lesson Objectives
After completing this lesson, you will be able to:
Add a pattern to a specific area of a drawing.
Add color fill to closed shapes in a drawing.
Edit hatch and gradient patterns.
GRANULAR FREE DRAINING BACKFILL

8.1.17 Lesson 9: Layouts and Printing

Welcome to Learning BricsCAD. A fundamentals level course for the practical use and application of BricsCAD in a design environment. This course is intended for the learner who is new to computer-aided design (CAD). No experience with BricsCAD or any other CAD product is necessary.

8.1.18 Course chapters

After completing this course you will be able to:

1/ Work withLayouts and Viewports.

- Organize a drawing for printing and electronic output.
- Navigate between different layouts in a drawing.
- Make a frame around specific areas of a drawing for viewing purposes.
- Add more than one viewport to a drawing.
- Define what in the drawing is shown in a viewport.
- Set up a scale for the size indicated in a viewport.
- Resize a viewport to fit information in a drawing.

2/ Create and managePage Setups.

- Utilize a page setup to determine the appearance and format of a final output.
- Use the Page Setup Manager to create, modify, and import page setups for a drawing.

3/ Share yourdesigns.

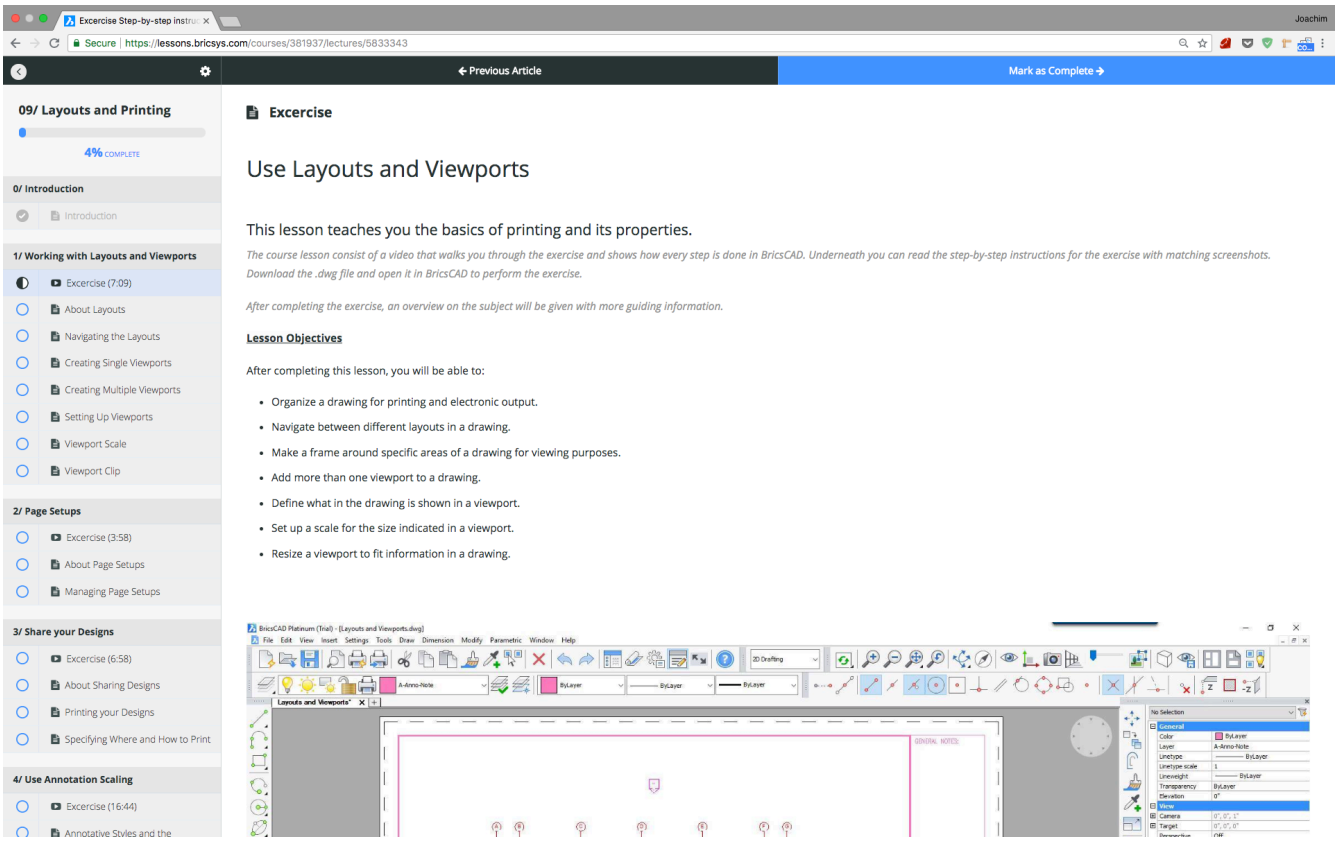
- Share a design with others physically or electronically.
- Print a design using a plotter or printer.
- Specify in the Plot dialogue box where and how the design should be printed.

4/ UseAnnotation Scaling.

- Define what drawing annotation is.
- Use and determine an annotative scale on an object.
- Understand when to use different annotative styles.
- Set an annotative scale in a drawing or a viewport.
- Edit a preexisting annotative scale on an object.
- Create individual scale representations for an object.
- Determine how a line will be affected by an annotative scale.

This is only a preview Get access to the full course materials and step by step exercises at lessons.bricsys.com **The course lesson consists of a videothat walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screen-shots. Download the .dwg file and open it in BricsCAD to perform the exercise. After completing the exercise, an overview on the subject will be given with more guiding information.** (\vref{fig:C:/Dev/Task-2448/images/staticbricsyscom/h

Figure 8.9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← Previous Article
Mark as Complete →
09/ Layouts and Printing
4% COMPLETE
0/ Introduction
Introduction
1/ Working with Layouts and Viewports
Excercise (7:09)
About Layouts
Navigating the Layouts
Creating Single Viewports
Creating Multiple Viewports
Setting Up Viewports
Viewport Scale
Viewport Clip
2/ Page Setups
Excercise (3:58)
About Page Setups
Managing Page Setups
3/ Share your Designs
Excercise (6:58)
About Sharing Designs
Printing your Designs
Specifying Where and How to Print
4/ Use Annotation Scaling
Excercise (16:44)
Annotative Styles and the
Excercise
Use Layouts and Viewports
This lesson teaches you the basics of printing and its properties.
The course lesson consist of a video that walks you through the exercise and shows how every step is done in BricsCAD. Underneath you can read the step-by-step instructions for the exercise with matching screenshots. Download the .dwg file and open it in BricsCAD to perform the exercise.
After completing the exercise, an overview on the subject will be given with more guiding information.
Lesson Objectives
After completing this lesson, you will be able to:
Organize a drawing for printing and electronic output.
Navigate between different layouts in a drawing.
Make a frame around specific areas of a drawing for viewing purposes.
Add more than one viewport to a drawing.
Define what in the drawing is shown in a viewport.
Set up a scale for the size indicated in a viewport.
Resize a viewport to fit information in a drawing.

8.2 Getting started with BricsCAD BIM

8.2.1 Lesson 1: Getting started with BricsCAD BIM

Welcome to Learning BricsCAD BIM. A fundamentals level course for the practical use and application of BricsCAD BIM in a design environment. This course is intended for the learner who is new to building information modeling (BIM). No experience with BricsCAD BIM or any other BIM product is necessary. ****Lesson available in English only****

Course chapters

After completing this course, you will be able to:

1/ Identify key elements of the user interface.

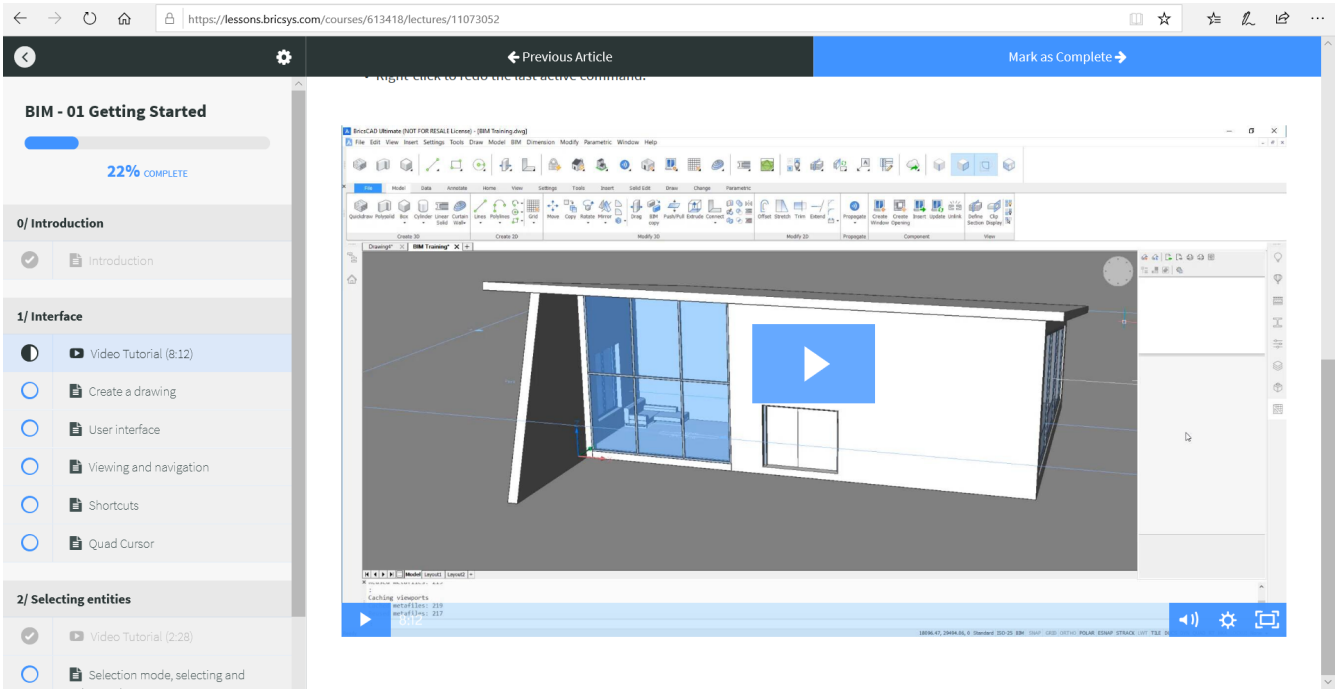
- Open an existing file in BricsCAD BIM.
- Make a new file in BricsCAD BIM using the start-up wizard.
- Make a new file in BricsCAD BIM using a default template drawing.
- Make a new file in BricsCAD BIM using a template drawing of your choice.
- Identify the important elements of the BricsCAD BIM user interface.
- Use the different methods in the BricsCAD BIM user interface to access different tools.
- Hide and isolate entities.
- Reveal all hidden entities.
- Cut through a BIM model horizontally to show the details on a floor plan.
- Toggle the Clip Display property of a BIM section entity.
- Control the visibility of your BIM section cuts.
- Navigate in 3D.
- Make use of a series of shortcuts to help you model faster in BricsCAD BIM.
- Use the Quad (to draw a simple box).
- Right-click to redo the last active command.

2/ Select entities.

- Select whole entities.
- Select the desired sub-entities.
- Select multiple entities at once.
- Select multiple sub-entities at once.

This is only a preview. Get access to the full course materials and step by step exercises at lessons.bricsys.com. **The course lesson consists of a videothat walks you through the exercise and shows how every step is done in BricsCAD BIM. Underneath you can read the step-by-step instructions for the exercise with matching screen-shots. Download the .dwg file and open it in BricsCAD to perform the exercise.** (\vref {fig:C:/Dev/Task-2448/images/staticbric

Figure 8.10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Previous Article
Mark as Complete
BIM - 01 Getting Started
22% COMPLETE
0/ Introduction
Introduction
1/ Interface
Video Tutorial (8:12)
Create a drawing
User interface
Viewing and navigation
Shortcuts
Quad Cursor
2/ Selecting entities
Video Tutorial (2:28)
Selection mode, selecting and

8.2.2 Lesson 2: Basic Modeling

Welcome to Learning BricsCAD BIM. A fundamentals level course for the practical use and application of BricsCAD BIM in a design environment. This course is intended for the learner who is new to building information modeling (BIM). No experience with BricsCAD BIM or any other BIM product is necessary. ****Lesson available in English only****

Course chapters

After completing this course, you will be able to:

1/ Creation and deletion of rooms

- Create a new room.
- Create interior walls.
- Break in part of a room.

2/ Editing existing walls

- Drag a wall to a different position.
- Multiply an existing wall.

3/ Creating walls

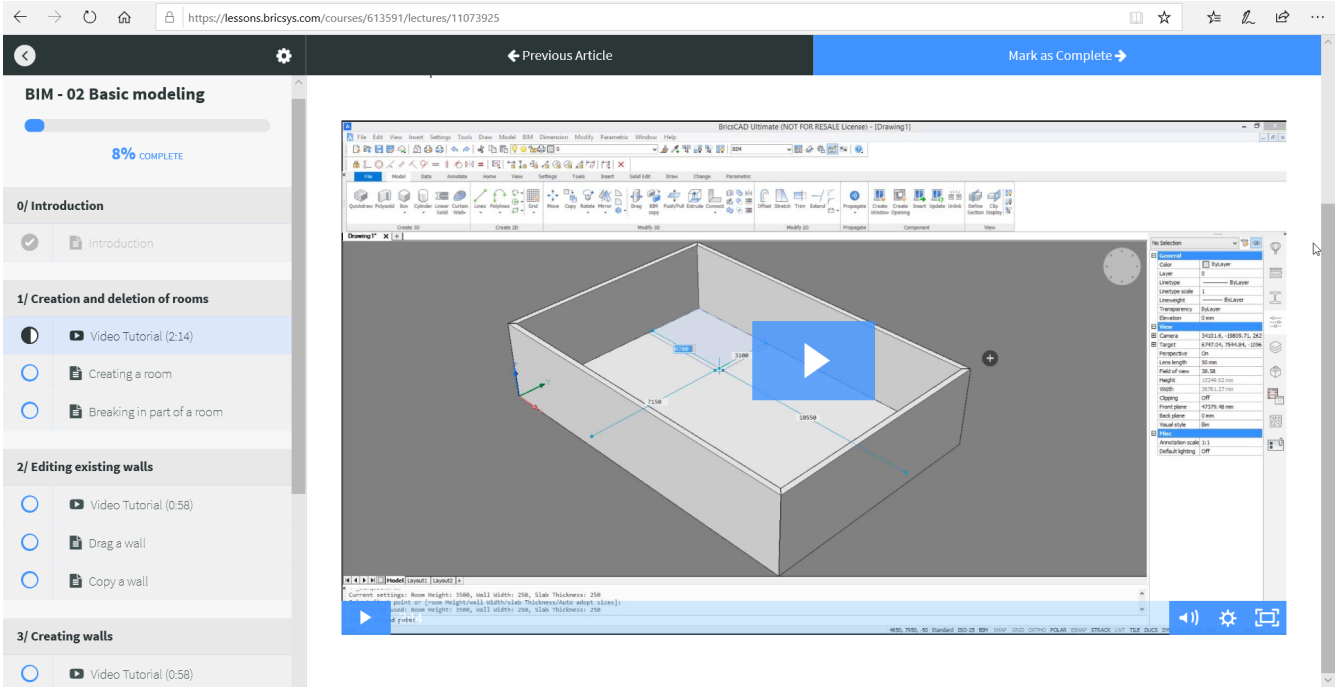
- Create new inner walls.

4/ More editing tools

- Change the height of a wall.
- Connect walls to each other.
- Change the connection type of walls.
- Create a pitched roof.
- Connect walls with the roof.

This is only a preview. This is only a preview. Get access to the full course materials and step by step exercises at lessons.bricsys.com. **The course lesson consists of a video that walks you through the exercise and shows how every step is done in BricsCAD BIM. Underneath you can read the step-by-step instructions for the exercise with matching screenshots. Download the .dwg file and open it in BricsCAD to perform the exercise.** (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/Lessons/GettingStarted/BIMLessons2.png})

Figure 8.11


https://lessons.bricsys.com/courses/613591/lectures/11073925
Previous Article
Mark as Complete
BIM - 02 Basic modeling
8% COMPLETE
0/ Introduction
Introduction
1/ Creation and deletion of rooms
Video Tutorial (2:14)
Creating a room
Breaking in part of a room
2/ Editing existing walls
Video Tutorial (0:58)
Drag a wall
Copy a wall
3/ Creating walls
Video Tutorial (0:58)

8.2.3 Lesson 3: Working with components

Welcome to Learning BricsCAD BIM. A fundamentals level course for the practical use and application of BricsCAD BIM in a design environment. This course is intended for the learner who is new to building information modeling (BIM). No experience with BricsCAD BIM or any other BIM product is necessary. ****Lesson available in English only****

Course chapters

After completing this course, you will be able to:

1/ Create Components

- Subtract a solid from another.
- Use modeling tools to create a window frame and glazing.
- Classify your manually created window as a window.
- Insert your window in other drawings.

2/ Insert Components

- Insert windows in your model.
- Copy windows.
- Create a window using a grid.
- Create a custom window, using a base profile.
- Create a curtain wall.

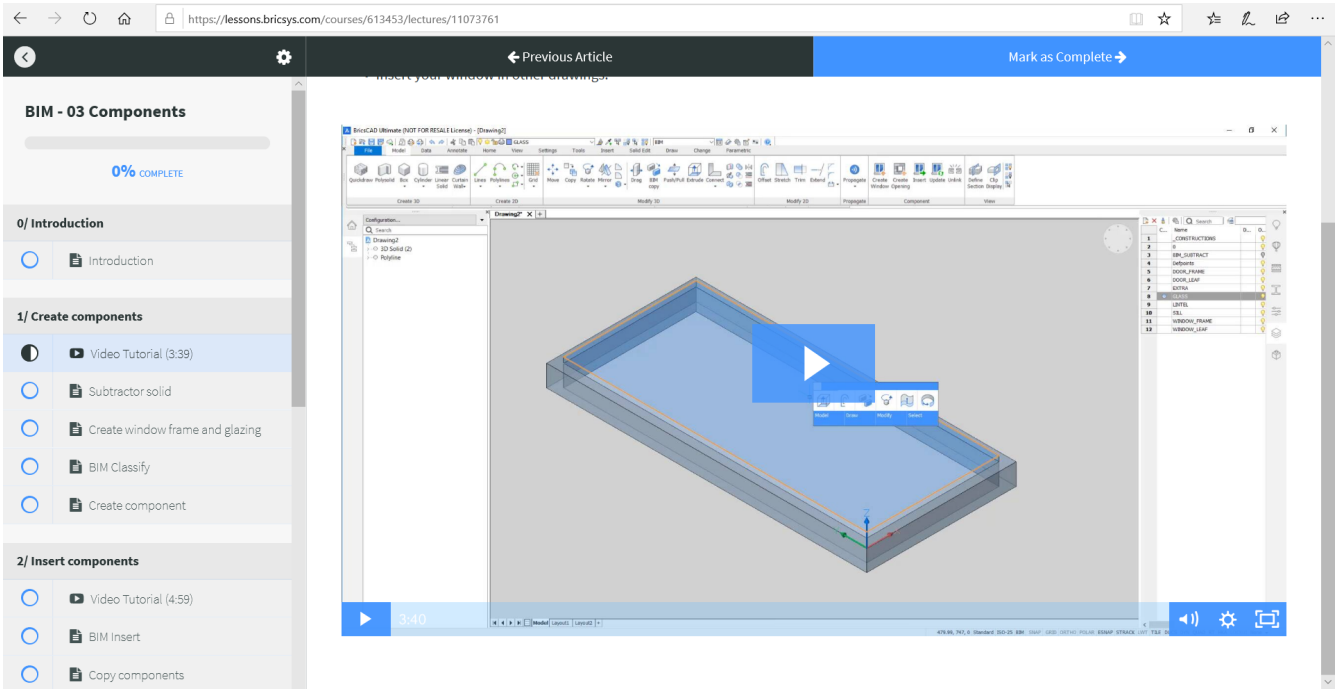
3/ Edit Components

- Flip a door to the other side of a wall.
- Flip a door swing from one side to the other.
- Change the parameters of a component.

4/ Review the lesson

This is only a preview. Get access to the full course materials and step by step exercises at lessons.bricsys.com. **The course lesson consists of a video that walks you through the exercise and shows how every step is done in BricsCAD BIM. Underneath you can read the step-by-step instructions for the exercise with matching screen-shots. Download the .dwg file and open it in BricsCAD to perform the exercise.** (\vref {fig:C:/Dev/Task-2448/images/staticbric

Figure 8.12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Previous Article
Mark as Complete
BIM - 03 Components
0% COMPLETE
0/ Introduction
Introduction
1/ Create components
Video Tutorial (3:39)
Subtractor solid
Create window frame and glazing
BIM Classify
Create component
2/ Insert components
Video Tutorial (4:59)
BIM Insert
Copy components
3:40

8.2.4 Lesson 4: Compositions and structure browser

Welcome to Learning BricsCAD BIM. A fundamentals level course for the practical use and application of BricsCAD BIM in a design environment. This course is intended for the learner who is new to building information modeling (BIM). No experience with BricsCAD BIM or any other BIM product is necessary. ****Lesson available in English only****

Course chapters

After completing this course, you will be able to:

1/ Classifications and Bimify

- Classify building elements manually.
- Classify building elements automatically.
- Rename the buildings and stories in your model.

2/ BIM Compositions

- Attach predefined compositions to your model.
- Create custom compositions.
- Alter composition plies to create correct connections.

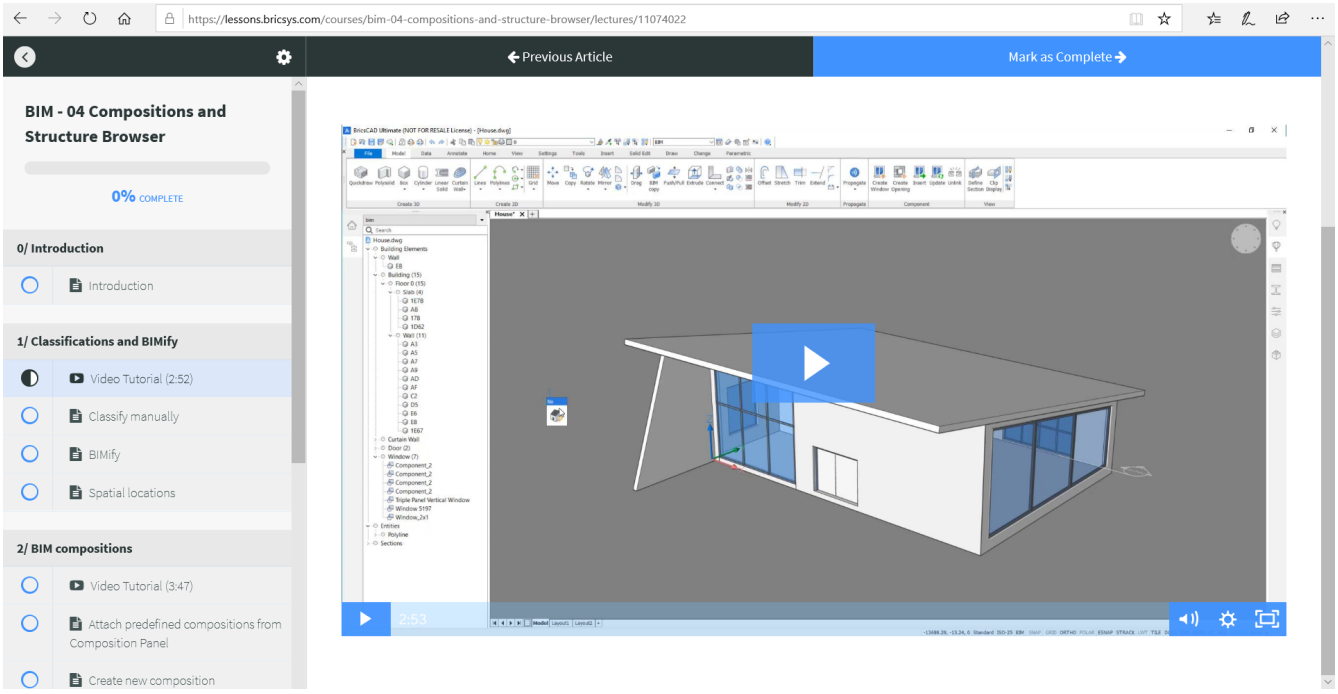
3/ BIM Propagate

- Copy connection details to the same connection types in your model.

4/ Review the lesson

This is only a preview. Get access to the full course materials and step by step exercises at lessons.bricsys.com. **The course lesson consists of a video that walks you through the exercise and shows how every step is done in BricsCAD BIM. Underneath you can read the step-by-step instructions for the exercise with matching screen-shots. Download the .dwg file and open it in BricsCAD to perform the exercise.** (\vref {fig:C:/Dev/Task-2448/images/staticbric

Figure 8.13


https://lessons.bricsys.com/courses/bim-04-compositions-and-structure-browser/lectures/11074022
Previous Article
Mark as Complete
BIM - 04 Compositions and Structure Browser
0% COMPLETE
0/ Introduction
Introduction
1/ Classifications and BIMify
Video Tutorial (2:52)
Classify manually
BIMify
Spatial locations
2/ BIM compositions
Video Tutorial (3:47)
Attach predefined compositions from Composition Panel
Create new composition

8.2.5 Lesson 5: Project Browser and Schedule

Welcome to Learning BricsCAD BIM. A fundamentals level course for the practical use and application of BricsCAD BIM in a design environment. This course is intended for the learner who is new to building information modeling (BIM). No experience with BricsCAD BIM or any other BIM product is necessary. ****Lesson available in English only****

Course chapters

After completing this course, you will be able to:

1/ Project Browser

- Create section drawings of your model.
- Create schedules of your model.

2/ Plans and sections

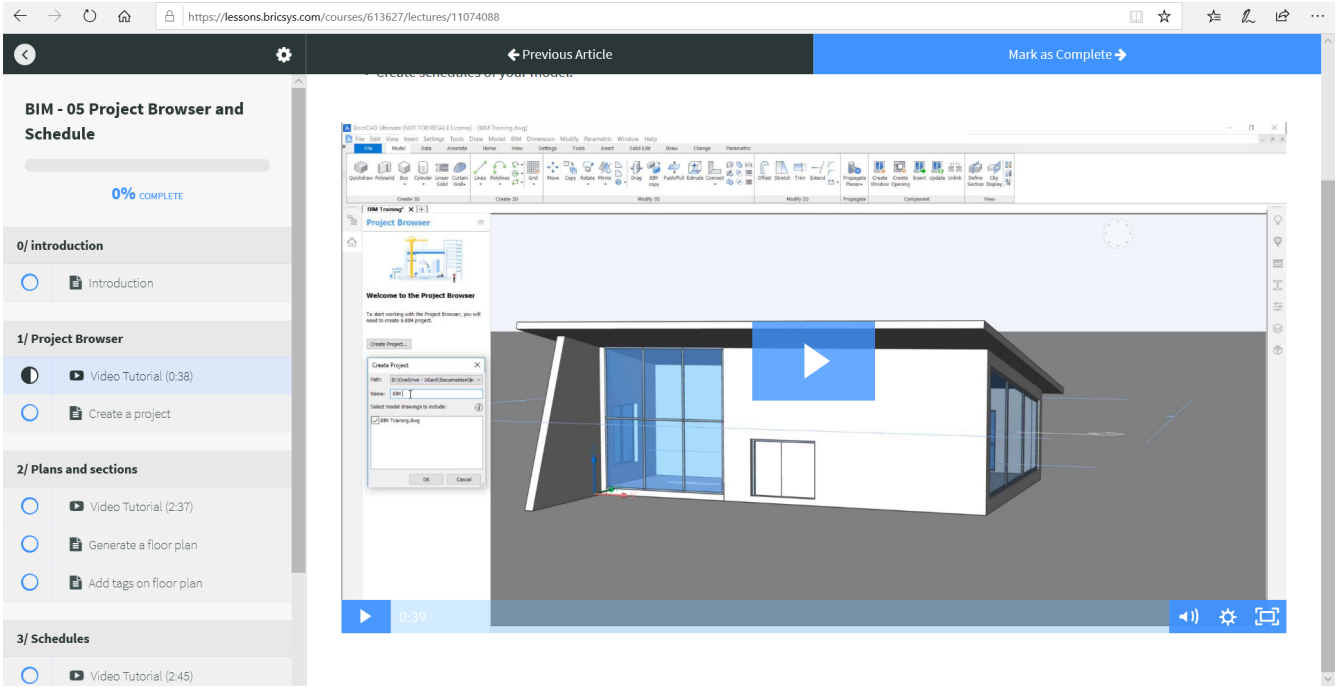
- Generate a floor plan on the paper space.
- Update your floor plan, to match changes made in your model.
- Add information like room area, number and name to your floor plan.

3/ Schedules

- Generate a floor plan on the paper space.
- Update your floor plan, to match changes made in your model.
- Add information like room area, number and name to your floor plan.

This is only a preview. Get access to the full course materials and step by step exercises at lessons.bricsys.com. **The course lesson consists of a video that walks you through the exercise and shows how every step is done in BricsCAD BIM. Underneath you can read the step-by-step instructions for the exercise with matching screen-shots. Download the .dwg file and open it in BricsCAD to perform the exercise.** (\vref {fig:C:/Dev/Task-2448/images/staticbric

Figure 8.14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Previous Article
Mark as Complete
BIM - 05 Project Browser and Schedule
0% COMPLETE
0/ introduction
Introduction
1/ Project Browser
Video Tutorial (0:38)
Create a project
2/ Plans and sections
Video Tutorial (2:37)
Generate a floor plan
Add tags on floor plan
3/ Schedules
Video Tutorial (2:45)
Project Browser
Welcome to the Project Browser
0:39

8.2.6 Lesson 6: Import and Export

Welcome to Learning BricsCAD BIM. A fundamentals level course for the practical use and application of BricsCAD BIM in a design environment. This course is intended for the learner who is new to building information modeling (BIM). No experience with BricsCAD BIM or any other BIM product is necessary. ****Lesson available in English only****

Course chapters

After completing this course, you will be able to:

1/ Importing

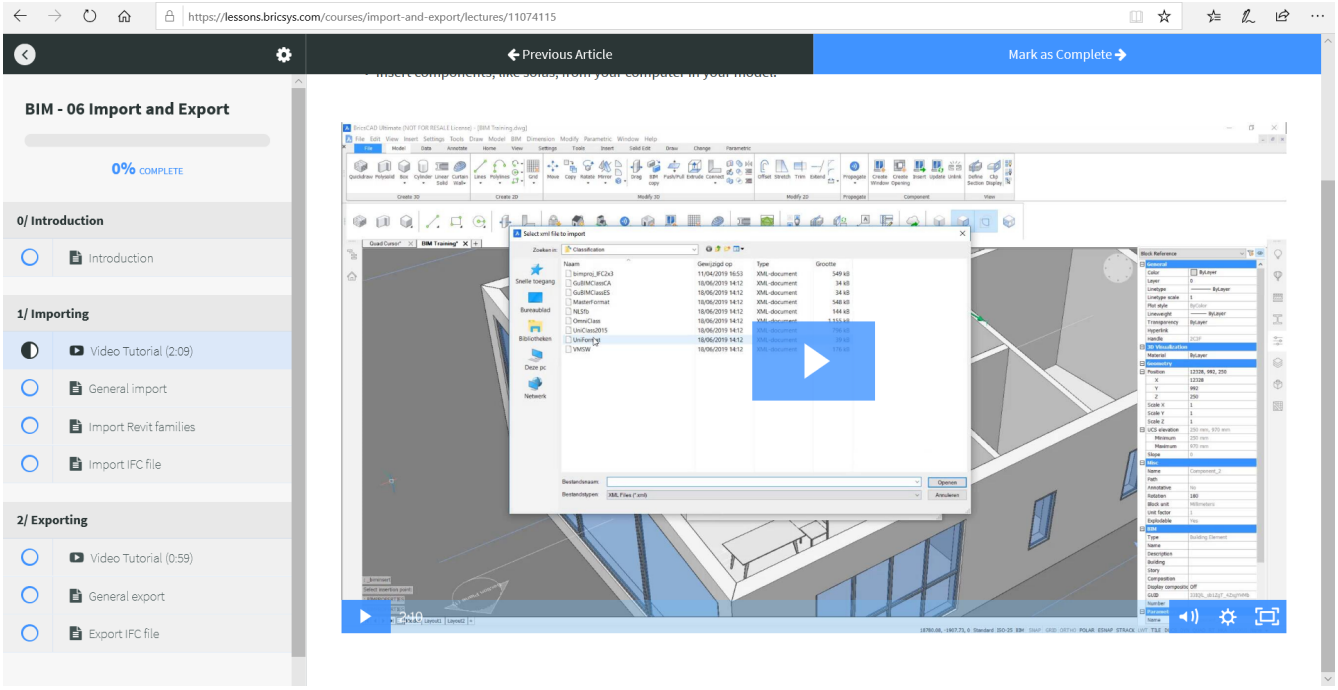
- Import different filetypes into your drawing.
- Insert components, like sofas, from your computer in your model.

2/ Exporting

- Export your drawing as different filetypes.

This is only a preview. Get access to the full course materials and step by step exercises at lessons.bricsys.com. **The course lesson consists of a video that walks you through the exercise and shows how every step is done in BricsCAD BIM. Underneath you can read the step-by-step instructions for the exercise with matching screen-shots. Download the .dwg file and open it in BricsCAD to perform the exercise.** (\vref {fig:C:/Dev/Task-2448/images/staticbric

Figure 8.15


https://lessons.bricsys.com/courses/import-and-export/lectures/11074115
Previous Article
Mark as Complete
BIM - 06 Import and Export
0% COMPLETE
0/ Introduction
Introduction
1/ Importing
Video Tutorial (2:09)
General import
Import Revit families
Import IFC file
2/ Exporting
Video Tutorial (0:59)
General export
Export IFC file

Chapter 9

Developer Tools

9.1 BricsCAD Solution Build (BSB)

9.1.1 What is BSB (BricsCAD Solution Build)?

OEM software is often associated with a stripped- down or “lite” version of the original product. Bricsys follows a different path. We provide our OEMs with full CAD functionality, fully supported by our trusted partner eco-system. **We call our offering BSB (BricsCAD Solution Build) because the word “solution” defines our vision for the way that applications should be delivered.** The strategy is to bring together all of the ecosystem participants - Bricsys distribution partners, Bricsys application partners and Bricsys itself - and go to market as one organization with a unified approach: “The Collective”.

9.2 Developer Reference

9.2.1 Developer Reference

You can customize BricsCAD by adding custom programs written in any of several programming languages that run within BricsCAD, including LISP, C/C++, VB/VBA/VB.NET. BricsCAD offers several API's : COM, BRX, TX, .NET. With BricsCAD, you can run many programs originally created for use with AutoCAD. Specifically, you can use programs written entirely in AutoLISP with no modification. Also .dvb project files (VBA macro files) originally created on AutoCAD should run without modification on BricsCAD. BricsCAD supports the AutoCAD dialog control language (DCL), which is used by other applications such as Lisp/TX/BRX to customize dialog boxes. You can use all DCL files unmodified within BricsCAD. To write applications in C or C++, BricsCAD offers BRX, a C/C++ language interface, code-compatible with the AutoCAD Runtime eXtension (ARX) interface in AutoCAD. BricsCAD includes, a C/C++ language interface compatible with the AutoCAD Development System (ADS) interface in AutoCAD.

Compatible programming languages

For more information about adding custom programs to BricsCAD in one of the compatible programming languages, click the appropriate link below:

- LISP (LISt Processing)
- DCL (Dialog Control Language)
- DIESEL (Direct Interactively Evaluated String Expression Language)
- COM (Component Object Model)
- VBA (Visual Basic for Applications)
- BRX (BricsCAD Runtime eXtension)
- TX (Teigha eXtension)
- .NET

9.3 Application Porting Process

9.3.1 How to port and add my application?

Tired of the exorbitant pricing and strict licensing policies of the major CAD vendors? Expand the market for your applications, become a BricsCAD developer. With the BricsCAD development platform you can port your AutoCAD ® based applications using unmodified source code. And the BricsCAD team will always be there for you, not only to support the porting but also to assist with the marketing of the solution to the worldwide community of BricsCAD users.

- Reach new cost conscious customers worldwide.
- Straightforward port with unmodified source code.
- Active sales & marketing campaigns, free of charge.
- Application Catalog and Online Store.
- Customer Support Infrastructure.

On the Bricsys website, register as developer and contact us to get started!

1. Register as developer
2. Explore the API
3. Port your application to BricsCAD
4. Add your app to the Application store
5. We promote your application

9.3.2 What are Bricsys LGS components?

Bricsys offers two families of components for constraint management: LGS 2D and LGS 3D. They can be embedded into any end-user software. The Bricsys LGS 2D and 3D components are available in binary form for several platforms: 32 and 64 bit Windows, Mac OS X, Linux and other OS. Please contact us to discuss licensing conditions specific to your business. Visit our website for more information...

Using the Bricsys LGS component technology

Using the Bricsys LGS component technology, any company or individual can add support for constraint management into their CAD, CAM or CAE software with minimal development cost and a short time to market. Bricsys offers two families of components for constraint management: LGS 2D and LGS 3D. Geometric, dimensional, and engineering constraints play a fundamental role in modern CAD, CAM and CAE software. They are used to specify and keep interrelations between geometric elements of any 2D/3D CAD model, and thus allow users not only to significantly reduce design time, but also to express their design intent, vastly simplifying future changes in geometry. Please contact us to discuss licensing conditions specific to your business.

9.3.3 Supported API's

With BricsCAD, you can run many programs originally created for use with AutoCAD without modification on BricsCAD. BricsCAD supports the AutoCAD dialog control language (DCL). You can use all DCL files unmodified within BricsCAD. To write applications in C or C++, BricsCAD offers BRX, a C/C++ language interface, code-compatible with the AutoCAD Runtime eXtension (ARX) interface in AutoCAD. **BricsCAD offers several APIs**

- LISP (LISt Processing)
- DCL (Dialog Control Language)
- DIESEL (Direct Interpretively Evaluated String Expression Language)
- COM (Component Object Model)
- VBA Visual Basic for Applications)
- BRX (BricsCAD Runtime eXtension)
- TX (Teigha eXtension)
- .NET

9.4 Application Catalog

9.4.1 How to add your application to our catalog

BricsCAD offers a CAD development platform to which you can port your AutoCAD®-based applications. Porting is easy: you don't even have to modify any source code. And anyway: the BricsCAD team offers full technical support.
https://https://www.youtube.com/embed/YCuRZeKvjDE

Chapter 10

Customer Service and Website

10.1 Website

10.1.1 What is ”My Account” and where can I find it?

On the Bricsys web site, your account can be accessed by clicking on your profile icon in the upper right corner. The page gives an overview of your:

- Personal details
- Licenses and support contracts
- Support requests
- Invoices and payments
- Mailing subscriptions

In case you are a developer, you have also access to:

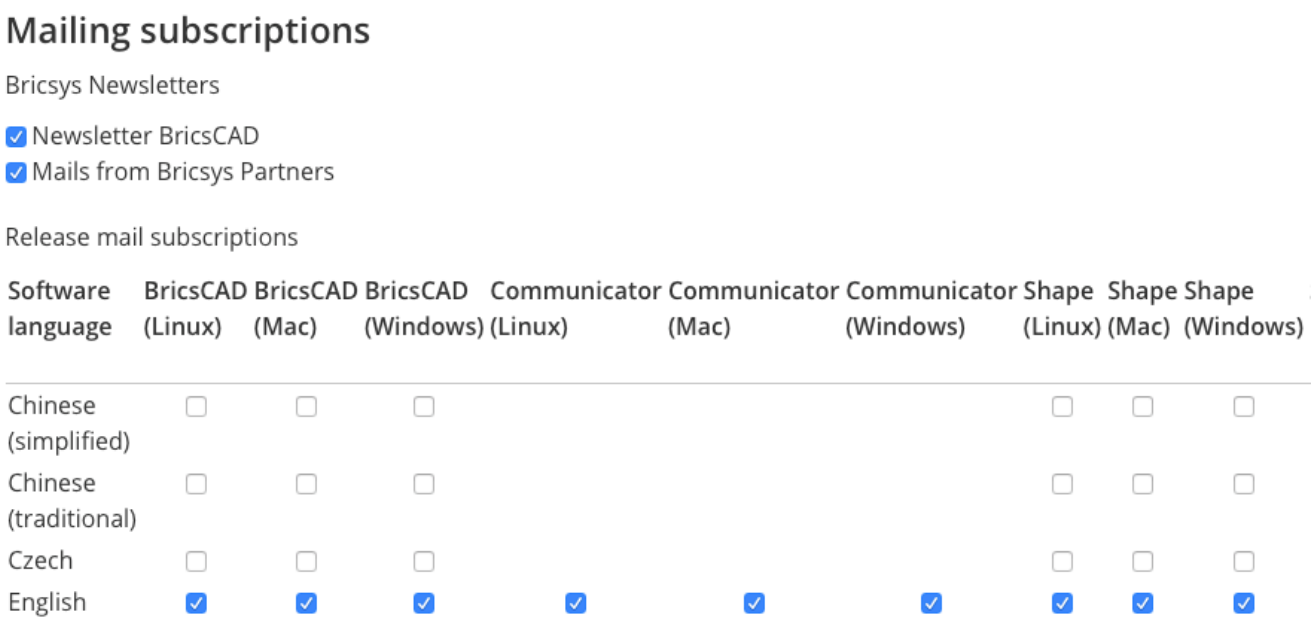
- My applications
- Press room
- Application Support

10.2 Customer Service

10.2.1 Where can I change my mailing subscriptions?

Your mailing subscriptions can be managed under ”My Account” on the Bricsys web site. Your account can be accessed by clicking on your profile icon in the upper right corner of the Bricsys web site. In the menu on the left, click on ”Mailing subscriptions”. There you can check/uncheck the newsletters or release mails you like to receive. (\vref {fig:C:/Dev/Task-2448/images/staticbricsyscom/help/enUS/V19/BCAD/FAQ/Mailingsubscriptions.png})

Figure 10.1


Mailing subscriptions
Bricsys Newsletters
Newsletter BricsCAD
Mails from Bricsys Partners
Release mail subscriptions
Software language
BricsCAD (Linux)
BricsCAD (Mac)
BricsCAD (Windows)
Communicator (Linux)
Communicator (Mac)
Communicator (Windows)
Shape (Linux)
Shape (Mac)
Shape (Windows)
Chinese (simplified)
Chinese (traditional)
Czech
English